|  |
| --- |
| Delhi college of engineering |
| Nonlinear Deformation Theory using FEM |
| for Elastostatic Analysis |
|  |
| **By: NEELANJAN CHAKRABARTY** |
| **University Roll No: 9054** |

|  |
| --- |
| Major project towards the completion of Masters in Engineering (M.E.) in Production Engg. for the academic session of 2009-2011. |

**CANDIDATE’S DECLARATION**

I hereby declare that the work done in this project entitled **“Nonlinear Deformation Theory using FEM, for Elastostatic Analysis”** in the partial fulfillment for the award of degree of “**MASTER OF ENGINEERING”** with specialization in “**PRODUCTION ENGINEERING”** submitted to **Delhi College of Engineering, University of Delhi, as major project** is an authentic record of my own work carried out during the period August 2010 - July 2011 under the supervision of **Dr. Atul Kumar Agrawal,** Associate Professor, Department of Mechanical Engineering, Delhi College of Engineering, University of Delhi. I have not submitted the matter in this dissertation for the award of any other degree or diploma or any other purpose whatsoever.

Dated - July 2011

Neelanjan Chakrabarty

M.E. Production Engg.

University Roll No. 9054

**CERTIFICATE**

This is to certify that major thesis entitled **“Nonlinear Deformation Theory using FEM, for Elastostatic Analysis**” being submitted by **Neelanjan Chakrabarty** in the partial fulfillment for the award of degree of “**MASTER OF ENGINEERING”** with specialization in “**PRODUCTION ENGINEERING”** submitted to **Delhi College of Engineering, University of Delhi,** has been carried out by him under my guidance and supervision and this work has not been submitted elsewhere for a degree.

### Dated - July 2011

### Dr. Atul Kumar Agrawal

Associate Professor

Department of Mechanical Engineering

Delhi College of Engineering, Delhi

**ACKNOWLEDGEMENT**

I express my sincere gratitude to my supervisor, Dr. Atul Kumar Agrawal, for his supervision, support, optimism and continuous encouragement during the course of this project work. Sir has been a constant source of theoretical as well as computational knowledge. He guided me on the latest university level studies being conducted abroad besides providing me with literature being studied on top class universities.

I am grateful to Prof. B.D. Pathak (HOD, Deptt. Of Mechanical Engg, DCE), Prof. S.K Garg (Head, Production Engg.), Prof. V.K. Sethi, Shri Vijay Gautam, Shri R.C. Singh, Shri Girish Kumar, Shri M.S. Niranjan, and all other faculty members for their efforts in providing necessary guidance and support, throughout the duration of ME course at DCE.

My special thanks to Shri Ajay Chhillar, PHD Scholar and Lecturer, DCRUST-Murthal, for helping me in completing the thesis. I would also like to thank all my student friends at DCE for knowledge sharing, co-operation and co-ordination at different stages during my tenure.

Dearest thanks to my mother, my family, and god.

Neelanjan Chakrabarty

New Delhi, July 2011

**CONTENTS**

**Page no.**

**Introduction** 6

**Chapter 1: Overview of FEM based Elastostatics** 7

1.1 Basic Steps in the Finite Element Method Time Independent Problems 7

1.2 Principle of Virtual Displacement (Work) 8

1.3 Lagrangian Finite Elements 11

1.4 Incremental Analysis-Measures 11

1.5 Matrix Based Equations 13

1.6 Error Due to Linearization 14

**Chapter 2: Some Mathematical and Other Supplements** 16

2.1 Eigendecomposition of a matrix 16

2.2 Polar decomposition 17

2.3 Lagrangian and Eulerian descriptions 21

2.4 Strain and Stress measures 23

**Chapter 3: Linear Elasticity Theory** 27

3.1 Continuum Kinematics 27

3.2 Continuum Kinetics 32

3.3 Initial and Boundary Conditions 36

3.4 Hyperelastic Constitutive Laws 38

3.5 Initial Boundary Value Problem of Elastomechanics 44

3.6 Weak Form of the Initial Boundary Value Problem 45

**Chapter 4: Finite Element Method using Spatial Isoparametric Truss Elements**  49

4.1 Fundamental Equations of One-dimensional Continua 49

4.2 Finite Element Discretization 56

**Chapter 5: Nonlinear Elasticity Theory**  64

5.1 Types of mechanical non-linearities 64

5.2 Material non-linearity 65

5.3 Geometrical non-linearity 66

5.4 Consistent Linearization of internal virtual work 73

**Chapter 6: Finite Element Discretization of 1-D Geometrically Non-Linear Continua** 78

6.1 Non-linear continuum-mechanical formulation for Finite truss elements 78

6.2 Truss elements of arbitrary polynomial degree 80

6.3 Linear truss element 82

**Chapter 7: Computational Simulation** 84

**Conclusion** 93

**Bibliography** 94

**Introduction**

Stress-strain analysis forms an integral part of the design of mechanical structures. In the stress-strain graph linear elasticity is followed till the proportionality limit. After that there is an intermediate region till the point where the yield or plasticity begins, this region is referred to as the nonlinear elastic region. The nonlinear strains for this region onwards are considered to be large and can no more be neglected as is the practice in linear elastic region. For the linear region the constitutive equations of stress and strain are well established, but for nonlinear region the development of the constitutive is relatively recent and sparsely known.



**A typical stress-strain curve**

In the following text an attempt is made to explain the nonlinear deformation theory and its departure from the linear analysis. The focus will be on the geometric nonlinearity rather than the material nonlinearity which is a property of the material only. Only elastostatic analysis is considered here, though elastodynamics may be considered as an application of the basic theory. The theory of virtual displacement or simply virtual work is used as the basis of the development of the theory. Suitable examples and figures are provided to better understand the physical significance. The Finite Element Method (FEM) is employed because eventually the algorithm needs to be computationally implemented. The text is a simplified assortment of the various literatures available on the subject. At the end a simulation based on a computational code given by Messer’s Owen and Hinton is also presented based on the theory presented.

The development of the theory is as follows. In the first chapter the whole theory is presented in a concise way to show the complete algorithm. In the second chapter some mathematical and other required formulations are given. In the third chapter the linear elastic analysis is presented in detail. In the fourth chapter finite element discretization for the linear case is developed giving the example of the one dimensional truss. In the fifth chapter the basis for the nonlinear analysis is given, again in detail. In the sixth chapter the finite element discretization for the non-linear case is given, again with an example for the one dimensional truss. In the seventh chapter the simulation of the numerical method employed using computational code is given.

**Chapter 1: Overview of FEM based Elastostatics**

Finite Element methods allow us to model solution schemes for complex geometries with relative ease. It employs discretization of body into small elements so that response can be studied as a sum of effects due to each element. In linear analysis we study small changes in the finite element assemblage where material is linearly elastic, displacement response is a linear function of load change. This is not case in non-linear analysis as we will see during the following development. Here only the case of small strain elastic deformation (usually for metals with yield stress being < 0.05% to 0.1% of material’s elastic modulus) for static, time independent model is considered.

1.1 Basic Steps in the Finite Element Method Time Independent Problems

- Domain Discretization

- Select Element Type (Shape and Approximation)

- Derive Element Equations (Variational and Energy Methods)

- Assemble Element Equations to Form Global System

**[K]{U} = {F}**

[K] = Stiffness or Property Matrix  
 {U} = Nodal Displacement Vector

{F} = Nodal Force Vector

- Incorporate Boundary and Initial Conditions

 - Solve Assembled System of Equations for Unknown Nodal   
 Displacements and Secondary Unknowns of Stress and Strain Values

*Simple Element Equation Example Direct Stiffness Derivation*

Equilibrium at Node 1 *F1 = ku1 – ku2*

Equilibrium at Node 1 *F2 = - ku1 + ku2*

Or in matrix form =

or [*K*]{*u*} = {*F*}

1.2 Principle of Virtual Displacement (Virtual Work)

In this in we assume infinitesimal changes of coordinates while time is held constant. It is called virtual rather than real since no actual displacement can take place without the passage of time. Based on the concepts of virtual displacements we get a relation between internal and external virtual work and to minimization of system potential energy for equilibrium.

*Example: One-Dimensional Bar Element*

*A = Cross-sectional Area*

*E = Elastic Modulus*  *x*

*f(x) = Distributed Loading*





**(*i*)** *L* **(*j*)**

Virtual Strain Energy = Virtual Work Done by Surface and Body Forces



For One-Dimensional Case



Matrix Equations for Bar



Linear Approximation Scheme



k(*x*) – Lagrange Interpolation Functions

Element Equation Linear Approximation Scheme, Constant Properties



Linear Elastic Analysis

When we consider linear analysis we assume a small deformation which implies that the work is formulated over the original configuration. i.e.

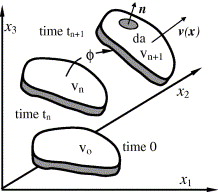
**= + +**

(internal virtual work) = (external virtual work = R)

Over bar terms denote the virtual components where are the virtual displacements and are the corresponding virtual strains. The superscript *T* gives the transpose. gives the externally surface tractions over a small area, and is the concentrated load. The complete linear elastic analysis along with the meaning of virtual components is further elaborated in chapters 3 and 4.

Nonlinear Elastic Analysis

In nonlinear analysis the aim is to evaluate the equilibrium positions of the complete body at discrete time points 0, ∆t, 2∆t, 3∆t …….. .



**Fig.1.1 Element showing incremental motion**

In this analysis we follow all particles of the body in motion, i.e. Lagrangian formulation.

In Lagrangian incremental analysis approach we express the equilibrium of the body at time t+∆t using principle of virtual displacements. Using tensor notations this requires that

**=**

(internal virtual work) = (external virtual work)

where, = cartesian component of the Cauchy stress tensor (forces/area, in *deformed geometry*);

**=**

is the strain displacement corresponding to virtual displacement.

= components of virtual displacement vector imposed on configuration at time t+∆t, a function of, *j* = 1, 2, 3.

= cartesian coordinates of material points at t+∆t,

= volume at time t+∆t; and

**= +**

where,

= component of externally applied forces per unit volume at time t+∆t,

= component of externally applied surface tractions per unit surface area at time t+∆t,

= surface at time t+∆t on which external tractions are applied,

= evaluated on surface (since is zero initially on the real deformed surface at the time t+∆t).

The difficulty here being that configuration of the body at time t+∆t is unknown. This is an important difference compared to linear analysis in which it is assumed that the deformations are infinitesimally small, so that original configuration is used. It is important to recognize that the virtual work principle stated here is an application of the one used for linear analysis applied to the body considered in configuration at time t+∆t.

1.3 Lagrangian Finite Elements

Two types of approaches are usually considered when formulating Lagrangian finite elements:

(explained further in section 2.3)

* Total Lagrangian(TL):
  + The stress and strain measures, derivatives and integrals are computed with respect to the original configuration at time 0 (see fig.1.1).
* Updated Lagrangian(UL):
  + The stress and strain measures, derivatives and integrals are computed with respect to the current configuration time t.

1.4 Incremental Analysis-Measures

We now define two new stress and strain measures (further elaborated in section 2.4):

1. *Green Lagrange strain (E) or simply Green strain*

***E =* []**

***=* []**

***=* [*u* + *u* + *u* . *u*]**

* *Where C is the Cauchy-Green deformation tensor (explained in chapter2), F is the deformation gradient, used to describe the stretches, rotations, change in angle between adjacent material fibres from time 0 to t.*

***F* =**

1. *2nd Piola Kirchhoff stress(S)*

where**,**

= Cauchy stress (forces/area, in deformed geometry), =Jacobian relation.

* The purpose of introducing these new measures is that the components of their respective tensors have the property of not changing when the material is subjected to rigid body translation and undergoing rigid body rotation. So we can compute without worrying about the changing orientation of the element.
* *S* is the work conjugate of *E*.

Incremental Analysis-Equations

Now *linearizing* the equation of virtual work,

**=**

where, the measures are with respect to time 0 and t for TL and UL respectively, we finally get,

* **For TL:**

**+ = -**

* **For UL:**

**+ = -**

where, and are incremental stress-strain material property tensors (= ), , are the 2nd Piola-Kirchhoff stress and Cauchy stress at time t; ,and, are the linear and nonlinear incremental strains, at configuration at time 0, t respectively.

*The above two incremental lagrangian relations are employed to calculate,*

- An increment in the displacements,

- Which then is used to evaluate approximations to the displacements, strains, and stresses corresponding to time t+∆t,

- The displacement approximations corresponding to time t+∆t are obtained simply by adding the calculated increments to the displacements at time t,

- And the strain approximations are evaluated from the displacements using the Green-Lagrange relation.

Non-linear analysis including the concept of linearization is elaborated in chapters 5 and 6.

1.5 Matrix Based Equations

The basic approach in an incremental step by step solution is to assume that the solution for the discrete time t is known and that the solution for the discrete time t+∆t is required.

So,

***t+∆tR - t+∆tF=0***

Where,

*t+∆tR* = vector of externally applied nodal point loads at time t+∆t,

*t+∆tF* = vector of nodal point forces equivalent to element stresses at time t+∆t.

(Assume t+∆tR is independent of the deformations).

Since solution at time t is known we write

***t+∆tF = tF + F***

Where F is the increment in nodal point forces corresponding to increment in stresses and displacements of element from t to t+∆t. This vector (F) can be approximated using a *tangent stiffness matrix ‘ tK ’* (explained in chapter 6) which corresponds to geometric and material conditions at time t,

***F =  tKU***

where U is a vector of nodal point displacements and *tK = dtF/dtU*.

Hence the tangent stiffness matrix corresponds to the derivative of the internal element nodal point forces *tF* with respect to the nodal point *tU*

or, ***tKU = t+∆tR - tF.***

Solving for *U*, we can evaluate an approximation to the displacement at time t+∆t,

***t+∆tU =  tU + U***

The exact displacement at time t+∆t is those that correspond to the applied loads *t+∆tR*.

*Displacement based Incremental equations for static analysis:*

***For TL: (t0KL + t0KNL)U = t+∆tR - t0F***

***For UL: (ttKL + ttKNL)U = t+∆tR - ttF***

Where,

***t0KL , ttKL*** = linear strain incremental stiffness matrix,

***t0KNL , ttKNL*** = nonlinear strain incremental stiffness matrix.

1.6 Error Due to Linearization

Assuming that the appropriate incremental measures have been obtained, we can now check the difference between the calculated internal virtual work for time t+∆t and the given external virtual work, already at time t+∆t. Denoting in bracketed superscript for the iteration necessary,

* **In TL:**

**-**  **or *t+∆tR - t+∆t0F*(i)**

* **In UL:**

**-**  **or *t+∆tR - t+∆ttF*(i)**

This ‘out of balance’ virtual work is reduced by iteration until the difference is within suggested convergence measures to get the exact displacement, strain and stress, i.e.

***t+∆tK(i-1) ∆U = t+∆tR - t+∆tF(i-1)***

*and then*

***t +∆tU(i) =  t +∆tU(i) + ∆U(i)***

Here we have assumed deformation-independent loading which can be specified prior to incremental analysis, e.g. concentrated loading.

