Engineering

Software

Solutions

General specifications on Frame 3D Library

Basic Procedure of the Stiffness Method

Use a two-member spring example to illustrate these steps.

- Assign a coordinate system for the structure. Assign node numbers for the structure.
- Define degrees of freedom for the structure and assign numbers for them.
- Assign numbers to the members of the structure.
- Break-up the structure into smaller pieces called elements or members. Define element nodal displacements and forces.
- Identify boundary conditions for the structure in terms of displacements.
- Write compatibility conditions between the structural nodal displacements and element nodal displacements for each member.
- Identify external loads for each degree of freedom
- Write equilibrium equations for each element in terms of displacements. kd = f
- Combine elements equilibrium equations to form equilibrium equations for the entire structure. KD = F
- Apply boundary conditions to the system equilibrium equations and solve for the system nodal displacements.
- Finally, consider equilibrium equation for each element and solve for the element nodal forces.

Direct Stiffness Method for Truss Analysis

- A truss is a structural system that satisfies the following requirements:

a. The members are straight, slender, and prismatic. The crosssectional dimensions are small in comparison to the member lengths. The weights of the members are small compared to the applied loads and can be neglected. Also when constructing the truss model for analysis, we treat the members as a onedimensional entity (having length and negligible cross-sectional dimensions).

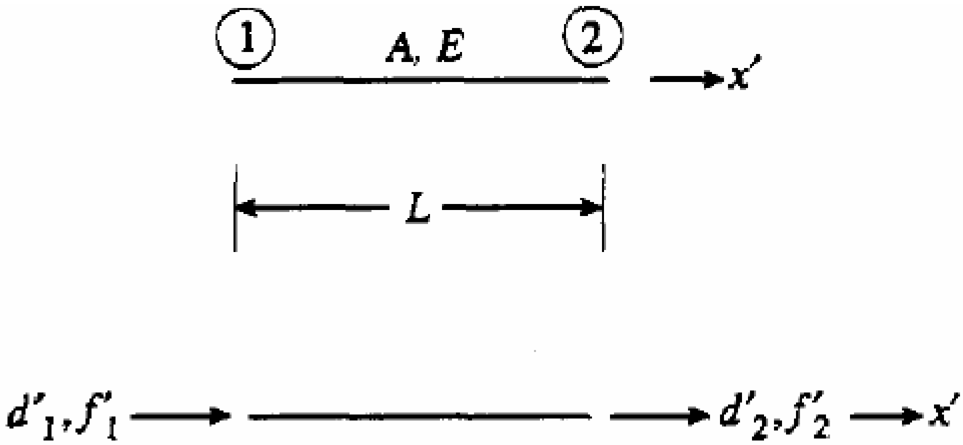
b. The joints are assumed to be frictionless pins (or internal hinges).

c. The loads are applied only at the joints in the form of concentrated forces.

As a consequence of these assumptions, the members are two-force

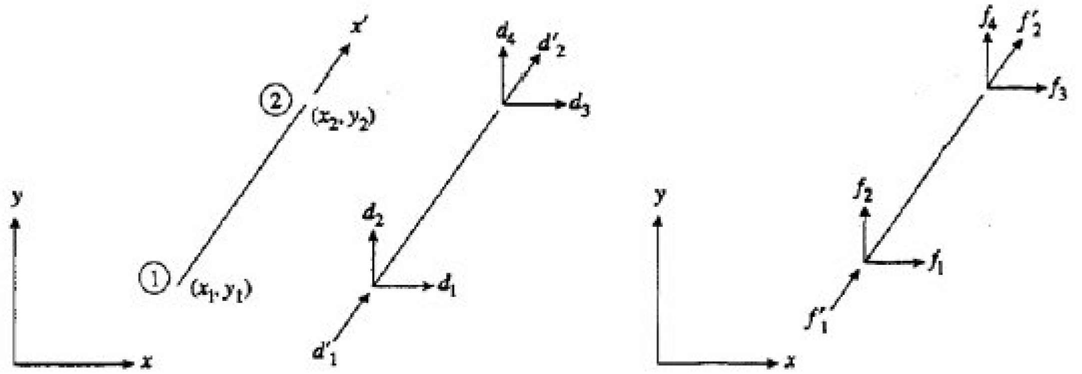
members, meaning that they carry only axial forces. In very many
ways, a truss member is quite similar to the typical linear spring. Two
nodes define a typical truss element.