*The elastic material models are elastic, hyperelastic, hypoelastic. In order for a formulation to be effective for a certain response prediction, it is imperative that both the kinematic and constitutive descriptions are appropriate, for e.g., TL formulation is employed to describe the kinematic behaviour, and material law is used which is formulated for small strain conditions. In this case analysis can model only small strains although TL kinematic formulation does admit large strains.*

**Chapter 2: Some Mathematical and Other Supplements**

2.1 Eigendecomposition of a matrix

* Eigendecomposition or spectral decomposition is the factorization of a matrix into a canonical form
* i.e. Λ = U\*AU for some unitary matrix U (U\*U=I) and Λ is the diagonal matrix,
* Where the matrix is represented in terms of its eigenvalues and eigenvectors.
* Let A be a square (N×N) matrix with N linearly independent eigenvectors,
* Then A can be factorized as A=QΛQ-1 , where Q is the square (N×N) matrix whose ith column is the eigenvector qi of A and Λ is the diagonal matrix whose diagonal elements are the corresponding eigenvalue, i.e., Λii = λi.
* Note that only diagonalizable matrices can be factorized in this way.
* If we are given the components of the strain tensor in an arbitrary orthonormal coordinate system, we can find the principal strains using eigenvalue decomposition.
* This system of equations is equivalent to finding the vector along which the strain tensor becomes a pure stretch with no shear component.
* These are useful when we represent deformation gradient *F* as a product of orthogonal matrix, *R* (*rotation*) and symmetric matrix, *U* (*stretch*).

*Example-diagonalizable matrix*

* Consider a matrix

***A*** =

* This matrix has eigenvalues

**= 3, = 2, =1**

* The corresponding eigenvectors of *A* are

**= , = , =**

* Now, let *P* be the matrix with these eigenvectors as its columns:

***P* =**

* Then *P* diagonalizes *A*, as a simple computation confirms:

***P-1AP* = =**

2.2 Polar decomposition

* The Polar decomposition theorem states that any second order tensor whose determinant is positive can be decomposed uniquely into a symmetric part (*Pure Stretch,U or V)* and an orthogonal part (*Rotation,R)*.
* In continuum mechanics, the deformation gradient is such a tensor because *det(f)>*0.
* Therefore we can write *F=R.U=V.R*
* where *R, the rotation* is an orthogonal tensor and *U,V* are the symmetric tensors and are called the *right stretch tensor* and the *left stretch tensor*, respectively. This decomposition is called the polar decomposition of *F*.
* The right Cauchy-Green deformation tensor is defined as *C=FT.F*, which is a *symmetric tensor*.
* From the polar decomposition of F we have

***C*=*UT.RT.R.U*=*U.U*=*U*2**

* If you know *C* then you can calculate *U* and hence *R* using *R=F.U-1*.

*How to find the square root of a tensor?*

* If you want to find U given C you will need to take the square root of C. How does one do that?
* We use is the *spectral decomposition* or *eigenprojection* of *C*.
* The spectral decomposition involves expressing *C* in terms of its eigenvalues and eigenvectors. The tensor product of the eigenvectors acts as a basis while the eigenvalues give the magnitude of the projection.
* Thus,

***C* =**

* where *λ****i****2* are the principal values (eigenvalues) of *C* and *Ni* are the principal directions (eigenvectors) of *C*.
* Therefore,

**=**

* Since the basis does not change, we then have

***U* =**

* Therefore the *λ****i*** can be interpreted as principal stretches and the vectors, *Ni* are the directions of the principal stretches.

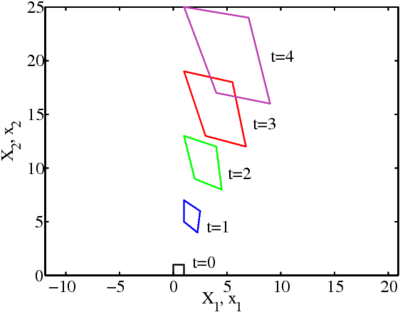
*Example*

* Let us assume that the motion is given by

***x1 =*  [4*X*1 + (9 - 3*X*1 - 5*X*2 - *X*1*X*2)*t*]**

***x2 = X*2 + (4 + 2*X*1)*t***

* The figure shows how a unit square subjected to this motion evolves over time.



Solution:

*(1) Deformation**gradient*

* The deformation gradient is given by

***F =*  =**

* Therefore

**= = [(4 + (-3 – *X*2)*t*]**

**= = [(-5 - *X*1)*t*]**

**= = 2*t***

**= = 1**

* At *t* = 1 at the position we have

***F* = =**

* We can calculate the deformation gradient at other points in a similar manner.

*(2) Right**Cauchy-Green**deformation**tensor*

* We have

* Therefore,

***C* = *FTF* =**

* To compute we have to find the eigenvalues and eigenvectors of C. The eigenvalue problem is

**= 0**

**det = 0**

**-**   **+**   **= 0**

* This equation has two solutions

**= = 5.159**

**= = 1.466**

* Taking the square roots we get the values of the principal stretches

**= 2.2714 = 1.2107**

* To compute the eigenvectors we plug the eigenvalues into the eigenvalue problem to get

**=**

* this system of equations is not linearly independent, we need another equation to solve this system of equations for and .This problem is eliminated by using the following equation (which implies that N is a unit vector)

**=**

* Solving, we get

**= =**

* We can do the same thing for the other eigenvector N2 to get

**= =**

* Therefore,

**= = =**

and

**= = =**

* Therefore,

**C = + C = 5.159+1.466**

* We usually don't see any problem to calculate ***C*** at this point and go straight to the right stretch tensor.

*(3) Right**stretch tensor*

* The right stretch tensor is given by

**U= + U=2.2714+1.2107**

or,

**U =**

* We can invert this matrix to get

**U-1 =**

*(4) Rotation*

* We can now find the rotation matrix by using the relation

***R* = *F. U-1***

* In matrix form,

**R = =**

* We check whether this matrix is orthogonal by seeing whether

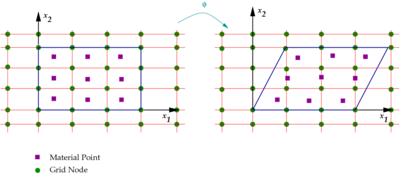
***RRT* = *RTR* = *I***

*We thus get the polar decomposition of* ***F***.

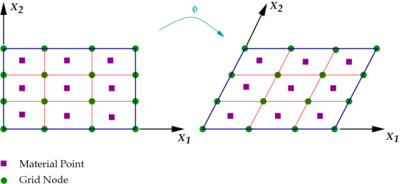
* In an actual calculation you have to be careful about floating point errors. Otherwise you might not get a matrix that is orthogonal.

2.3 Lagrangian and Eulerian descriptions

* *Spatial or Eulerian coordinates(x)***:**
* These coordinates are used to locate a point in space with respect to a fixed basis. We can think of these coordinates as the ones we are familiar with.
* The Eulerian mesh is a background mesh. The body flows through the mesh as it deforms. The nodes remain fixed and the materials points move through the mesh. The position of a material point relative to the nodes varies with the motion.



* Eulerian coordinates of nodes are fixed and coincide with spatial points. Spatial coordinates of material points vary with time.
* Material flows through the mesh.
* The material point at a given element quadrature point changes with time. This makes dealing with history-dependent materials difficult.
* Boundary nodes and the material boundary may not coincide. Therefore, boundary conditions and interface conditions are hard to apply.
* There is no mesh distortion because the mesh is fixed in space. However, the domain that needs to be modeled is larger because we do not want the body to leave the domain.
* *Material or Lagrangian coordinates(X)***:**
* These coordinates are used to label material points. If we sit on a material point, the label does not change with time. We do start with a reference label which we usually choose as the initial spatial coordinates of a material point.
* We can think of the Lagrangian mesh as being drawn on the body. The mesh deforms with the body. Both the nodes and the material points change position as the body deforms. However, the position of the material points relative to the nodes remains fixed.



* Lagrangian coordinates of nodes move with the material. Material coordinates of material points are time invariant.
* No material passes between elements.
* Element quadrature points remain coincident with material points.
* Boundary nodes remain on the boundary. Therefore, boundary conditions and interface conditions are easily applied.
* Severe mesh distortion can occur because the mesh deforms with the material
* *Two types of approaches are usually taken when formulating Lagrangian finite elements*:
* *Total Lagrangian:* 
  + The stress and strain measures are Lagrangian, i.e., they are defined with respect to the original configuration.
  + Derivatives and integrals are computed with respect to the Lagrangian (or material) coordinates.
* *Updated Lagrangian:* 
  + The stress and strain measures are Eulerian, i.e., they are defined with respect to the current configuration.
  + Derivatives and integrals are computed with respect to the Eulerian (or spatial) coordinates.

2.4 Strain and Stress measures

* A basis for this description is the consideration of a body as an ensemble of material points as well as the characterization of their initial and current position by means of the position and displacement vectors. By considering the immediate vicinity of material points one finally gets to the concept of strains, which describe the deformation of a material body.
* For the description of non-linear kinematics the material or Lagrangian approach will be used, according to which the state of a point is defined as a function of its initial position and time.

Physical Interpretation of the Green Strain (E)

The physical interpretation of the *Green Strain* (*E*) is demonstrated by means of the following example.

*Example*

Consider the uniform deformation of a square blockof side length 2 units, initially centered at **X** = (0*,* 0), as shown in Figure. The deformation is defined by the mapping

***=* (18 + 4*X*1 *+* 6 *X*2) + (14 +6 *X*2) + *X*3**



Fig. Undeformed and Deformed configuration of a rectangular block

Solution:

*The matrix form of the deformation gradient tensor*

***F* = =**

*The right Cauchy–Green deformation tensor*

***C* = *FTF***

***C =***

*The Green strain tensor*

***E =* []**

***E =***

Second Piola–Kirchhoff Stress Tensor (S) and its physical significance

The *second Piola–Kirchhoff stress tensor* **S**, which is used in the study of large deformation analysis, is introduced as the stress tensor associated with the force *dF* in the undeformed elemental area *dA*that corresponds to the the force *d*fon the deformed elemental area *d*a.

***dF = S. dA***

Thus, the second Piola–Kirchhoff stress tensor gives the *transformed current force* per unit *undeformed area*.

The second Piola–Kirchhoff stress tensor is related to the Cauchy stress tensor (also denoted by ) according to the equation

Clearly, *S*is symmetric whenever is symmetric.

We introduce a psuedo stress vector associated with the second Piola–Kirchhoff stress tensor by

***dF = dA = S. dA = S. dA***

where is the unit normal vector to the undeformed cross-sectional area. The following example demonstrates the physical significance of the second Piola–Kirchhoff (*S*).

*Example*

Consider a bar of cross-sectional area *A* and length *L*. The initial configuration of the bar is such that its longitudinal axis is along the *X*1 axis. If the bar is subjected to uniaxial tensile stress and deformation that stretches the bar by an amount and rotates it, without bending, by an angle , as shown in Figure, the deformation mapping is given by

**=**

where and are constants; denotes the stretch of the bar and denotes the volume change from undeformed configuration to deformed configuration.



Fig. The undeformed and deformed bar

Solution:

*The components of the deformation gradient tensor and its inverse*

***F = , =***

and the Jacobian is equal to *J* = *.* Clearly, denotes the ratio of deformed to undeformed cross-sectional area.

The unit vector normal to the undeformed cross-sectional area is , and the unit vector normal to the cross-sectional area of the deformed configuration is

**=**

The Cauchy stress tensor is = and associated stress vector is *t*= .

*The components of the Cauchy stress tensor*

**= =**

The second Piola–Kirchhoff stress tensor is given by

*The components of the second Piola–Kirchhoff stress tensor*

***S =***

=

The second Piola–Kirchhoff stress tensor can be written as

***S =***

So the second Piola–Kirchhoff stress tensor is non-varying with the deformation, it satisfies the balance equations in the undeformed body. Hence we can see the advantage of employing it.

Also an important consideration is that it forms the energy conjugate with the Green strain.

**Chapter 3: Linear Elasticity Theory**

The purpose of this chapter is to derive the Principle of Virtual Work as fundamental for the formulation of the Finite Element Method. The basis of the so-called weak formulation of the Initial Boundary Value Problem of elastodynamics is characterized by the description of the deformation of a material body by means of the displacement field and the corresponding strains (Kinematics), the force equilibrium of stresses on a differential volume element (Kinetics), the formulation of geometric and static boundary conditions and the constitutive relationship between stresses and strains (Material Law).

The primary variables of elastostatics are the displacements, since the stresses can be described by means of the Constitutive Law as a function of the stresses. In case of structures in motion, the primary variables along with their second time derivatives, the accelerations, are considered. The change from the strong form of the partial differential equation and its boundary conditions to the weak form gives in the end the Principle of Virtual Work. In the weak form the geometric boundary conditions are strongly satisfied, whereas the balance of momentum and the static boundary conditions must only be satisfied in an integral form. This integral formulation hence allows the exact solution of the Initial Boundary Value Problem to be replaced by an approximated solution, which satisfies the integral but not the local form of the corresponding differential equation. This shows the significance of the weak formulation of the fundamental equations of structural mechanics for the design of approximation methods in general and of the Finite Element Method in particular.

3.1 Continuum Kinematics

Continuum kinematics describes the geometry of a body, its motion in space as well as its deformation during motion. A basis for this description is the consideration of a body as an ensemble of material points as well as the characterization of their initial and current position by means of the position and displacement vectors. By considering the immediate vicinity of material points one finally gets to the concept of strains, which describe the deformation of a material body. First, the strains are described without further assumptions in a non-linear form and afterwards they are reduced to a linear description according to the deformation theory of small displacements or strains. For the description of non-linear kinematics the material or Lagrangian approach will be used, according to which the state of a point is defined as a function of its initial position and time.

Displacement Field

The motion of continuum in three-dimensional space is completely defined by the position vector of a material point *X = [X1 X2 X3]T*and its change of position at deformation under arbitrary internal or external influence. This motion of the material point from the undeformed to the deformed state is described by means of the displacement vector *u = [u1 u2 u3]T* as a function of the position of the material point (fig3.1). The components of the position and displacement vectors are defined in the cartesian basis with the orthogonal unit vectors, base vectors or simply bases *ei* for *i* {*1, 2, 3*}. Thus, the vectors can be described by their components and the base vectors as follows:

***Xi = ei .X X = eiXi***

***ui(Xi) = ei .u(X) u(X) = ei ui(Xi) (3.1)***

where the dot ‘.’ represents the scalar product of two vectors or tensors of the same order.

Furthermore, Einstein's summation convention is assumed to hold. The current position of the

material point under consideration at time t is given by the position vector

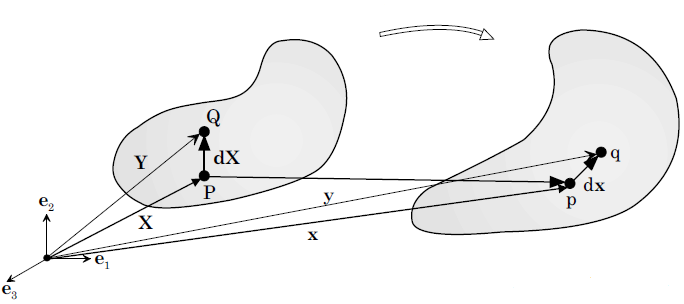
***x(X, t) = X+ u(X, t) x(X, 0) = X (3.2)***

The Lagrangian approach is to be observed clearly here in the context of the dependence of the current position on the initial position and on time t. Here, time is of physical relevance only in dynamic considerations. In the static case, time is transformed into pseudo-time, which only serves to characterize the state of deformation. On the basis of this formulation the state and shape of the deformed body can be fully described, but an expression for the local strains or elongations, actually is not possible.

Definition of a Non-Linear Strain Measure

According to the explanations above, an expression for the local strains can be obtained by considering the immediate vicinity of a material point. Here, the motion of a body is described by its displacement field *u(X, t)*. Fig. 3.1 illustrates a material body in its undeformed and deformed states. These positions are designated as reference configuration and current configuration. The deformation of the body from the reference to the current configuration is described in general by means of the time-dependent mapping *φ(X, t)* of all particles of the body. The displacement vector of a point with the coordinate X is given by Eq. (3.2) as the difference between its deformed and undeformed positions.

***u(X, t) = φ(X, t) - φ(X,0) = x(X, t) - X (3.3)***



**Figure 3.1: Undeformed and deformed configurations of a material body**

*x(X, t) = φ(X, t)* is the current state of a particle under consideration in the deformed body, characterized by its position in the reference configuration X and by the mapping of the position in the current configuration. The behaviour of the immediate vicinity of a material point according to the mapping *φ(X, t)* can be observed by means of a differential line element *dX*.

This line element is defined by the connection between two points P and Q at a differential distance from one another, expressed by the differential vector *dX = XQ - X* in the reference

configuration, and between the points p and q respectively, described by the vector *dx = xq – x* in the current configuration of the body. By a Taylor series expansion of the current configuration *φ(X, t)* with respect to the reference configuration *X*, one obtains the differentially distant point *y = x(X, t) + dx(X, t)* on the deformed configuration.

***x(X, t) + dx(X, t) = φ(X, t) + (XQ – X) + . . . = x(X, t) + dX + . . . (3.4)***

By truncating the endless series after the linear term and by using Eq. (3.3) in the above equation, the mapping or transformation of the differential line element *dX* of the reference

configuration to the current line element *dx* can be obtained .

***dx = dX = (u(X, t) + X)dX = ( + 1)dX (3.5)***

Here, 1 is the second order unit tensor, the components of which represent the *Kronecker symbol* *δij*,

***1 = = δij ei \* ej δij = (3.6)***

and the first term defines the derivative of the displacement vector with respect to the position vector of the reference configuration. This term is designated as the *Displacement gradient* **u**.

***= u(X, t) = = = ei \* ej (3.7)***

As a measure for the change in length of a line element *dX* during deformation, the square of the length *dS2 = 2 = dX . dX, or ds2 = 2*, of the line elements is observed in the reference configuration, and in the current configuration, respectively.

***ds2 = dx . dx = (dX + u dX) . (dX + udX)***

***= dX . dX + dX .u .dX + dX .u .dX + dX . u . u .dX (3.8)***

After some additional simplifications and taking into account the definition of *dS2*, half of the relative change in length can be obtained.

***= dX .* [u + u + u . u] *. dX (3.9)***

The middle tensor in Equation (3.9) represents the strain state of continuum. It defines the *Green Lagrange Strain Tensor* ***E***.

***E =* [u + u + u . u] *(3.10)***

Through this definition of a strain measure it is guaranteed that the reference configuration

(*u = 0*), and the displacements of a rigid body (*u* = 0) are free of strain. It should be noted, however, that this is not the only possible definition of a strain measure. Nevertheless, in this text the Green Lagrange strain tensor will be exclusively used in its original and linearized form. For simplification of Eq. (3.10), the displacement gradient *u* is decomposed into a symmetric and a skew-symmetric part.

**u = + = [u + u] + [u - u] *(3.11)***

Based on this decomposition, the Green Lagrange strain tensor can be written in the following compact form:

***E* = +  u . u = [u + u] *(3.12)***

The first term in this equation is a linear function of the displacement gradient *u*. In contrast to this, the second term *(1/2) u . u* is non-linear in *u*. This non-linearity, based on the mapping of geometry from the undeformed to the deformed state, is called geometrical non-linearity. The non-linear term affects the strain tensor decisively only when the gradient of the displacement field is big. This can occur in slender structures like rope structures and shells or in the case of plastification or damage of materials which is of importance, for instance, in

geomechanics or in the analysis of highly-loaded structural elements.

Definition of a Linear Strain Measure

In contrast to the previous section, the non-linear term of the strain tensor can be neglected if the deformations are very small (*(1/2) u .u = 0*). In this case we speak of the geometrically linear theory, which is also known as the theory of small strains. The strain measure of the geometrically linear theory is thus defined by the symmetric part of the displacement gradient

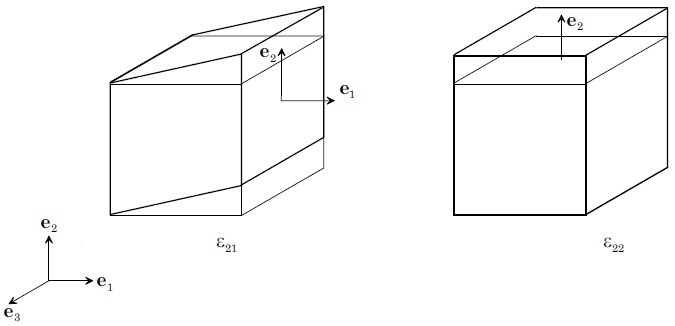
*u*.

**= = [u + u] *(3.13)***

The linear strain tensor, which is also described as the infinitesimal strain tensor, is denoted with to represent the theory of small strains. The components of the symmetric strain tensor can be described by the definitions of the symmetric part of a second order tensor and the gradient.

**= = ( + ) *ei \* ej (3.14)***

The definition of the strain tensor components is illustrated in Fig. 3.2, with *=* being valid due to the symmetry of the strain tensor. In the chosen definition, the first indexcharacterizes the strain direction. The second index characterizes the normal to the distorted surface of the representative volume element.



**Fig 3.2: Shear and normal strains on a volume element**

In the context of the Finite Element Method, the strain state is characterized by means of the *strain vector* . The strain vector defined below contains the normal strains *,,* andas well as the three differing shear strains and .

**= [ ]T  = *(3.15)***

The construction of the strain vector from the strain tensor is shown in the left part of Eq. (3.15). Factor two, with which the shear strain components are equipped, is of special importance. By means of this factor, the formally equivalent formulation of the specific internal energy in the tensor and vector notation ( . = :) is possible in connection with the stress tensor and vector yet to be defined. A further advantage of this definition will manifest itself in the equivalence of the differential operator and the transposed differential operator in the representation of the strains and the balance of momentum by means of differential operators. The first differential operator has to be developed as a basis for the direct calculation of the strain vector from the displacement vector. The desired kinematic relation of the strain and the displacement vectors is derived from the definition of the strain components in Eq. (3.14), whereby the components of the differential operator represent rules for derivatives.

**= = u *(3.16)***

The validity of the differentiation model (3.16) can be tested by the calculation of separate strain components and by comparison with their definition according to (3.14). As an example, the strain components and are computed here

**= = = + = + *(3.17)***

3.2 Continuum Kinetics

Kinetics describes the relation between external and internal forces acting on a material body.

According to the stress principle of Cauchy, a tensor field of stresses exists in a material body as a consequence of the external forces. Together with the static and dynamic loads acting throughout the volume, these stresses form the local balance of momentum or the equilibrium of forces. The balance of momentum must be satisfied throughout the deformed configuration. In the context of the here utilized geometrically linear theory it is admitted, however, to form the equilibrium of forces for the undeformed state.

Cauchy's Theorem

Cauchy's theorem is based upon the postulate of a stress vector ***t***on an arbitrary cross section of a material body.

This stress vector is defined as the ratio of the force *f*, acting on the section and the cross-sectional area *A*, when the area approaches zero.

***t = (3.18)***

Here, the orientation of the surface is characterized by means of its *normal vector n*. According to the Cauchy Lemma, the stress vector in the interior of the body as a function of the outward directed normal is balanced with the stress vector of the inward directed normal (t(n)+t(-n) = 0). The theorem of Cauchy now demands that a tensor field related to the vector *t* exists, which satisfies a linear mapping as follows:

***t(X, n) = (X) . n (3.19)***

The so-postulated symmetric stress tensor is known as *Cauchy's stress tensor*.

**= = *ei \* ej  = T (3.20)***

Analogously to the definition of strains, the first index indicates the stress direction and the second one the surface with the corresponding normal. By estimating the balance of angular momentum the symmetry of the Cauchy stress tensor  *=T* can be shown. If the moment equilibrium of all the stress components multiplied with the areas on which they act is formed around the middle point of the representative volume element with dimensions *dX1, dX2* and *dX3*, the symmetry of the stress tensor follows. As an example, the equilibrium of moments around the e3-coordinate axis is shown.

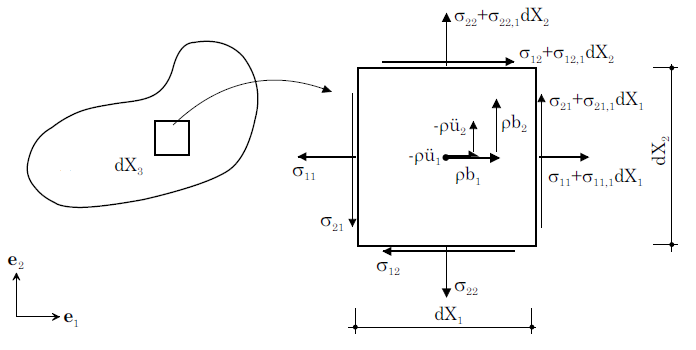
**2*dX1 dX3  -* 2*dX2 dX3  = 0 - = 0 (3.21)***

Balance of Momentum

The balance equation of the linear momentum describes the equilibrium of the internal forces and the stresses. The forces acting on a body can be classified as:

* deformation-independent, volume-specific loads *b =* [*b1 b2 b3*]T (physical units ),
* volume-specific inertial forces, which according to the *Newton Axiom* are opposite in direction to the acceleration *- ü = -* [*ü1 ü2 ü3*]T (physical units = ), and forces resulting from the stresses

The local balance of momentum can be derived in accordance with continuum mechanics, based on the integral balance of momentum and under consideration of Cauchy's theorem and some mathematical simplifications. Alternatively, a clear argument must lead to the equilibrium of forces. The derivation of the internal forces equilibrium or the momentum law is limited to the two-dimensional case and afterwards is expanded for spatial considerations. Consider the differential area element *dX1dX2* of depth *dX3*. The volume-specific loads*b* and *– ü* act in the centre. At the boundaries of the volume element, the stress components with the corresponding area elements contribute to the force equilibrium. Here, the differential changes of the stress components inside the area element and the symmetry of the stress tensor ( = ) are taken into account. The force equilibrium in the direction of the base vector e1 contains the stress components *,*  and the components of the volume-specific loads *b1* and *ü1*.



**Fig. 3.3 Momentum balance of a differential volume element (2-D)**

**0 *= ( + ) -***

***+* *( + ) -***

***+ (b1 -ü1) (3.22)***

The stress components and vanish, which means that only differentiated stress components take part in the equilibrium formulation. The division by the element volume

*dX1 dX2 dX3* results in the local form of the momentum law in *e1*-direction. Analogously, the partial differential equation for the orthogonal direction *e2* can be developed and expanded for three-dimensional considerations.

***ü1= + + b1 = + + b1***

***ü2= + + b2 = + + b2 , = (3.23)***

This results in the following system of partial differential equations in 3-D:

***ü1 = + + + b1***

***ü2 = + + + b2 = (3.24)***

***ü3 = + + + b3***

Hence, in tensorial form, the local form of the momentum balance, the force equilibrium or the

*Cauchy's equation of motion* is:

***ü =* div *+ b = ( + bi )ei (3.25)***

Herediv symbolizes the divergence of the Cauchy stress tensor . The application of divergence to the second order stress tensor yields a volume-specific force vector, which according to the momentum balance (3.25) is in equilibrium with the inertial forces and the volume loads.

**div = = *ei , = (3.26)***

Alternatively, the momentum law can be represented in component form.

***b1 -ü1 = = , = (3.27)***

The tensor is the conjugated stress magnitude to the strain tensor as defined in the kinematic equation (3.13). In the geometrically non-linear case, the stress tensor must be replaced by the *second Piola Kirchoff stress tensor* ***S***, conjugated to the *Green Lagrange strain tensor* ***E***. The latter appears in the non-linear balance of momentum in the transformed form of the material deformation gradient ***F = dx/dX***. It should be noted that in this case the density is also measured in the instantaneous configuration (seen later in non-linear elasticity). In the geometrically linear considerations of the deformations, differentiation between the stress tensors defined in the different configurations is not necessary.

Analogous to the definition of the strain vector, the components of the stress tensor can be written in a vector form. The defined stress vector contains the normal stress components , and as well as the shear stress components , and . In contrast to the strain vector, the shear components are not factorized.

**= [ ]T =  *(3.28)***

By means of equation (3.24), the balance of momentum (3.25) can be formulated based on the stress vector and the definition of the differential operator .

**= + = + *b* (3.29)**

By comparing equations (1.16) and (1.29), the relation between the differential operators and is obtained.

= [**]T *(3.30)***

Initial Stresses

Equilibrium stresses , which satisfy the balance of momentum (3.25), can have different origins. The first and also the most important cause of stresses are the strains in the material body. These constitutive stresses can be calculated by means of the constitutive law (seen later) on the basis of the strain state. On the other hand, initial stresses can be present in a material body in the undeformed state. These can be internal stresses, which appear, for example, in the cooling process of castings or in prestressed concrete elements or rope structures.

**= +  *(3.31)***

3.3 Initial and Boundary Conditions

The basic equations of kinematics and kinetics, derived in the previous sections are valid inside a material body or domain **Ω** at an arbitrary point in time. This system of equations has to be supplemented with initial conditions for the displacement or the acceleration field and with boundary conditions concerning the characteristic kinematic and kinetic size of the body's surface or the domain boundary **Γ**.

Classification of Initial and Boundary Conditions

The volume or domain Ω is limited by the boundaryΓ. The balance of momentum (3.25), including the definition of the strain measure (3.13), holds throughout. Furthermore, in the case of time-dependent problems, initial conditions in the domain Ω have to be prescribed. The domain's boundary Γ is divided into the non-overlapping *Dirichlet boundary* and *Neumann boundary*

**Γ = , = *(3.32)***

Here, as a rule, the primary variable is prescribed on the Dirichlet boundary, and dependent quantities are prescribed on the Neumann boundary. In the context of elastomechanics, these are the displacements ***u*** and the stress vector ***t***, respectively.