- For a two-node truss element shown below, governing equations with respect to the local coordinate system x Œ can be expressed as:



$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d'_1 \\ d'_2 \end{Bmatrix} = \begin{Bmatrix} f'_1 \\ f'_2 \end{Bmatrix} \quad \text{or} \quad \mathbf{k}'_{2\times2}\boldsymbol{d}'_{2\times1} = \boldsymbol{f}'_{2\times1}.$$

- From the transformation matrix between the local and global
- coordinate systems shown below, the relationship between the
- local nodal displacements and global nodal displacements is
- derived as



$$\begin{Bmatrix} d_1' \\ d_2' \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{Bmatrix} \quad \text{or} \quad \boldsymbol{d}'_{2\times1} = \boldsymbol{T}_{2\times4}\boldsymbol{d}_{4\times1}.$$

The corresponding nodal force relationship reads

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix} = \begin{bmatrix} l & 0 \\ m & 0 \\ 0 & l \\ 0 & m \end{bmatrix} \begin{Bmatrix} f_1' \\ f_2' \end{Bmatrix} \quad \text{or} \quad \boldsymbol{f}_{4\times1} = \boldsymbol{T}^{\mathrm{T}}_{4\times2}\boldsymbol{f}'_{2\times1}.$$

The global governing equations for a truss element are therefore written as

$$\frac{AE}{L}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix} \quad \text{or} \quad \mathbf{k}_{4\times4}\boldsymbol{d}_{4\times1} = \boldsymbol{f}_{4\times1}.$$

kij = force required in the direction of dof i to produce unit displacement along the dof j.

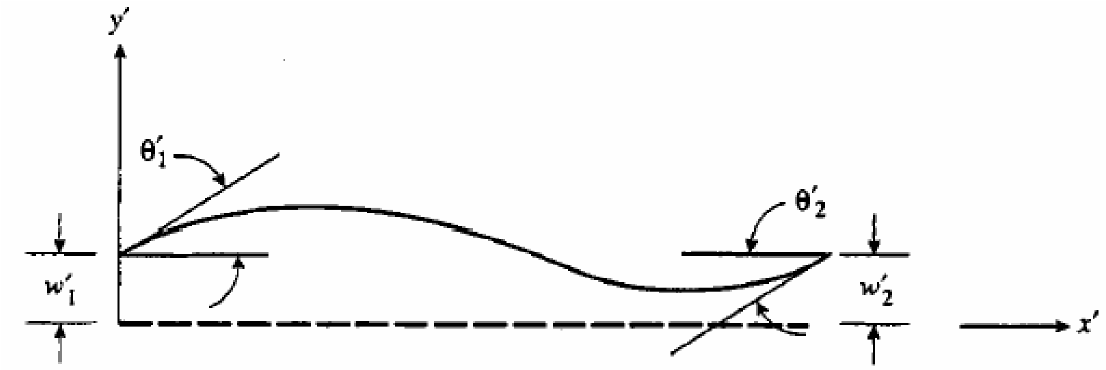
The major steps in solving planar truss problems using the direct stiffness method:

Step 1: Select the problem units. Set up the coordinate system. Identify and label the nodes and the elements. For each element select a start node (node 1) and an end node (node2). We use an arrow along the member to indicate the direction from the start node to the end node. This establishes the local coordinate system for each element. Label the two global dof at each node starting at node 1 and proceeding sequentially.
Step 2: Construct the equilibrium-compatibility equations for a typical element.
Step 3: Using the problem data, construct the element equations from Step 2 for all the elements in the problem.
Step 4: Assemble the element equations into the system equations, 2 2 2 1 2 1 j j j j X X X = k d f where j is the number of joints in the truss.
Step 5: Impose the boundary conditions.
Step 6: Solve the system equations KD = F for the nodal displacements D.
Step 7: For each element using the nodal displacements, compute the element nodal forces.