Dirichlet Boundary Conditions

Continuum kinematics is supplemented by the essential, geometrical or Dirichlet boundary conditions. Dirichlet boundary conditions are prescribed displacements at a given time *t* for the region of the boundary Γ.

***u(X, t) = u\*(X, t) X (3.33)***

If the prescribed displacements are identical to zero, they are referred to as homogeneous Dirichlet boundary conditions, which are prescribed, for instance, by supports.

***u(X, t) = 0 X (3.34)***

Neumann Boundary Conditions

For the derivation of the static, natural or Neumann boundary conditions, the two-dimensional case is considered first. Afterwards, the derived system of equations is expanded to three dimensions. The surface is characterized by the normal vector *n* = [*n1 n2*]T with = 1. The stress vector *t\** = [ ]T related to the line element *dS* is held in equilibrium by the stresses on the surface elements *dX1* und *dX2*. The force equilibrium in the direction of *e1*

***dX2dX3 + dX1dX3 = dSdX3 (3.35)***

divided by the depth *dX3* and side length *dS* yields the following condition:

**+ =  *(3.36)***

Here, the derivatives *dX1/dS* and *dX2/dS* can be obtained from the similarity of the normal vector triangle with sides *n1*, *n2*, = 1, and the geometrical triangle with sides *dX1*, *dX2*, *dS*

= = ***n2***  = = ***n1 (3.37)***

The force equilibrium is formed analogously in the direction of *e2*, obtaining the system of equations for the two-dimensional case,

**n1 + n2 =**

**n1 + n2 = *(3.38)***

For an expanded three-dimensional consideration, the force equilibrium of the surface element with normal vector *n* = [*n1 n2 n3*]*T*and stress vector on the surface *t\** = [ ]*T* is as follows:

***n1 + n2 + n3 =***

***n1 + n2 + n3 = = (3.39)***

***n1 + n2 + n3 =***

The force equilibrium at the stress or Neumann-boundary (Eq. (3.39)) can be written in a compact form in tensorial notation in the form of the Cauchy equation.

***(X, t).n = t\*(X, t) X (3.40)***

Here, the simple contraction of a first-order and a second-order tensor was used

***nj = = (3.41)***

Usage of the stress vector according to the definition in Eq. (3.28) results in the operator representation of the static boundary conditions.

**= = *t\* (3.42)***

3.4 Hyperelastic Constitutive Laws

In the previous sections stresses and strains were defined based on the momentum balance and the displacement field, respectively. Hence, both the stress tensor and the displacement vector are variables which are needed for the unambiguous description of the continuum's state of motion. This number of variables can be reduced by the postulate of a constitutive relationship which relates the stresses on the one hand, and the strains on the other. As a consequence of this postulate, the stresses become dependent on the displacement vector. This postulate is based on the observation of material behaviour under monotonous or cyclic loading. The variety of materials and their states induces various possibilities of mathematical description or modeling of material behaviour. First, the fundamental material models can be classified as linear and non-linear material models. Here, we want to restrict the variety of material models to linear models, which have proved to be representative in various engineering applications.

In this section, the fundamental assumptions for the formulation of the constitutive equations are stated first. Then, the material models of potential character, the so-called hyperelastic material models, are specified. Afterwards, the generalized Hooke's Law is formulated as a basis for the Finite Element Method in linear structural mechanics and specialized for the plane stress and strain state as well as for the classical one-dimensional Hooke's Law.

Fundamental Assumptions and Classification

Constitutive equations in the classical sense presume the existence of a relation between forces and deformation, respectively between stresses and strains, which is exclusively local, i.e., at the considered material point. In the context of this axiomatic prerequisite and assuming vanishing initial stresses ( = 0), a material law sets the relation between stresses , strains , strain rates , which describe the velocity dependence of the stress tensor, and internal variables , which represents the dependence of the stresses on the history (plastification or damage).

***= (, , ) (3.43)***

This generalized material law contains a number of material models for the description of non-linear material behaviour, taking into account microstructural damage, residual plastic strains and time-dependent effects. If, however, we focus our attention on the modelling of reversible, time-independent, elastic processes, the stress state can be defined only based on the strain state, with the stress tensor turning into a null tensor in the undeformed configuration.

***= () (3.44)***

Furthermore, it is to be assumed that the material is homogeneous and that the material properties are not dependent on the direction. The latter restriction characterizes an isotropic material model. It should be noted that the undertaken restriction to isotropic material models only has effects on the formulation of the material law in the following sections, and not on the formulation of linear finite elements.

Elastic Material Models

Elasticity means that the stress state only depends on the instantaneous strain state and not on the stress path. The desired path-independence is only guaranteed, if the stress tensor can be derived by differentiation of an elastic potential function *W()* with respect to the strain tensor.

***() = (3.45)***

If one integrates from *()*to *()* along an arbitrary path in the strain space, one obtains an energy difference independent of the path.

***= = W( -W() (3.46)***

If the deformation is independent of the path, the corresponding material laws are hyperelastic. Derivation of the stress tensor with respect to the strain tensor yields the *tangential modulus of elasticity, constitutive tensor or material tensor* ***C***. On the other hand, the material tensor represents the linear mapping of the strain tensor onto the stress tensor.

***C =*** ***= = ei \* ej \* ek \* el = C: = ekl ei \* ej  (3.47)***

As a consequence of the symmetry of the stress and strain tensors, the constitutive tensor satisfies the following symmetry properties:

***= = = (3.48)***

If the material tensor *C* is independent of the strains, i.e., a linear relationship exists between stresses and strains, we are talking about a physically or material linear constitutive law. All other material models are characterized correspondingly by the attributes physically or material non-linear.

Isotropic, Elastic Material Relation of Continuum

Based on the fundamental ideas for the formulation of material models in the previous sections, the generalized *Hooke's law* is to be derived as representative of three-dimensional, linear, elastic and isotropic material models. The isotropic, elastic material law is characterized by means of two material parameters. The representation of the constitutive equation is realized with the so-called *Lamé-constants* ***μ*** and **λ**. The relation of the *modulus of elasticity* ***E***, the *shear modulus* ***G*** and the *Poisson-transverse contraction ratio* is given by

***μ = = G*  λ *= (3.49)***

The potential function *W()* of the generalized Hooke's material law of the isotropic continuum is postulated as a quadratic function of the strain tensor and the chosen material parameters as follows:

**= *μ*: *+* λ(:1)2 *(3.50)***

By differentiation of the scalar-valued potential with respect to the strain tensor, the stress tensor is obtained according to Eq. (3.45).

***=* 2*μ*: *+* λ(:1) = (2*μij* + kkδij )ei \* ej *(3.51)***

In the above equation, 1 characterizes the second order identity tensor and the term:1 characterizes the trace of the tensor . The trace of the tensor can be designated alternatively by tr().

**tr() = :1 = *δijij = ii* = *11 + 22 + 33 (3.52)***

The fourth order constitutive tensor C is obtained by additional differentiation according to

Eq. (3.47)

***C =*** **2*μIsym +* λ1 \* 1 = [(*μ (δilδjk + δikδjl*) + λ *δijδkl*] *ei \* ej \* ek \* el*  *(3.53)***

Herein, *Isym* symbolizes the symmetric fourth-order unit tensor and 1 \* 1 symbolizes the *dyadic* product of second order unit tensors. The result of this dyadic product is a fourth order tensor.

***Isym* = *(δilδjk + δikδjl*) *ei \* ej \* ek \* el* 1 \* 1 = *δijδkl ei \* ej \* ek \* el (3.54)***

The symmetrical properties  *= = =*  of the constitutive tensor *C* result from the symmetry of the strain and stress tensors.

**= *kl =* [(*μ (δilδjk + δikδjl*) + λ *δijδkl*]*kl***

**= *μ (δilδjk kl + δikδjl**kl*) + λ *δijδklkl = μ (δil jl + δik**kj*) + λ *δijkk***

***= μ (ji + ij*) + λ *δijkk =* 2*μ ij* + λ *δijkk (3.55)***

Using the definition of stresses and strains in vector form in the context of the development of finite elements, one obtains the linear relation between kinematics and kinetics (Eqs. (3.47) and

(3.53)), or strain and stress vectors in matrix notation, respectively,

**= C  *(3.56)***

with the components of the constitutive matrix C connecting the components of the strain vector *kl*= *lk*and the stress vector = as follows.

**=**  ***(3.57)***

The entries of the constitutive matrix can be developed with Eq. (3.53) using the definition of the Kronecker symbol. As an example, the development of the components , ,

, is demonstrated.

**= *μ (δ11δ11 + δ11δ11*) + λ *δ11δ11 =* 2*μ* + λ**

***= μ (δ12δ12 + δ12δ12*) + λ *δ11δ22 =* λ**

***= μ (δ12δ11 + δ11δ12*) + λ *δ11δ12 =* 0**

***= μ (δ12δ21 + δ11δ22*) + λ *δ12δ12 = μ (3.58)***

All other components of the material stiffness matrix C can be obtained accordingly. Thus, the material stiffness matrix C is defined with the Lamé-parameters *μ* and λ.

**=**  ***(3.59)***

After a transformation of the material parameters according to Eq. (3.49), the constitutive matrix can be described by means of the modulus of elasticity *E* and the Poisson ratio .

**C =**  ***(3.60)***

For the deformation analysis of two-dimensional continua, the plane stress and the plane strain states are of interest. Typical applications of plane stress states are structural members of small depth, e.g. membranes, disks, plates and shells. The plane strain state is mostly used in cases where the dimension in one direction is very big with the loading in this direction remaining unchanged. The plane strain state is very common in the field of geo and soil mechanics.

Plane Stress State

In the case of a plane stress state it is assumed that the stress components , and vanish.

***= = = 0 (3.61)***

with the remaining stress components being constant. Substituting these in Equation (3.59) and solving we finally obtain the linear elastic material law of the plane stress state in the form  **= Ces**

**=  *(3.62)***

Or, alternatively, in terms of the material constants and *E*:

**=  *(3.63)***

Plane Strain State

Again in the case of a plane strain state it is assumed that the strain components *, ,* andvanish, or

**== = 0  *(3.64)***

with the remaining strains components being constant. Substituting these in Equation (3.59) and solving we finally obtain the linear elastic material law of the plane stress state in the form

**= Cev**

**=  *(3.65)***

By alternative parametrization the constitutive equation of plane strain turns into:

**=  *(3.66)***

The Classical Hooke's Law

The one-dimensional stress-strain relationship in the direction of the base vector *e1*, is based upon the assumptions

**= = = 0 *(3.67)***

such that on application of Equation (3.59) the following relation between stresses and strains results

**= =  *(3.68)***

The strain components  and can be expressed as functions of the normal strain (say from second row of Eq. (3.59))

***= = = (3.69)***

The *classical Hooke's law* describes the one-dimensional stress-strain relationship of a truss element or a spring.

***= = E (3.70)***

3.5 Initial Boundary Value Problem of Elastomechanics

The summary of the fundamental equations of the three-dimensional continuum, developed in the previous sections, forms the initial boundary value problem of elastomechanics. In detail, these were the description of deformation in the context of kinematics, the formulation of the force equilibrium based on kinetic considerations, the constitutive equation as well as the initial and boundary conditions.

Characterization

The character of the initial boundary value problem of structural mechanics depends on the type of structure and loading that have to be described, which, on the other hand, decisively affect the modelling of the load-carrying behaviour. In the previous sections, the essential modeling aspects were already discussed on a geometrical and material level. In summary, the modeling can be classified, in essence, according to the aspects of

* geometrical linearity or non-linearity,
* material linearity or non-linearity,
* and time-dependence or time-independence.

Geometrically and Materially Linear Elastodynamics

Under corresponding prerequisites, namely small deformations and small strains, now we perform structural analyses in the context of the geometrically and materially linear theory. The essential components of the description of small, linear elastic deformations make for the formulation of the relationship between displacement and strain field, the equilibrium of forces and the constitutive equation relating the stresses and strains. All three components (in tensor notation these are Equations (3.13, 3.25, 3.53)) together form the second order partial differential equation of linear elastodynamics with the displacement field as the solution variable.

***ü =* div *+ b***

**= *C* : *ü b* = div(*C*: *u*) *(3.71)***

**= *u***

Here, thermal strains and initial stresses were presumed to be zero

Geometrically and Materially Linear Elastostatics

In the case of static or quasi-static analyses of structures, the problem is reduced to that got by neglecting transient effects. The resulting differential equation is given by

**0 = div(*C*: *u*) + *b* *(3.72)***

3.6 Weak Form of the Initial Boundary Value Problem

The local behaviour of an elastic body was fully described in the previous sections by means of the initial boundary value problem. In general, the solution of this differential equation is not possible analytically. Therefore, approximation methods, in particular the Finite Element Method, are used in order to find an approximate solution. This method actually does not solve the so-called strong form of the differential equation. It merely solves its integral over the domain, the so-called weak form of the differential equation. This weak formulation forms the basic prerequisite for the application of approximation methods. Integral principles of mechanics are

* *the principle of virtual displacements or principle of virtual work,*
* *the principle of virtual forces*
* *and the principle of the minimum of total potential or its generalization for transient*

*considerations, the Hamiltion's principle of continuum*.

The principle of the minimum of the total potential requires the existence of a potential, whereby its applicability remains restricted to the structural mechanics of hyperelastic materials. Applied to structural mechanics, the principle of virtual forces represents the method of force magnitudes, which turned out to be inconvenient in the computer-oriented implementation. In contrast to that, the finite element method based on the principle of virtual work is universally applicable for arbitrary materials and excellently programmable. The derivation and discussion of the principle of virtual work in linear structural mechanics is what this section focuses on.

Principle of Virtual Work

For the generation of the principle of virtual work, the strong form of the differential equation, which corresponds with the local balance of momentum, as well as the static boundary condition are scalarly multiplied by a vector-valued test function and integrated over the volume, respectively over the Neumann boundary, of the body under consideration. As test function the *virtual displacements* *δu* are chosen.

* ***δu*** satisfies the geometrical boundary conditions

***δu* = 0  *X (3.73)***

* ***δu*** satisfies the field conditions

***δu = δ (3.74)***

* ***δu***  is infinitesimal
* ***δu*** is arbitrary

The weak formulation of the balance of momentum (3.25) and of the static boundary condition

(3.40) results from the reformulation of these fundamental equations,

**0 = *ü -* div *- b 0 = .n – (3.75)***

multiplication by the test function *δu*, integration over the volume, respectively over the Neumann boundary, and addition of the integral terms.

***ü –dV*** - **div *dV* +.*n* ) *dA=* 0 *(3.76)***

For the further simplification of this equation, the term *δu*.div is considered first. The latter can be transformed into div(*δu*.) by application of the product rule for divergence. Additionally, the interchangeability of the order of application of variation with the symbol *δ* and differentiation with the symbol is utilized for further simplification.