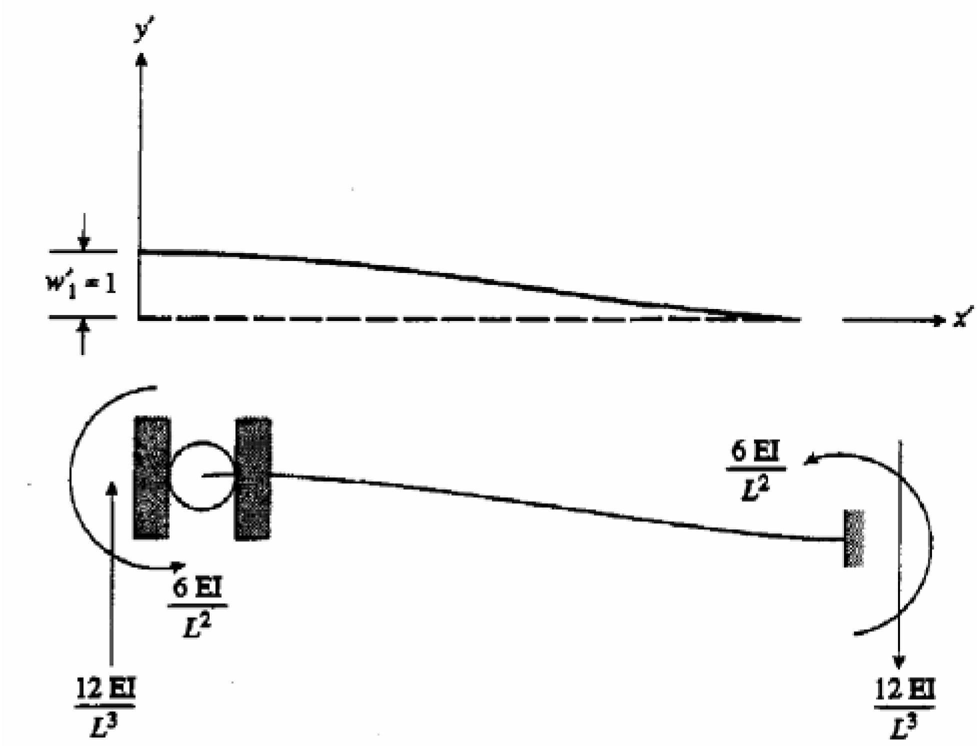
Direct Stiffness Method for 2d Frame Analysis

- A planar frame is a structural system that satisfies the following requirements:
  - a. The members are slender and prismatic. They can be straight or curved, vertical, horizontal, or inclined. The cross-sectional dimensions are small in comparison to the member lengths. Also when constructing the frame model, we treat the members as one-dimensional entities (having length and negligible crosssectional dimensions).
  - b. The joints can be assumed to be rigid connection, frictionless pins (or internal hinges), or typical connections.
  - c. The loads can be concentrated forces or moments that act at joints or on the frame members, or distributed forces acting on the members.

It is assumed that the frame is made of straight members and that the connections are rigid. We develop the element capable of modeling a planar frame in two stages. In the first stage, the flexure effects (due to shear force and bending moments) will be considered. In the second stage, the axial effects will be considered. Using the superposition principle, we can then construct the behavior of a frame element. As before, the superposition is valid only if the displacements are small. In structural analysis terminology, members that are subjected primarily to flexural effects are said to be beams whereas members with combined axial-flexural effects are called beam-columns. By the end of this section we will have developed the element equations for the combined effects that can also be used to model pure beam behavior. To avoid construction and usage of several different terminology, we refer to this element simply as the beam element. The planar beam element displacements (degrees of freedom) in the local coordinate system (x'-y')



For a two-node planar frame element shown below, governing equations with respect to the local coordinate system x' expressed as:



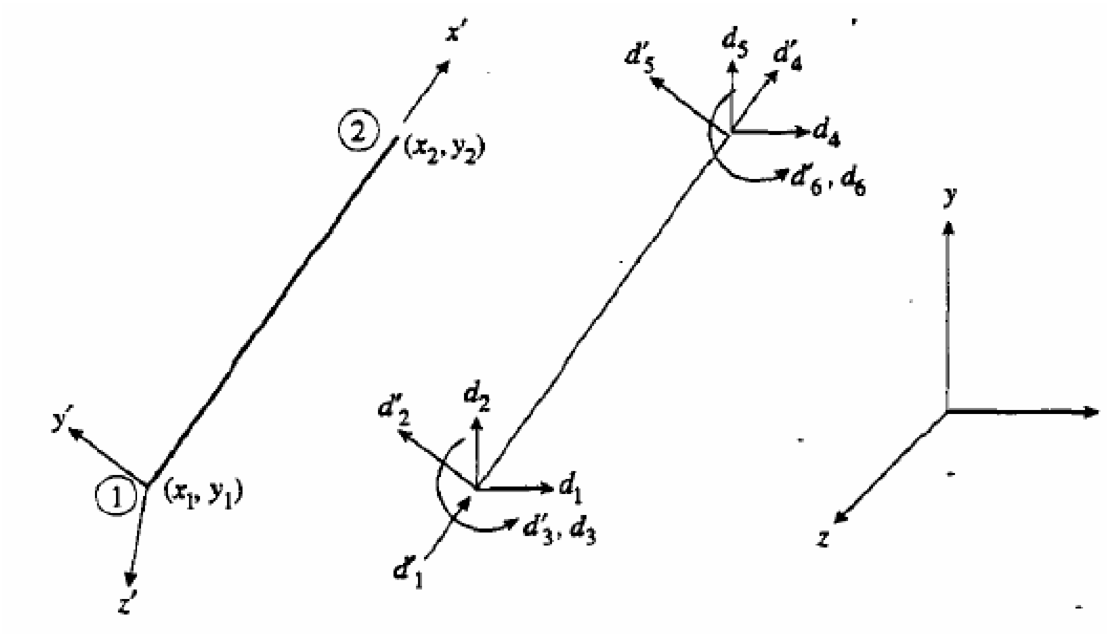
without axial forces

$$\boldsymbol{k'}_{4\times4}\boldsymbol{d}'_{4\times1} = \boldsymbol{f}'_{4\times1}\,. \quad \boldsymbol{k'}_{4\times4} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

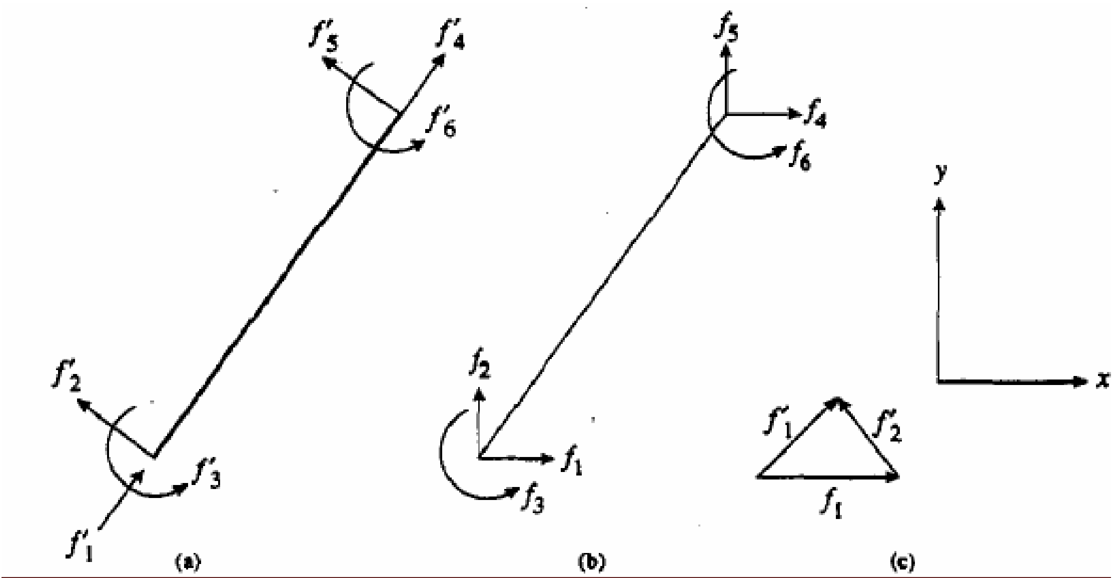
Or with axial forces $\boldsymbol{k'}_{6\times6}\boldsymbol{d}'_{6\times1} = \boldsymbol{f}'_{6\times1}$,

$$\boldsymbol{k'}_{6\times6} = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix}$$

- From the transformation matrix between the local and global coordinate systems shown below, the relationship between the local nodal displacements and global nodal displacements is derived as



Beam element’s local and global coordinate systems and defrees of freedom. Z and z’ axes coincide and point out of the page.