**div(*δu*.) = *δu*.div + *δu*: *= δu*.div + *δu*:**

***δu*.div = div(*δu*.) - *δu*: *=* div(*δu*.) - *δu*: *(3.77)***

This simplification can be derived or proved by means of components.

**= + *(3.78)***

Furthermore, the *Gauss theorem for the divergence* of a first order tensor is applied to the volume integral of the term

***dV* = *.n dA* = *.n dA (3.79)***

It is possible to substitute the boundary in the above equation by the Neumann boundary , since the test function *δu* is zero at the Dirichlet boundary, in accordance with (3.73). Using the equations (3.76), (3.77) and (3.79), the weak form of the momentum equation can be represented as follows:

***ü –dV* + *dV - .n dA* + *.n –*) *dA* = 0**

***(3.80)***

Finally, the term is examined and rewritten in an alternative form. This term represents the double contraction of the symmetrical stress tensor (see Eq. (3.20)) and the non-symmetrical tensor . can be substituted by the symmetrical part of this tensor. The latter again can be substituted by using the definition of the strain tensor (3.13).

**= ()*sym* = *sym* =  *(3.81)***

This reformulation can be proved by component representation.

= = =

***(3.82)***

After the first equality sign, the symmetry of the stress tensor () was introduced and after the second one - the interchangeability of dumb indices. Thus, the principle of virtual work is derived in its usual form, with the scalar product of the variation of the strain tensor and the stress tensor.

***ü dV* + *dV* = *dV* + *dA*  *(3.83)***

The separate summands in Eq. (3.84) are described as virtual work of the inertial forces *Wdyn* ,

internal virtual work *Wint* and virtual work of the external forces or external virtual work *Wext .*

***Wdyn = ü dV***

***Wdyn +Wint = Wext Wint = dV (3.84)***

***Wext = dV* + *A***

Componentwise, the principle of virtual work reads as follows:

***dV + dV* = *dV* + *dA (3.85)***

If, alternatively, the definition of stresses and strains as vectors is used for the generation of finite elements, and additionally the kinematic equation (3.16) and the constitutive law (3.59) are taken into account, one obtains the internal virtual work as function of the displacement vector *u*, the constitutive matrix C and the differential operator .

***Wint = dV = dV (3.96)***

Properties of the Principle of Virtual Work

Since the stresses () are functions of the strains and since these, in turn, depend on the displacements through the geometrical relation = (*u*), the equations (3.84)-(3.86) represent a conditional equation for the unknown displacements *u*. If one knows the solution of these equations, then this is also the solution of the corresponding strong form, namely the equilibrium condition (3.25). Since the equations (3.84)-(3.86) must hold for arbitrary test functions *u*, they contain the differential equations of the momentum law and the static boundary conditions.

If, however, the principle of virtual work is not solved exactly but by means of approximation functions (as is the case with the Finite Element Method), the solutions of the weak and the strong form are not identical. The approximate solution for the displacements, when introduced into the strong form of the momentum law, results in an error, the so-called residuum. This means that the strong and the weak forms are identical in the continuous case but not in the discrete one. Since the integral form of the equilibrium equation and the Neumann boundary condition allows local errors, it forms the basis for the development of the Finite Element Method.

As a consequence of the choice of the virtual displacement *u* as a special type of a test function which satisfies the geometrical boundary conditions (3.73), the geometrical boundary conditions are strongly fulfilled in the principle of virtual work. On the other hand, the equilibrium and the static boundary conditions are only weakly fulfilled, i.e. in an integral sense, because of their multiplication by the test function and integration over the volume.

Due to the significance of the principle of virtual work for the development of the Finite Element

Method, the obtained notions should be summarized in the end:

* The Dirichlet boundary conditions are strongly fulfilled in the principle of virtual work.
* The Neumann boundary conditions and the equilibrium equation must be fulfilled only weakly in the principle of virtual work.
* The advantage of the integral form over the differential form lies in the fact that the weak form 'forgives' local errors which can arise during approximation, as long as the differential equation is fulfilled in an integral sense.
* Because of this reason, the weak formulation forms the basis for the Finite Element Method.

**Chapter 4: Finite Element Method using Spatial Isoparametric Truss Elements**

The practical application of finite truss elements is of secondary importance. Pure truss structures can also be computed in a simple and straightforward manner by the classical methods of statics. Only the combination with other finite elements, such as volume or plate elements, for the modelling of complex structures, distinguishes the finite truss element from the classical methods of static analysis of truss structures.

The development of isoparametric finite truss elements presented here is methodically oriented towards the generation of multidimensional isoparametric finite elements, and of structural elements, such as beam and plate elements. Since the truss element is considered as the simplest element with respect to the element development and the mechanical and mathematical understanding related to it, it is superbly suited to represent the fundamental methodologies and relationships of the Finite Element Method. Therefore, the significance of the truss element is to be evaluated quite differently from its practical relevance. It is well suited to understand modelling, approximation and finite element generation.

Based on the motion of the elastic three-dimensional continuum formulated in the previous chapter, the fundamental equations of one-dimensional continua are derived first. In analogy to essentially more complicated structural models, such as disks, plates, beams or shells, this derivation is based, beside the definition of geometry, only on two additional assumptions related to kinetics and kinematics. These are the assumptions of the emerging one-dimensional stress state and the constant displacement field over the cross section of the truss. The necessary steps for the finite element analysis of a structure will be explained by means of spatial truss structures and the resulting differential equation of separate truss elements. In particular, this can be decomposed into the three main processes of domain partitioning or localization, finite element

discretization, assembly and solution of the resulting system equation. The main emphasis of the finite element development is placed here on the discretization, which in turn is decomposed into the approximation of the primary variables, the dependent variables and, in the end, the virtual work. Through the last step, the internal virtual work, the external virtual work and the virtual work of the inertial forces can be calculated in an approximative way by means of the element stiffness matrix, the consistent load vector, the element mass matrix and a finite number of element parameters, the element displacement vector and the element acceleration vector.

4.1 Fundamental Equations of One-dimensional Continua

Geometry

A truss element is characterized geometrically by the aspect of slenderness. This means that the length *L* of the truss in one direction is essentially greater than the dimensions in orthogonal directions. Furthermore, for the simplification of the following derivations, a truss element with a constant cross-sectional area *A* is assumed. Without loss of generality, the coordinate system is chosen in such a way, so that the base vector *e1* points in the direction of the longitudinal axis of the truss.

Kinetics

The first fundamental assumption for the modelling of the truss is that five of the six stress components of the stress tensor are zero.

***= = = = =* 0 *(4.1)***

The remaining stress state 0. Here the constant normal stress distribution follows by means of the material law from the kinematic assumption given in next section. From the requirement postulated in Eq. (3.1), and taking into consideration the Cauchy Eq. (3.40)

**= = *(4.2)***

It follows directly that only the component of the surface stress vector could be different from zero.

**=**  =  **= 0 *(4.3)***

For a truss of a constant cross-sectional area *A* the normal vector *n* of the outer surface of the truss is parallel to the plane, spanned by the base vectors *e1* and *e2*, i.e., the component of the normal vector being identically zero on the outer surface. Hence, also the stress component vanishes on the outer surface of the truss. However, on the faces this stress component is present.

**= 0 *X , X1* =  *L/2 (4.4)***

Kinematics

The second fundamental assumption for the truss concerns the displacement field: all material points of a truss cross section experience the same displacement *u1* in the longitudinal direction.

**= ()  *(4.5)***

Consequently, the strain component is also constant across the cross section, according to the defining equation of the strain tensor (3.13).

**= () = () *(4.6)***

The other components of the strain vector of a truss can be obtained using the assumption postulated in Eq. (4.1) and by application of the constitutive law of the three-dimensional continuum (3.59)

**=**  ***(4.7)***

This equation is fulfilled for the prescribed stress state when the shear terms of the strain vector vanish = = = 0. For the longitudinal normal strain and the transverse normal strains and the following relationship results

***= = = (4.8)***

Constitutive Equation

The constitutive equation of the truss relates the longitudinal normal stresses and strains. Based on the three-dimensional linear elastic constitutive law (3.47, 3.53), the classical Hooke's law can be developed by application of Eq. (4.8) and the transition to the modulus of elasticity as the characteristic material parameter

**= + ( *+ = = E (4.9)***

As already mentioned, it follows from here that the normal stresses are constant across the truss cross section. From the static boundary condition in Eq. (4.2) also the stress vector is constant at the end cross sections (for all other regions of the outer surface = 0, see Eq. (2.4))

= **(*X*1)** = **(*X*1)**  ***X*1 *= L/2 (4.10)***

Principle of Virtual Work

With kinematics, kinetics and the material law at hand, the essential relations for the formulation of the Principle of Virtual Work of a truss are now available. The fundamental assumptions concerning the stress state and the displacement field, formulated in Eqs. (4.1) and (4.5), together with the introduction of the resulting consequences for the strain state into the principle of virtual work of the three-dimensional continuum (Eq. (3.86)) allow the shear-free stress and strain representation of the internal virtual work.

**+ = = +**

**+= *dV* +  *dV (4.11)***

**+ = *dA* + *dV***

The internal virtual work and the virtual work of the surface loads contain only terms in the longitudinal direction. Since the accelerations and the volume-specific loads exhibit transverse components different from zero, further detailed studies should provide information for their physical meaning and relevance.

Virtual Work of Inertial Forces

For a detailed analysis of the virtual work of inertial forces , the integration over the volume element *dV* is split into the integration over the area element of the cross section *dA* and the line element of the length of the truss *dX1*. Before integration of the virtual work of inertial forces, a functional dependence of the transverse accelerations and and the distribution of the transverse components of the test function from the coordinates *X2* and *X3* have to be explored. From the definition of the strain components and in Eq. (4.6), the fundamental assumption Eq. (4.5) and Eq. (4.8), and by integrating, one obtains a linear distribution of the displacement and acceleration components across the thickness or width of the truss.

**= = = - = - = -  *,* = -**

**= = = - = - = -  *,* = -  *(4.12)***

Since the displacements *u2* and *u3* depend linearly on the coordinates *X2* und *X3*, the variation of these displacement components is independent of the cross-sectional coordinates *X2* and *X3*.

**= (*X*1) = (*X*1)  *(4.13)***

Further, the acceleration component (*X1*) and the variation(*X*1) are constant across the cross section *A*, according to the kinematic assumption (4.5),

**= (*X*1) = (*X*1)  *(4.14)***

wherefrom the virtual work of the inertial forces can be split into a longitudinal and a transverse part, as follows:

**= *- . (4.15)***

The integral gives the cross-sectional area *A* of the truss and the integration of *X2* and *X3* across the cross section is zero when the coordinate origin is in the centre of gravity of the cross section. Under these prerequisites, only the acceleration in the longitudinal direction is contained in the virtual work of the inertial forces.

**= *A (4.16)***

Virtual Work of the Surface Loads

The examination of the virtual work of the surface loads is based upon the utilization of Eq. (4.11). The integral over the surface of the truss is split into integrals over the outer surface and the faces of the truss, where the first integral results in zero, since on the outer surface the component of the stress vector is identical to zero, according to Eqs. (4.3) and (4.4). The virtual work of the stress component , which is constant across the cross section, remains to be determined at the end sections of the truss *X1*= -*L/2* and *X1* = +*L/2*.

**= [()**(**) + ()**(**)] = ()**(**) + ()**(**) *(4.17)***

In the last simplification of this equation, the integral = *A* was introduced as well as the definition of the prescribed normal forces at the boundary, resulting from the integration of across the cross section.

(**)*A* =** (**)** (**)*A* =** (**) *(4.18)***

Virtual Work of the Volume Loads

In comparison to Eq. (4.11), the virtual work of the volume loads cannot be simplified directly. If one actually applies the *Fundamental Lemma of Variational Calculus* to Eq. (4.11), taking into account Eq. (4.16), then *b2*and *b3* have to become zero for arbitrary variations and , as the virtual work of the inertial forces (Eq. (4.16)) and the internal virtual work (Eq.

(4.11)) as well as the virtual work of the surface loads (Eq. (4.17)) do not contain any terms in and . This means that the assumptions for the truss allow no volume loads normal to the longitudinal axis.

**= = 0 *(4.19)***

The simplest case of a volume load, the self-weight, can be therefore realized only for the special case of identical longitudinal and gravitation vectors consistent with the truss theory. Therefore, the component *b1* gives the only contribution to the virtual work of the volume loads.

**=  *() () = dA (4.20)***

Here, the local balance of momentum *ü1 = + b1*, according to Eq. (3.24), was used for the definition of the distributed load *()*as an integral of the volume load over the cross section of the truss. According to it must also be independent of the coordinates *X2* and *X3*. In order to realize the influence of the volume loads and on a multi-truss structure within the framework of structural analysis, these quantities are introduced as kinematically equivalent node loads *rb2* and *rb3*. The kinematically equivalent node loads must satisfy the condition

**.) + .) =  *i =* 2, 3 *(4.21)***

If, in addition, constant specific loads are assumed across the cross section, which is the case, for instance in the formulation of the self-weight, the equivalent node loads result in:

**.) + .) =  *i =* 2, 3 *(4.22)***

Internal Virtual Work

The internal virtual work is formed according to Eq. (4.11) only with the normal components of the strains and stresses in *e1* direction. With the splitting of the volume integral, analogously to the virtual work of the inertial forces, the internal virtual work follows as

**=**  =  ***(4.23)***

where care was taken, so that the normal strains and stresses would be independent of the coordinates *X2* and *X3* (Eqs. (4.6) and (4.10)).

Principle of Virtual Work of the Truss

The summary of Eqs. (4.11), (4.16), (4.17), (4.20) and (4.23) results in the principle of virtual work for the variation of the displacement , for the development of a finite truss element.

***A*  + = + +  *(4.24)***

In Eq. (4.24), the loads and the virtual displacements at the truss ends were defined in the following manner:

**=** (**) , =** (**) = () , = ()  *(4.25)***



**Fig. 4.1 Model of the truss and volume loads**

Euler Differential Equation and Neumann Boundary Conditions

In order to derive the *Euler differential equation* and the Neumann boundary condition of the truss element by means of the fundamental lemma of variational calculus, the variation of the strain in the term of the internal virtual work, according to Eq. (4.23), must be replaced by the variation of the displacement component. Therefore, the derivative

**= + = +  *(4.26)***

was obtained with the help of the product rule and was introduced in Eq. (4.23).

**= ,1  -  *(4.27)***

Here, the definition of the normal force of the truss *N1* was introduced as a stress resultant.

**= =  *(4.28)***

The first integrant in Eq. (4.27) is now transformed by application of the basic law of integral calculus, *i.e.* *Application of Gauss integral theorem in one-dimensional space*.

**= - -  *(4.29)***

Together with Eq. (4.23), the weak form of the differential equation and of the Neumann boundary condition of the truss is produced.

***A*  - + -**

**= +** (**) +** (**) *(4.30)***

The Euler differential equation or the kinetics of the truss follows from the consideration of Eq. (4.30) for arbitrary test functions . The differential equation is supplemented by the constitutive law according to Eq. (4.9), the kinematics are supplemented according to Eq. (4.6).

**=  *A - () Kinetic***

**= *EA* *Constitutive (4.31)***

= ***Kinematic***

The Neumann boundary conditions result from Eq. (4.30) for arbitrary variations at the truss ends and .

***=*** (**)  *=*** (**) *(4.32)***

4.2 Finite Element Discretization

Partitioning of the Structure into Elements

From the remarks to the principle of virtual work in section 3.6 it follows that during the transition from the infinite element to the total body, or from the differential to the integral form, respectively, local errors in the solution are tolerated. The integral balance of the virtual work, however, is fulfilled. This fact motivates the subdivision of a structure into finite elements, in which the principle of virtual work is always fulfilled in an integral sense. However, local errors in the fundamental balance equation and Neumann boundary conditions are accepted. It is obvious that with diminishing finite elements the exact solution is approximated more accurately. In the limiting case of an infinitely great number of finite elements, the finite element turns into the infinite or differential element and the approximate solution becomes the exact one.

Mathematically, the structure or the domain is formed by the union of domains of finite dimensions . Furthermore, the partial domains may not overlap.

**= = *for i j (4.33)***

The principle of virtual work must be fulfilled for the domain and for each domain ,

**+ = + =  *(4.34)***

where the sum of the virtual work terms of all finite elements must result in the corresponding virtual work terms of the structure

**= = =  *(4.35)***

Approximation of Variables of One-dimensional Continua

In the finite element discretization of a truss element, the (unknown) exact displacement field is replaced by an approximation 1(the displacement distribution *u1* depending on the physical coordinate *X1*, or on the natural coordinate , respectively). The corresponding approximation (shape) function is defined by few parameters and the assumption for its qualitative form. Although here only linear shape functions are discussed, polynomial functions have established themselves to be used as shape functions due to their favourable properties. For the systematic representation of such approximation functions of finite elements of similar geometry, shape functions are defined in the natural parameter space and afterwards the developed finite element is transformed to the particular element geometry. Within the framework of the isoparametric finite element concept, the discretization of position *X1*(), of variation of displacement () and of acceleration () follow directly from the discretization of displacement. Here, it is sufficient to discuss at first the approximation of displacement field and afterwards to apply it analogously to the other quantities that are to be approximated.

Shape Function

For the formulation of finite elements the shape functions must

* be conformable,
* supply at least constant strains
* and may not supply any strains during rigid body displacements.

The first viewpoint refers to the interaction and compatibility of neighbouring finite elements, whereas the other conditions are restricted locally to each finite element of a structure.

In order for no gaps or overlappings to occur at the element ends during deformation of a structure, the displacements must be conformable. This is guaranteed for a finite element with displacement approximations which satisfy the Dirichlet boundary conditions of the element. In the example of the truss, this requires that the displacements of neighbouring elements have the same value at the common node. In case non-conformable shape functions are used, the external virtual work terms of the nodal loads do not vanish during the assembly of the structure.

Constant strains are required, so that the variation of the strains or the strains themselves, and hence also the internal virtual work, do not become zero. This statement can be verified in the general three-dimensional case by taking into account Eq. (4.6) and the principle of virtual work of a truss (Eq. (4.24)).

The third criterion, which the shape functions must satisfy, is the requirement to be able to describe strain-free rigid body motion. This means that during a deformation-free motion of an element no internal virtual work may arise due to the shape functions.

Physical and Natural Coordinates



**Fig 4.2 Example of coordinate system for rectangular element**

In the case of a truss element, the position of a point relative to the longitudinal axis is measured in terms of the physical coordinate *X1* or the natural coordinate .

***(4.36)***

The relationship between the coordinates and is described by the equation

**=  *(4.37)***

wherefrom also the transformation of the corresponding differential line elements and is determined by the derivative of the physical coordinate with respect to the natural one.

**J = = = = =  *(4.38)***

Here, the index ‘;1’ was defined to symbolize the derivative with respect to the natural coordinate . Furthermore, in Eq. (4.38) the *Jacobi or functional matrix J* and its determinant, the Jacobi or functional determinant were defined. In the one-dimensional special case of a truss, the

Jacobi matrix is a (1 1) matrix or a scalar. Thus, in the one-dimensional case the Jacobi determinant is identical to the Jacobi matrix.

Linear Shape Functions

The linear shape functions ***N*()** of a truss element are as follows

***N1*() = (1)  *N2*() = (1) *(4.39)***

written in matrix form as

**N() =**  ***(4.40)***

Approximation of Variables

The approximation of the continuous fields (),() and () is realized by means of the matrix of shape functions N(), the element displacement vector , the variation of the element displacement vector and the element acceleration vector .

**() 1() = N() *= T***

**() 1() = N() *= T (4.41)***

**() 1() = N() *= T***

Isoparametric Approximation of Coordinates

In preparing the isoparametric finite element concept, the typical approach of this concept in the example of the linear truss element is to be demonstrated here. By the isoparametric concept the physical coordinate *X1* is described, in analogy to the displacement *()*, by means of the shape functions *Ni*() in the parameter space and the physical coordinates of the element nodes and .

***X1* () 1() = *N*() *= T (4.42)***

In the case of a linear truss element, this results in the mapping of coordinates from the parameter space , identical to Eq. (4.37), into the physical space *X1.*

***X1* () = (1) + (1) = (1) + (1+) = *(4.43)***

Consequently, also the approximated Jacobi matrix corresponds to the exact Jacobi matrix (4.38)

**J = =  *(4.44)***

Approximation of Strains

For the approximation of the strain field (), the coordinate transformation between the physical and natural coordinates (*X1* and ) must be taken into account in the derivation of the displacement field = . This means that the derivative of the approximated displacement field (), with respect to the coordinate in Eq. (4.6) must be computed by the application of the chain rule.

**() = () = = = . *(4.45)***

In the last reformulation, the inverse of the Jacobi matrix J-1= 2/L according to Eq. (4.38) was introduced. Thus, the approximation of the strain field is given by means of the approximation of the displacement field.

**() 11 = ( *(4.46)***

or by the vector notation

**() 11 = = B()  *(4.47)***

with the *B-operator or differential operator* B

**B() = =  *(4.48)***

being defined. In general, this differential operator is a matrix and in the special case of one-dimensional linear elements, it is a transposed vector or a (2 1) matrix. It relates the approximation of local continuous strains with the discrete element displacement vector .

Analogously the variation of the normal strain can also be approximated.

**11 = = B()  *(4.49)***

It should be noted that in the special case of a linear truss element, the differential operator B is independent of the position .

**= =  *(4.50)***

Approximated Internal Virtual Work

The approximated internal virtual work is obtained from Eq. (4.23) by plugging-in the (approximation) of the variation of the strain **11** and the material law  *= E* , in which, on the other hand, the approximation of the strain **11** is inserted

***= EA = EA = EA (4.51)***

Now applying Eqs. (4.47) and (4.49) we get in the matrix notation

***= .***

***= . (4.52)***

Since the shape functions are linear it follows that their derivatives = are constant. Because of this, the integral in Eq. (4.52) must be formed only over the line element with = 2

***= . (4.53)***

The matrix is the *element stiffness matrix* with respect to the coordinate system e*i* and the element displacement vector . Thus, the approximated internal virtual work can be computed with the element displacement vector, the variation of the element displacement vector and the

element stiffness matrix.

***= (4.54)***

Alternatively to the above method for development of the discrete virtual work, now the B-operator (Eq. (4.48)) must be used for the computation of the approximated virtual work. When introduced in Eq. (4.51), for the substitution of the approximated strains and their variation with the element displacement vector and its variation , respectively, the differential operator B results in:

***= = (4.55)***

So we get

**= =  *(4.56)***

The advantage of Eq. (4.56) lies in its standardized computational pattern of the element stiffness matrix *ke*, which is further found in generalized form in the formulation of two- and three dimensional finite elements.

Approximated External Virtual Work

The virtual work of the external loads can be described, according to Eq. (4.24), as function of the prescribed nodal loads and the prescribed distributed load

**= +  *(4.57)***

If the distributed load *p1* is given as function of the physical coordinates, it must be transformed to the natural coordinates. Therefore, from Eq. (4.57) only the variation of displacements has to be approximated according to Eq. (4.41).

**= . +  *(4.58)***

In Eq. (4.58)), the *load vector of the nodal loads*  and the *consistent element load vector*  are defined. This finally yield the virtual work of the external loads

**= + = .  *(4.59)***

For the special case of a constant uniform load, the consistent element load vector

**= = = =  *(4.60)***

represents the integral load *p1L*, distributed equally onto the element nodes.

Approximated Virtual Work of the Inertial Forces

The virtual work of the inertial forces in Eq. (4.24) can be approximated by substitution of and with the Eqs. (4.41) and writing in vector form, assuming constant cross section area *A* and density

**=  *(4.61)***

Further developing using the definition of shape function we get

**=**

= ***(4.62)***

is the *element mass matrix* and is the *element acceleration vector*. So the approximated virtual work of the inertial forces can be written as

**= . *(4.63)***

Alternatively we can write in a generalized form by means of the matrix of shape functions N()

and introducing the Jacobian determinant, as

**= . = .  *(4.64)***

**Chapter 5: Nonlinear Elasticity Theory**

In the last two chapters there were two simplifying assumptions: the material behaviour is linear elastic (materially linear) and that deformations are small (geometrically linear). The former assumption was prescribing the validity of the Hooke law and latter excluding the modelling of irreversible material behaviour such as plastification or damage. These assumptions make it inadequate for design analysis of real structures and machines.

The demands on models of structural engineering will notably increase due to growing replacement of development and verification experiments by cost-reducing, transparent and faster computer simulations. For example, structures which are slender and light for technical or aesthetic reasons can only be adequately simulated and examined regarding stability with the help of a geometrically nonlinear calculus. On the other hand, concrete or reinforced concrete is a material characterized by distinct non-linear behaviour due to inevitable cracks which has to be taken into account in the structural analysis calculus. Cupping and shaping processes in the field of industrial production or car crashes are application examples for simulations dependent on modelling of metal plastification with large deformations. On the one hand it is the lasting deformations and on the other hand the dissipated plastification energy that is of crucial significance for the product quality, that is, the safety of the passenger. That shaping processes are impossible without large deformations is self-evident; also the aftermath of a crash seldomly justifies the assumption of small deformations. Other structures are so intensively loaded that they are impossible with structural exclusion of non-linear material behaviour. Plastic deformation during exploitation has to be accepted and simulated accordingly in the design and development process e.g. simulation of incineration chambers. The previous list of necessities of non-linear simulation techniques can be extended almost at will, but we nevertheless want to concentrate on elaboration of geometrically or/and materially non-linear problems, the modelling thereof. Metal materials for example display a linear behaviour until they reach a certain stress level, the so called flow limit, above which their plastic deformations occur in connection with a notably reduced material stiffness. The consequence is that the affected structures undergo a load redistribution which still leaves them serviceable even though plastic deformations were already localized.

5.1 Types of mechanical non-linearities

* materially non-linear and geometrically linear
* materially linear and geometrically non-linear
* materially and geometrically non-linear with assumed moderate strains
* materially and geometrically non-linear with finite strains

The theory here will be limited to pure geometric non-linear local impulse equilibrium and its formulation, discretization, linearization. The material non-linearity will be presented only schematically and the corresponding FE discretization will be elaborated briefly.

5.2 Material non-linearity

Mathematical formulation of material non-linearity

The rate-independent material non-linearity is, contrary to materially linear formulations characterized by the fact that the stress state

***= C: (5.1)***

cannot be obtained by linear mapping of the strain state with the help of material tensor *C*. The stress tensor or vector is rather an arbitrary function of the strain tensor or vector , and other values , described as internal variables or as time history variables that characterize non-elastic deformations or damage

= **(**,)  ***(5.2)***

where the partial derivative of the stress tensor/vector with respect to the strain tensor defines the *tangential material tensor* or the *tangential material matrix*.

**(**,) =  ***(5.3)***

In order to make the solution of mechanical boundary problems or initial value problems possible, the stress function (5.2) has to be supplemented with the so called evolution equations of internal variables in the form

**= (**,)  ***(5.4)***

In the special case of non-linear elastic material laws, the stress state is only a function of the

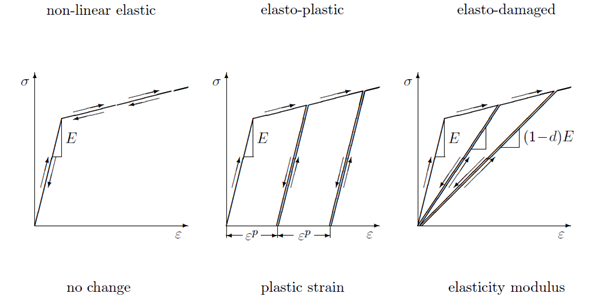
strain state

= **(**) **(**,) =  ***(5.5)***

and the evolution equations are dropped. For the simulation of the non-linear material law mostly

* non-linear elastic,
* elasto-plastic
* and elasto-damaged

material models and presented ground type combinations are in use.

**Fig. 5.1 Cyclic loading and unloading of elastic, elastoplastic, and elasto-damaged material models**

As shown in figure 5.1, the curves of three materially non-linear phenomena can basically be one and the same for loading sequence, whereas the differences in material formulations are decisive in the unloading sequence. In the non-linear elastic case, the stress-strain diagram for unloading runs along the loading path and after full unloading a strain-free state is reached and the new cycle is identical to the first one. In case of the elasto-plastic material model, the unloading sequence runs parallel to the initial rate *E* and after full unloading the structure is not strain-free because plastic strains remain. The next cycle is therefore different from the first one. As opposed to that, upon unloading in the case of elasto-damage model, no permanent strains remain. Unlike the linear-elastic model, the unloading sequence does not follow the loading path but it runs linearly to the diagram origin. A new load introduction is influenced by degradation of stiffness with the damage parameter *d*, due to which the repeated load cycles are not identical in their effects on the material and the structure.

5.3 Geometrical non-linearity

The following will be used:

* (total) Lagrange point of view which is also described as the material point of view
* Stress and strain quantities in the undeformed configuration (second Piola Kirchhoff stress tensor and Green Lagrange strain tensor)

Kinematics

The fundamental of the geometrically non-linear formulation of structural mechanics is based on the *material deformation gradient* ***F***, whose formulation is shown earlier (chapter 2 and 3).

In non-linear observations the material deformation gradient defines the transformation from reference to current configuration or from undeformed to deformed state and vice versa. These transformations are referred to in technical literature as push forward and pull back. The material deformation gradient is defined by a transformation of a line element *dX* of the reference configuration to the current configuration *dx*.