Beam element’s local and global nodal forces:

$$\begin{Bmatrix} d'_1 \\ d'_2 \\ d'_3 \\ d'_4 \\ d'_5 \\ d'_6 \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 & 0 & 0 \\ -m & l & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & 0 \\ 0 & 0 & 0 & -m & l & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ d_6 \end{Bmatrix} \quad \text{or} \quad \boldsymbol{d}'_{6\times1} = \boldsymbol{T}_{6\times6}\boldsymbol{d}_{6\times1}$$

The corresponding nodal force relationship reads

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix} = \begin{bmatrix} l & -m & 0 & 0 & 0 & 0 \\ m & l & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & -m & 0 \\ 0 & 0 & 0 & m & l & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} f'_1 \\ f'_2 \\ f'_3 \\ f'_4 \\ f'_5 \\ f'_6 \end{Bmatrix} \quad \text{or} \quad \boldsymbol{f}_{6\times1} = \boldsymbol{T}^{\mathrm{T}}_{6\times6}\boldsymbol{f}'_{6\times1}$$

- The global governing equations for a truss element are therefore written as $\boldsymbol{k}_{6\times6}\boldsymbol{d}_{6\times1} = \boldsymbol{f}_{6\times1}$

- Element Loads. All the loads on the elements must be transformed to equivalent loads at the node points. The equivalent force system (equivalent joint forces) is nothing but the opposite of the fixed-end forces.
- The major steps in solving any planar frame problem using the direct stiffness method:

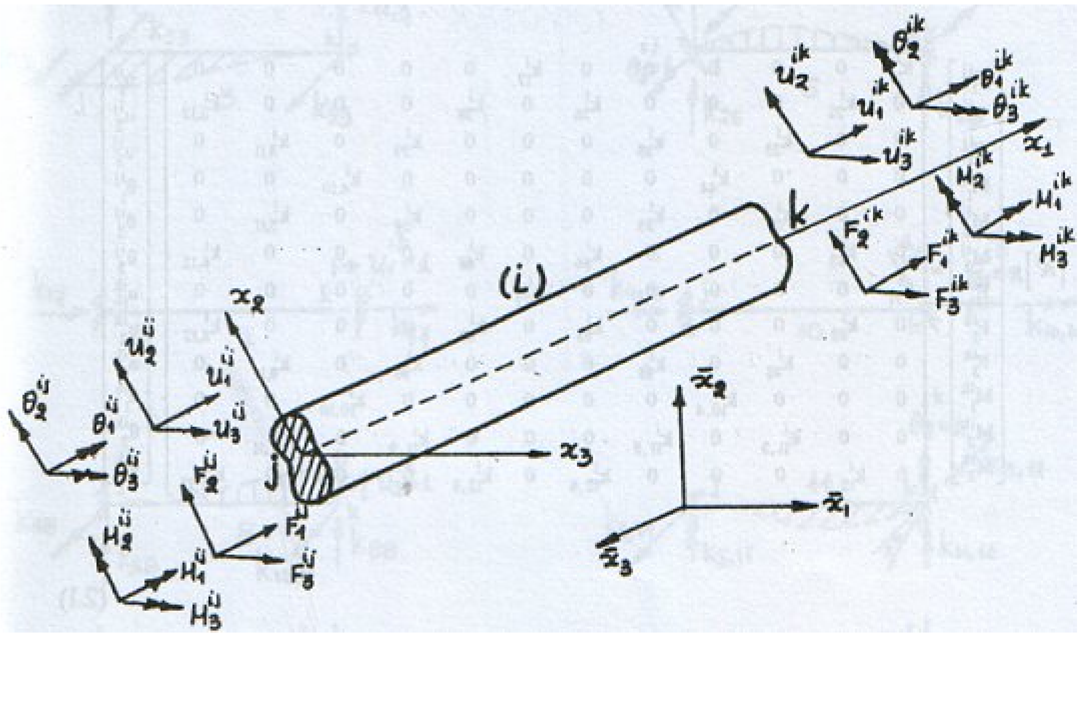
  Step 1: Select the problem units. Set up the coordinate system. Identify and label the nodes and the elements. For each element select a start node (node 1) and an end node (node 2). We use an arrow along the member to indicate the direction from the start node to the end node. This establishes the local coordinate system for each element. Label the three global dof at each node starting at node 1 and proceeding sequentially.