***dx = F.dX F = = x (5.6)***

The Green Lagrange strain tensor *E* was also already derived and is given in equation (3.12) as function of the displacement gradient ∇*u* and its transpose T*u*. If we describe the motion of the material point from the reference to the current configuration with help of the displacement vector *x = X + u*, the Green Lagrange strain tensor

***E =* [*FT.F -* 1] = *u* + T*u*.∇*u =* [*u* + *u* + *u* . *u*] *(5.7)***

can be represented as a function of the material deformation gradient.

***F = (X+u) =* 1+ ∇*u (5.8)***

From Eqs. (3.13) and (3.14) we have

**=  *(5.9)***

And from Eq. (3.7) we have

**∇*u* =  *(5.10)***

So from matrix multiplication we get

**T*u*.∇*u = (5.11)***

And the summation is performed over k = 1, 2, 3, respectively. The component presentation of the Green Lagrange strain tensor finally yields the following:

**= ( + + ) *E = \* (5.12)***

To be able to calculate in finite element method, convert the equation into vector form, i.e.

***E(u) = = + (5.13)***

According to Eq. (5.13), the Green Lagrange strain tensor of geometrically non-linear deformations is obtained by addition of the well-known linear part *u* and the *non-linear part*

**(*u*)**,

***E(u) = u +* (*u*)  *(5.14)***

Formulation of Dirichlet boundary conditions

***u*(*X*) = *u*\*(*X*) *X (5.15)***

and of initial conditions remains unaltered compared to linear structural mechanics

***X (5.16)***

Kinetics

Unlike the linear structural analysis, its non-linear counterpart requires that the dynamic or the static forces equilibrium be observed in the deformed configuration. This firstly calls for the evaluation of mass distribution which gives the relation between density in the current configuration c and the one in the reference configuration , with the help of determinant of the material deformation gradient

**= c  *(5.17)***

The forces equilibrium of a geometrically linear approximation, Eq. (3.25), was obtained by pure kinetic analysis of a differential volume element. Analogous analysis of a volume element in a deformed configuration gives us the Cauchy motion equation in the so called spatial or Euler formulation.

***ü =* div *+ b*  *X (5.18)***

*div* symbolises the tensor divergence of the real stresses or the Cauchy stress tensor, related to the current configuration. As the first description of this stress tensor might lead us to anticipate, this stress quantity is defined by a differential load in the current configuration, effecting a deformed surface element *da*, arbitrarily oriented with a normal vector *n* inside the body, leading to the consequence that the actual stresses occurring in the material can be described.

It is of advantage to numerical conversion in structural mechanics to utilize the motion equation in the material or Lagrange formulation. To perform this it is necessary to relate the Cauchy motion equation to the undeformed configuration. Multiplication of motion equation (5.18) by determinant of the material deformation gradient already transforms the density in the reference configuration, according to (5.17).

***ü =* div *+ b ü =* div *+ (5.19)***

Transforming the middle term the resulting identity in symbolic presentation and it component notation is

**div = DIV(*F.S*) = =  *(5.20)***

DIV symbolises the *divergence operator with respect to the reference configuration*. Tensor *S*, which is here used for the first time, is the second Piola Kirchhoff stress tensor defined with respect to the reference configuration. It should be noted that the Piola Kirchhoff stress tensor *S*, unlike the Cauchy stress tensor , does not refer to actual stresses but to 'pseudo stresses'. They are defined with respect to the reference configuration by a differential load effecting a surface element of the reference configuration *dA*, which is oriented with a normal vector *N*. Introduction of equation (5.20) into equation (5.19) eventually yields the material or the Lagrange formulation of impulse rule, that is, the Cauchy motion equation.

***ü =* DIV(*F.S*) *+*  *X (5.21)***

In order to formulate a well-defined problem, it is necessary to supplement the impulse rule with the static or Neumann boundary condition. Applying the procedure for derivation of equilibrium at the boundary, demonstrated in the geometrically linear case, to the deformed configuration

**=**   ***x*** ***(5.22)***

and thereupon to transform it. Material formulation of the Neumann boundary conditions is in the geometrically non-linear case defined by

***F.S.N =***  ***X (5.23)***

where *N* presents the normal vector of reference configuration. Stress vector (first Piola Kirchhoff stress tensor) is defined by a differential load vector of current configuration acting on a surface element of the reference configuration *dA* parallel to .

***dA = da (5.24)***

Constitutive Law

For moderate strains, the Saint Venant Kirchhoff material model is used to project the Green Lagrange strains on to second Piola Kirchhoff stresses in analogy with linear structural mechanics.

***S = C* : *E C = 2μI +* 1\*1  *(5.25)***

Principle of virtual displacements

Weak formulation of initial value problem

Weak formulation of the initial value problem of geometrically non-linear structural mechanics can be obtained in a way analogous with linear formulation, by choice of displacement vector transformation *u* as special test function, and by applying the calculation rule (3.77) to

***u.*DIV(*F.S*),= DIV(*u*(*F.S*)) - *u*:(*F.S*)  *(5.26)***

where the volume integral of the first right-hand side term DIV(*u*(*F.S)*) can be transformed to upper surface integral via *u*.*F.S.N*, by applying the Gauss integral law. With the help of mentioned transformations we get the virtual displacements principle.

.***ü +*** :**(*F.S*) =** . **+ *(5.27)***

*dV* and *d* describe the volume element, that is, the line element in the reference configuration.

First term in equation (5.30) is the virtual work of inertial forces *Wdyn*, the second term is the virtual work*Wint*. The sum of the third and the fourth term makes up the virtual work of external loads *Wext*. Internal virtual work in equation (5.27) should be transformed in such a way that one obtains the form of virtual displacements equivalent to equation (3.84). To get this form, we first vary the term *E*:*S* according to definition of *E* in equation (5.7), where in order to achieve further transformation we use the symmetry of the stress tensor

***E*:*S =* (*FT.F –* 1) :*S =* (*FT.F + FTF*):*S = FTF*:*S = F*: *F. S (5.28)***

With the definition of the material deformation gradient in equation (5.6) and the unchanging coordinates of the reference configuration (*X* = 0),

***F =* (*X + u*) = = *u (5.29)***

The virtual displacement principle with equations (5.27), (5.28) and (5.29) in the form preferred for further derivations (in analogy with equation (3.84) of geometrically non-linear structural mechanics) is written as

.***ü +*  =** . **+ *(5.30)***

It may be noticed once again that the Lagrange formulation of the impulse rule is used, and as a consequence, the integration over the volume, that is, over the boundary of a material body has to be performed in the reference configuration. From equation (5.30), it can be concluded that virtual work of inertial forces *Wdyn*, and of external loads *Wext*, did not undergo any formal change in comparison with linear observations. This practically means that for isoparametric finite elements calculation of the mass matrix *me*, and of the kinematically equivalent loads or the consistent element load vectors and , can be inferred from the linear formulation. This claim is valid for structural elements only in special cases due to rotational degrees of freedom, used to describe kinematics and deformation (displacement-based description of rotation parameters, isoparametric shear susceptible elements), and needs to be discussed next.

As opposed to the terms of virtual work of inertial forces and of external loads, the internal virtual work *Wint* for geometrically non-linear observations is crucially different. Instead of variation of Green strain tensor , comes the Green Lagrange strain tensor *E*, given in equation (5.7) or equation (5.14). The Cauchy stress tensor has to be replaced with the second Piola Kirchhoff stress tensor *S* which is linked with the Green Lagrange strain tensor through equation (5.25) in the materially linear case.

***Wint* = = =  *(5.31)***

Variation of Green Lagrange strains

Equation (5.31) contains the variation of Green Lagrange strain tensor . Before deriving this variation, first review the geometrically non-linear theory. In case of geometric non-linearity, the Green strain vector could be computed with the help of the deformation independent differential operator and the displacement vector *u*, = *u* (3.16). Accordingly, the variation of Green strain vector was obtained by variation of the linear mapping just mentioned =*u*. The corresponding relation should be derived as preparation for the variation of Green Lagrange strain vector, needed to discretize and formulate finite elements in chapter 6.

= ***T  (5.32)***

Variation of strain components *Eij* with *i, j* = 1, 2, 3 can be calculated as follows with equation

(5.12)

**= ( + + )  *(5.33)***

and with the exchangeability of sequence of variation and partial derivatives when applying the product rule.

**= ( + + + )  *(5.34)***

which is summated over *k =* 1, 2, 3. Setting *i, j* = 1, 2, 3 we get the respective strain components.

**= +**

**= +**

**= +**

**= + + +**

**= + + +**

**= + + +**  ***(5.35)***

Putting them in matrix form we get

=

***(5.36)***

The Green Lagrange strain vector is composed by addition of the constant part already defined in equation (3.16)

**=  *(5.37)***

and of the deformation-dependent part.

**=**  ***(5.38)***

The connection between variation of continuous displacements and variation of the Green

Lagrange strain tensor can be demonstrated with assistance of the differential operator and , where the product stands for the variation of the non-linear strain vector term .

**= + = +  *(5.39)***

Internal virtual work

With developments of strains and their variation in linear and non-linear parts, now transform the internal virtual work according to equation (5.31). Parts of Green Lagrange strain vector *E* that arise and their variation*E* are described in equations (5.13) and (5.39).

***Wint* = =  *(5.40)***

Remarks regarding combined material and geometric non-linearity

If one should combine the non-linearities dealt with in this and the previous section, namely material and geometric non-linearity, one should note that non-linear material models are formulated in true, that is in Cauchy stresses but also in strains (Euler strain tensor) related to current configuration. For this reason, the actual Green Lagrange strain tensor generally has to be related to the momentary configuration with a push forward. There, the Cauchy stress tensor is computed with the help of the used material model and thereafter related to the reference configuration with a pull back. The second Piola Kirchhoff stress tensor, which is the result of this procedure, is thereby a function of the Green Lagrange strain tensor, of the material deformation gradient (push forward and pull back) and of internal variables dependent on the material model.

***S = S*(*E,F,*) *(5.41)***

Compared to a pure material non-linearity, the reference configuration related stress and strain quantities have to be utilized here instead of and as well as the material deformation gradient.

5.4 Consistent Linearization of internal virtual work

The basis for the solution of geometrically non-linear finite element systems is the linearization of internal virtual work. Before the linearization is performed in the next chapter 6, the directional or the Gateaux derivative of a scalar or a vector need to be defined.

Linearization background

Gateaux derivative

*Gateaux derivative* of a scalar, vector, matrix or a tensor will henceforth be designated with a symbol. The definition is given with the help of an arbitrary scalar function *f*(*u*).

***f*(*u*) = = .  *(5.42)***

In this definition, *u* stands for the actual displacement vector and *u* stands for an incremental change of *u*. is the increase or the derivative of function *f* along a direction or a straight line set by . If this derivative is evaluated at = 0, we get the function *f* derivative of the displacement state *u* in direction . On the other hand, the scalar product of the normal vector defined by *∂f(u)/∂u* gives, along with the vector of incremental displacement , the change *f* of function f for the change in a tangential plane at *f(u)*.

Gateaux derivative of internal virtual work

Application of Gateaux derivative definition (5.42) to the internal virtual work *W*int, according to equation (5.40) with variation of the strain tensor according to equation (5.39), determines the linearization of internal virtual work.

***Wint = =***

***(5.43)***

Gateaux derivative is assembled by differentiation with respect to scalar and by applying the chain rule and evaluating at = 0.

***Wint =***

**+**

**=**

**+  *(5.44)***

If we take into consideration that, according to the properties formulated in section 3.6, the virtual displacement is arbitrary is independent of the displacement state *u*, making = = 0, we get the Gateaux derivative of internal virtual work,

***Wint =***

**+  *(5.45)***

which can further be transformed after introduction of equation (5.39) applied to the term .

***Wint* =**

**+ *(5.46)***

This equation represents the Gateaux derivative or the directional derivative of internal virtual work related to the incremental change of displacement state by . The Gateaux derivative is also called the *consistent linearization* of internal virtual work due to its strict mathematical derivation. This consistent linearized virtual work represents a *milestone* of non-linear structural mechanics since it is the basis of all iterative, incremental solution strategies.

We should notice that represents the linearization of the Green Lagrange strain vector *E*(*u*) and that the Gateaux derivative of the second Piola Kirchhoff stress vector for materially non-linear models is given by

**= = C *E(u) (5.47)***

We should further notice the identity of linearization of the Green Lagrange strain vector variation and its non-linear part

***E(u) = = (5.48)***

due to the vanishing linearization of the linear part. Consequently we can write the linearized internal virtual work (5.46) in a compact form as

***Wint* =  *(5.49)***

In order to discretize the directional derivative or linearize the internal virtual work, according to equation (5.46) or (5.49), the linearization of the non-linear part of the Green Lagrange strain vector and its variation needs to be determined. The non-linear part of the Green Lagrange strain vector and differential operators and  are already given in equations (5.14), (5.38) and (5.39).

Linearization of Green Lagrange strains

When using a linear material law, the linearization of the Green Lagrange strain vector, according to equation (5.45), multiplied by the material matrix is equivalent to the linearization of the second Piola Kirchhoff stress vector. Investigating the non-linear part since the linear part is already known to be ,

***E(u) = +***   ***= (5.50)***

The application of the Gateaux derivative to in its indexed notation yields

**=**

**= *(5.51)***

By comparing the above equation with the variation of the corresponding strain component in equation (5.34), we see the equivalence of variation and linearization of strains. As a consequence, we conclude that linearization of the Green Lagrange strain vector is given directly with equation (5.39), where the variation symbol needs to be replaced by the linearization symbol .

***E(u) = + =* ( *+* )  *(5.52)***

Linearization of variation of Green Lagrange strains

The linearization of variation of Green Lagrange strains follows from considering equation (5.48) based on equation (5.34) with the help of the Gateaux derivative in components. We can use the indexed form or the tensor form, and applying that for the linearization of the virtual displacement = 0, to get eventually get respectively

***= = (5.53)***

**= [ . + ]  *(5.54)***

For the following discretization of internal virtual work in a pure and linearized form in chapter 6, some additional considerations and transformations need to be done beforehand. The linearized variation of Green Lagrange strains appears in the linearized internal virtual work (equation (5.49)) as a scalar valued product with the second Piola Kirchhoff stresses. In tensor notation, this can be written as follows and transformed and summarized due to the symmetry of the stress tensor.

=  **[ .] : + [ ] :**

*=* **[ .] : = [ ] : =  *(5.55)***

The proof of equation (5.55) comes in components, where symmetry of the stress tensor = and exchangeability of dummy (summation) indices enable the particular steps of the proof.

***=*  []**

**= [] = []**

***=*  [] =  *(5.56)***

After this, it remains to specify the expression (analogy with calculation of , given in equation (5.11))

**[T*u*.∇*u*]:*S =*  :  *(5.57)***

and to transfer it into a suitable form in matrix notation.

**[T*u*.∇*u*]:*S* = .  *(5.58)***

Here, defines a vector component of the displacement vector gradient *u*.