  Step 2: Construct the equilibrium-compatibility equations for a typical element.

Step 3: Using the problem data, construct the element equations from Step 2 for all the elements in the problem. If there are element loads, compute the equivalent joint loads and transform them to the global coordinate system. Note that if there is more than one element load acting on an element, use linear superposition (algebraic sum) of all the element loads acting on that element.
Step 4: Assemble the element equations into the system equations.
Step 5: Impose the boundary conditions.
Step 6: Solve the system equations KD = F for the nodal displacements D.
Step 7: For each element using the nodal displacements, compute
the element nodal forces.

Direct Stiffness Method for 3d Frame Analysis

This procedure is based on the principles of the above described methods for analyzing truss and 2d finite elements. The degrees of freedom are now 12 for one element (3 translations and 3 rotations for each node).



The stiffness matrix of on element of this type has the following form:

$$
[k]=\begin{bmatrix}
\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} & 0 & -\frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\
0 & 0 & \frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 & 0 & 0 & -\frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 \\
0 & 0 & 0 & \frac{GK}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{GK}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_y}{L^2} & 0 & \frac{4EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 \\
0 & \frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{4EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{2EI_z}{L} \\
-\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & \frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12EI_z}{L^3} & 0 & 0 & 0 & -\frac{6EI_z}{L^2} & 0 & \frac{12EI_z}{L^3} & 0 & 0 & 0 & -\frac{6EI_z}{L^2} \\
0 & 0 & -\frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 & 0 & 0 & \frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 \\
0 & 0 & 0 & -\frac{GK}{L} & 0 & 0 & 0 & 0 & 0 & \frac{GK}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{4EI_y}{L} & 0 \\
0 & \frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{2EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{4EI_z}{L}
\end{bmatrix}
$$

The analysis continues according to the previous cases of truss/plane frame elements. The only difference is that the node coordinates, the loads, the supports, releases etc, are given in the 3d space. After the inversion of the structure stiffness matrix, the nodal displacements (3 translations and 3 rotations) as well as the support reactions are calculated. Finally, 3 moments, 2 shear forces and the axial force are calculated with respect to the local coordinate system of each element.

Frame 3D Kernel Files Structure

There are 2 alternatives in order to analyze your model with EngiSSOL Frame 3D routines:

1. You fill all needed variables by code (for example, Visual Basic, C++, etc.) and call a .dll function from your application. Then the results will be saved in specified variables and matrices.
2. You create a text file with the variables and then you execute an .exe file manually or by code from your application. An output file will then be produced with all output information.

The input variables are the following:

NODES:

- Nodes labels and coordinates
- Nodal forces
- Nodal constraints (full supports or spring supports)
- Initial nodal displacements

MATERIALS:

- Material label
- Modulus of elasticity
- Shear Modulus

SECTIONS:

- Section label
- Moment of inertias
- Area Section

MEMBERS (BEAMS):

- Member label
- Starting Node label
- Ending Node label
- Section label
- Material label
- Distributed loads
- Releases of degrees of freedom at starting node (0-6 releases)
- Releases of degrees of freedom at ending node (0-6 releases)

OUTPUT:

- Node displacements
- Support reactions
- Spring reactions (if any)
- With respect to the local coordinate system of each member:
    - Axial force
    - 2 Shear forces
    - 2 Bending moments
    - 1 torsion moment , totally 6 degrees of freedom

Our goal: Quality, reliability, easy use, high scientific standards

To satisfy your needs, we have the ability to create your customized type for giving the data (input) to our kernel files. For example you can send us a template text file with the input data with the desired format. Otherwise we will recommend you a way to call our kernel files or to integrate them to your custom application.

You will also be provided with complete examples that will show how you can set the input variables, call the analysis procedure and finally obtain all results.