**= *T***

***(5.59)***

**= *T***

**= = = =  *(5.60)***

and defines the hyper-diagonal matrix of the second Piola Kirchhoff stress components.

**= , = *(5.61)***

The validity of identity (5.58) can be examined by calculation of the corresponding scalar according to both left and right side of this equation, and also by applying definitions (5.59) and (5.61). In total the identity relevant to discretization of linearized internal work is obtained with equations (5.55) and (5.58).

= **.  *(5.62)***

**Chapter 6: Finite Element Discretization of 1-D Geometrically Non-Linear Continua**

The finite element discretization of the weak form of impulse balance, or of the principle of virtual work (5.30), gives the static or dynamic equilibrium in the form of a non-linear vector equation or vector differential equation. For the finite elements dealt with in this chapter, the latter differs from the discrete formulation of the principle of virtual work for small deformations (geometrically linear observation) only in the term of discretized internal virtual work. The result of discretization of this different term is the deformation-dependent vector of internal forces.

Besides the discretization of principle of virtual work, when considering definite deformations (geometrically non-linear), the discretization of linearized internal virtual work (5.46) or (5.49) is of crucial significance for numerical solution of geometrically non-linear elasto-mechanics.

Discretization of the linearized internal virtual work defines the so called *tangential stiffness matrix*. This tangential stiffness, identical to linearization of vector of internal forces, forms the basis of iterative Newton procedures on the one hand, and is of importance for characterization of stability properties of the structure on the other hand.

6.1 Non-linear continuum-mechanical formulation for Finite truss elements

Kinematics

The Green Lagrange strain in the longitudinal direction of the truss *E11* is determined based on equation (5.12) that is (5.13) with *=* = 0.

**= ( + + ) = +  *(6.1)***

The presentation of the Green Lagrange truss strain, analoguous with the presentation of the differential operator (5.14) can be acquired by the following transformation.

***=*  + = *+* ()  *(6.2)***

Kinetics

The Lagrange formulation of the forces equilibrium of a truss element is obtained with equation

(5.21) for = = = = = 0, and the vanishing derivatives in and directions (divergence DIV).

***=* () *+*   *(6.3)***

Constitutive law

The one-dimensional special case of the constitutive equation (5.25) can be described only with the assistance of the elasticity modulus.

**=  *(6.4)***

Principle of virtual work

The principle of virtual displacements of a one-dimensional continuum is obtained by reduction of the three-dimensional formulation (5.30), or by adjusting the linear formulation (4.24). In case of the latter procedure, the Green strain and the Cauchy stress have to be replaced by the Green Lagrange strain and the second Piola Kirchhoff stress , respectively, and the non-specified length *l* by the reference length *L*.

***A*  + = + +  *(6.5)***

The inertial term as well as the terms of internal and external loads is the same as in the linear observation. For the internal virtual work, the integrand is already known, with the exception of the variation of the Green Lagrange strain. is obtained by reduction of equation (5.34) for the one-dimensional continuum or by variation of equation (6.1).

**=** +  **+ = +  *(6.6)***

This can be presented by means of analogy to equation (5.39) with the help of the differential operator and .

**= + = +  *(6.7)***

Linearization of internal virtual work

The linearized internal virtual work can be extracted from the internal virtual work of the three-dimensional continuum (5.46), or much simpler, by linearization of internal virtual work given in equation (6.5).

***Wint* =  *(6.8)***

In this expression, we need to generate the linearization of variation of the Green Lagrange strain and of the second Piola Kirchhoff stress. The first linearization yields

**= + = = + =  *(6.9)***

The linearization of the second Piola Kirchhoff stress can be put down to the linearization of the Green Lagrange strain with the help of the constitutive equation (6.4), which is defined by the equivalence of variation and linearization in analogy to equation (6.7).

**= = +  *(6.10)***

For the discretization of linearized internal virtual work of the three-dimensional continuum, it is favourable to properly transform the expression equivalent to . For a deeper understanding of this transformation, we should do it for the one-dimensional continuum. Since and are scalar expressions, this transformation is of no further significance.

**= = = =  *(6.11)***

6.2 Truss elements of arbitrary polynomial degree

In order to develop a family of truss elements for the modelling of geometrically non-linear structural behaviour, we should assume equidistant element nodes. The development is separated into determination of the element vector of internal forces by discretization of the internal virtual work, and generation of the tangential element stiffness matrix based on linearized internal virtual work. Afterwards, the derivation of the special case of the linear truss element is done.

Element vector of internal forces

The element vector of internal forces is, developed with the discretization of virtual displacement in the internal virtual work expression (6.5). The basis of standardized discretization is the formulation of internal virtual work in natural coordinates, where the Jacobi matrix and the Jacobi determinant are identical to formulations of geometrically linear truss elements (section 4.2), as long as they are formed with the reference truss length L.

***Wint = = (6.12)***

The differential operators and as well as the stress are formed with the Jacoby transformation of corresponding values in physical coordinates (see equation (4.38)).

**= =**

**= = *(6.13)***

**= *E*[] = []**

The discretization of virtual displacement is as done in section (4.2), by means of which also the derivation of displacement with respect to natural coordinate can be computed for an approximate displacement state .

**=**

***(6.14)***

**= ()**

Insertion of equation (6.14) into equation (6.12) yields the approximation of internal virtual work.

***Wint***

**= =  *(6.15)***

Here, the approximation of the Green Lagrange strain was accomplished with the help of the B-operator and and the functional dependence on the displacement vector was specified as consequence of equation (6.14).

**=  *(6.16)***

By transformation of equation (6.15), the internal forces vector

**=  *(6.17)***

can be developed with the differential operator of geometrically linear truss elements (generalization of equation (4.48)) and the non-linear deformation-dependent differential operator

**= () = ()  *(6.18)***

with the second Piola Kirchhoff stress in natural coordinates being computable with the help of equations (6.13) and (6.14) for a prescribed displacement state .

Tangential element stiffness matrix

By discretization of linearized internal virtual work (6.8) in natural coordinates,

***Wint* =  *(6.19)***

the tangential stiffness matrix of hierarchically generated truss elements is derived. From the

terms in equation (6.19), is already known (equations (6.13) und (6.14)) and is already derived in equations (6.12) and (6.13) and discretized in equation (6.16). The approximation of can be done with the help of the constitutive law (6.4) and with the equivalence of variation and linearization based on equation (6.16).

**= = *(6.20)***

is derived from (6.9), by applying the Jacobi transformation (see (6.13)).

**= = =  *(6.21)***

The approximation of term is performed with the help of differential operator (), which is the special case of one-dimensional elements corresponds to the differential operator of geometrically linear truss elements. This can be shown with discretization of incremental displacement and the virtual displacement in equation (6.21).

**=  *(6.22)***

Insertion of equations (6.16), (6.20) and (6.22) into equation (6.19) yields the approximation of the linearized internal virtual work

***Wint*  *(6.23)***

with the deformation-dependent geometric element stiffness matrix ( = )

**=  *(6.24)***

and the deformation-dependent material element stiffness matrix

**=  *(6.25)***

Linearization of the element vector of internal forces is also determined with equation (6.23)

**=  *(6.26)***

6.3 Linear truss element

The detailed elaboration of the two-noded linear finite truss element, as part of the family of hierarchically generated finite truss elements, presented in section 6.2, is performed by reduction of element vectors, differential operators, matrix of shape equations and element matrices. The definition of the matrix of shape equations N(), the element displacement vector , as well as the differential operator B() = B can be inferred from derivations dealing with the linear theory in section 4.2 (equations (4.40), (4.41)) and (4.48)). The non-linear part of the B-operator = () is calculated according to (6.18) by introducing the derivatives of shape equations = and = according to equation (4.39).

**= = =  *(6.27)***

with the approximation

**= =  *(6.28)***

being constant in . When generating the element vector of internal forces according to (6.17), especially when integrating, it is advantageous that, due to the linear displacement shape equations, the B operators (6.27) and the normal stress ((6.13) are independent of the natural coordinate .

**=  *(6.29)***

The use of differential operators according to equation (6.27) in the end yields the element vector of internal forces of the two-noded truss element

**=  *(6.30)***

By executing the products of matrices in equations (6.24) und (6.25) with differential operators of the two-noded truss element (6.27) we get the geometric element stiffness matrix

**=  *(6.31)***

and the material element stiffness matrix

**=  *(6.32)***

*For = 0, the material element stiffness matrix equals the elastic element stiffness matrix*

*given in equation (4.53).*

**Chapter 7: Computational Simulation**

A Program for the Elasto-Plastic analysis for plane stress, plane strain and axi-symmetric solids was used. The program was collected from the text book "Finite Elements in Plasticity" by D R J Owen & E Hinton - Swansea UK. A code has also been written for developing an input file from ABAQUS so that complex geometries can be handled more easily.

This is written for elasto-plastic deformation considering material strain hardening and used small strain theory. By this we can solve plane stress, plane strain, axis-symmetric type of problems. It incorporates three types of elements and they are Linear quadrilateral element which consists of four nodes, Quadratic serendipity element which consists of eight nodes, Quadratic lagrangian element which consists of nine nodes.

Here in this code four basic Non-linear solution parameter and four yielding parameters are used and they are:

1. Initial stiffness method: - The element stiffnesses are calculated at the beginning of the solution process and remain unchanged thereafter.
2. Tangential stiffness method: - The element stiffnesses are recalculated for every iteration of each load increment.
3. Combined algorithm (Version I):-The element stiffnesses are recalculated for the first iteration of each load increment only.
4. Combined algorithm (Version II):- The element stiffnesses are recalculated for the second iteration of each load increment only.

Yield criterion parameter:

1. Tresca yield criteria.
2. Von Mises yield criteria
3. Mohr-Coulomb yield criteria.
4. Drucker-Prager yield criteria.

It also demands seven material properties and they are young’s Modulus of elasticity *E*, Poisson’s ratio *ν*, material thickness *t* (only for plane stress problems), mass density *ρ*, uniaxial yield stress *σy*, (or cohesion *c* for Mohr-Coulomb and Drucker-Prager), the strain hardening parameter *H*' and friction angle *Φ(measured in degree)*  for Mohr-Coulomb and Drucker-Prager.

Three types of loads can be entered, that is point loads, gravity and edge distributed load. In case of point load the direction in which point load is applied has to be given while in case of axis-symmetric case the total loading on the circumferential rings passes through the node has to be given. In case of gravity load, if it exists, the angle *θ* (it is the angle made by gravity axis by *y* axis) and the magnitude of gravity constant *g* needs to be specified. While in edge distributed load, if it exist, the total number of edge elements at which the load is distributed, the element numbers with which these edges are connected and nature of the loading over the edge i.e. tangential component or normal component or both needs to be specified. Here in this program the load applied is multiplied by a factor which gives accumulative effect.

Problem 1:

Problem solved using Program for the Elasto-Plastic analysis collected from text book "Finite Elements in Plasticity" by D R J Owen & E Hinton - Swansea UK. The problem studied is that of a thick cylinder of 200 mm bore and 400 mm outside diameter subjected to a gradually increasing internal pressure. The material properties of the cylinder are: Elastic modulus. E= 2.1 x 104 dN/mm2, Poissons ratio, *ν* = 0.3, Uniaxial yield stress. σy,= 24.0 dN/mm2, Strain hardening parameter, H' = 00.

Solution:

The above stated problem was solved treating it as a plane strain problem in Z direction and Von mises Yield criteria. Due to axis-symmetric geometry and loading situation only quarter part is discretised here using 12, 8 noded quadratic serendipity element and 51 nodes.

X

Y

100mm

200mm

1

2



4

12

10

Element No.

Nodes

51

500

49

48

47

46

45

44

7

29

1

40

18

41

33

**P**

**Fig.1**

**Fig.2**

**Fig.3**

**Fig.4**

*Figures1, 2, 3, 4 depict the Hoop stress distribution at incremental pressure along the cylinder thickness*

**Fig.5**

**Fig.6**

**Fig.7**

**Fig.8**

*Figures 5, 6, 7, 8 depict the Radial stress distribution along the thickness of the cylinder*

*In all of the above formulated outputs we have taken 2 Gauss Quadrature points and plotted the graphs for 4 different pressures. The trends of the above graphs were compared with the formulation given in Owen and Hinton and were found to be in agreement, thus verifying our result.*

Problem 2:

Problem solved using Program for the Elasto-Plastic analysis collected from text book "Finite Elements in Plasticity" by D R J Owen & E Hinton - Swansea UK. The problem studied is of a plate of 100mm 100mm unit thickness subjected to a gradually increasing pressure and small amount of unloading is also required. The material properties of the plate are Elastic modulus. E= 2.1 x 104 dN/mm2, Poissons ratio, *ν* = 0.3, Uniaxial yield stress. σy = 24.0 dN/mm2, Strain hardening parameter, H' = 10.

100 mm

100mm

P

X

Y

Solution:

The above stated problem was solved treating it as a plane strain problem and Von mises Yield criteria. Due to axis-symmetric geometry and loading situation only quarter part is discretized here using 400, 8 node quadratic serendipity element and 1281 nodes. The input file for the program was collected from ABAQUS and modified according to our. The schematic of the problem is shown as above. The plate is loaded in the steps of 10, 20, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 37.5 and 38 kN/mm2 and the result of the program is shown below in the form of graph.

100 mm

100mm

P

2

1

4

3

14

13

16

15

Nod

**Fig.9**

**Fig.10**

*The trends of the graphs (figures 9 & 10) were again compared with the formulation given in Owen and Hinton and were found to be in agreement, thus verifying our result.*

**Conclusion**

During the course of the thesis my main emphasis was on understanding the formulation of nonlinear elastic continuum mechanics. The theory was developed for general formulation and eventually simplified to the simple one dimensional truss case.

Along with the above theoretical coursework a finite element computational code was also used from an established source to justify the theory. The results from two different models were taken, and the output was plotted on graphs. The trends found on these graphs were a good match to the theoretical predictions given. They also showed the nonlinear region as expected.

**BIBLIOGRAPHY**

1. K.J. BATHE, FINITE ELEMENT PROCEDURES-PHI, *1996*.
2. D. KUHL & G. MESCHKE-RUHR UNIVERSITY, LECTURE NOTES ON FINITE ELEMENT METHODS, *2005.*
3. D.R.J. OWEN & E. HINTON, FINITE ELEMENTS IN PLASTICITY-SWANSEA PRESS, UK, *1980*
4. J. N. REDDY, AN INTRODUCTION TO CONTINUUM MECHANICS WITH APPLICATIONS-CAMBRIDGE UNIVERSITY PRESS, *2008*
5. J. BONET & R.D. WOOD-SWANSEA, UK, NONLINEAR CONTINUUM MECHANICS FOR FINITE ELEMENT ANALYSIS, CAMBRIDGE UNIV PRESS, *1997*
6. M.E. GURTIN, E. FRIED & L. ANAND, THE MECHANICS AND THERMODYNAMICS OF CONTINUA, *2004*
7. M. H. SADD, INTRODUCTION TO FINITE ELEMENT METHODS (PPT), *2005.*