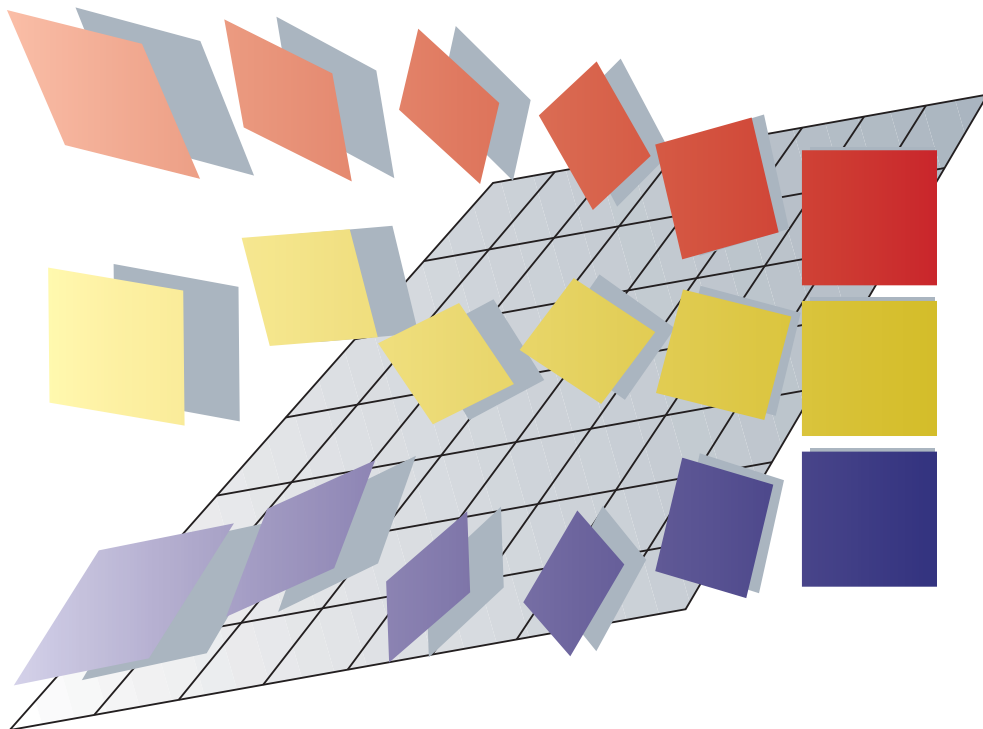




**LUND**  
UNIVERSITY



# CALFEM – A Finite Element Toolbox

## Version 3.6

Manual for frame and truss analysis

Division of Structural Mechanics and Division of Solid Mechanics

Lund University

The software described in this document is furnished under a license agreement. The software may be used or copied only under terms in the license agreement.

No part of this manual may be photocopied or reproduced in any form without the prior written consent by the Division of Structural Mechanics.

© Copyright 1992–2022 by the Division of Structural Mechanics at Lund University. All rights reserved.

CALFEM is the trademark of the Division of Structural Mechanics, Lund University.  
MATLAB is the trademark of The MathWorks, Inc.

E-mail address:

`calfem@byggmek.lth.se`

Homepage:

`http://www.byggmek.lth.se/Calfem`

Contacts:

The Division of Structural Mechanics  
Lund University  
PO Box 118  
SE-221 00 Lund  
SWEDEN  
Phone: +46 46 222 0000  
Fax: +46 46 222 4420

# Preface

CALFEM® is an interactive computer program for teaching the finite element method (FEM). The name CALFEM is an abbreviation of "Computer Aided Learning of the Finite Element Method". The program can be used for different types of structural mechanics problems and field problems.

CALFEM, the program and its built-in philosophy have been developed at the Division of Structural Mechanics, Lund University, starting in the late 70's. Many coworkers, former and present, have been engaged in the development at different stages, of whom we might mention

*Per-Erik Austrell*

*Susanne Heyden*

*Karl-Gunnar Olsson*

*Hans Petersson*

*Erik Serrano*

*Håkan Carlsson*

*Jonas Lindemann*

*Kent Persson*

*Matti Ristinmaa*

*Per-Anders Wernberg*

*Ola Dahlblom*

*Anders Olsson*

*Anders Peterson*

*Göran Sandberg*

This release represents the latest development of CALFEM. The functions for finite element applications are all MATLAB functions (.m-files) as described in the MATLAB manual. We believe that this environment increases the versatility and handling of the program and, above all, the ease of teaching the finite element method.

Lund, May 30, 2022

The authors



# Contents

|          |                                   |            |
|----------|-----------------------------------|------------|
| <b>1</b> | <b>Introduction</b>               | <b>1</b>   |
| <b>2</b> | <b>General purpose functions</b>  | <b>3</b>   |
| <b>3</b> | <b>Matrix functions</b>           | <b>19</b>  |
| <b>4</b> | <b>Element functions</b>          | <b>39</b>  |
| 4.1      | Introduction . . . . .            | 39         |
| 4.2      | Spring element . . . . .          | 40         |
| 4.3      | Bar elements . . . . .            | 45         |
| 4.4      | Beam elements . . . . .           | 64         |
| <b>5</b> | <b>System functions</b>           | <b>97</b>  |
| 5.1      | Introduction . . . . .            | 97         |
| 5.2      | Static system functions . . . . . | 99         |
| <b>6</b> | <b>Statements and macros</b>      | <b>111</b> |
| <b>7</b> | <b>Graphics functions</b>         | <b>117</b> |
| <b>8</b> | <b>User's Manual, examples</b>    | <b>135</b> |
| 8.1      | Introduction . . . . .            | 135        |
| 8.2      | Static analysis . . . . .         | 137        |
| 8.3      | Nonlinear analysis . . . . .      | 169        |



---

# 1 Introduction

The computer program CALFEM is a MATLAB toolbox for finite element applications. This manual concerns mainly the finite element functions, but it also contains descriptions of some often used MATLAB functions.

The finite element analysis can be carried out either interactively or in a batch oriented fashion. In the interactive mode the functions are evaluated one by one in the MATLAB command window. In the batch oriented mode a sequence of functions are written in a file named .m-file, and evaluated by writing the file name in the command window. The batch oriented mode is a more flexible way of performing finite element analysis because the .m-file can be written in an ordinary editor. This way of using CALFEM is recommended because it gives a structured organization of the functions. Changes and reruns are also easily executed in the batch oriented mode.

A command line consists typically of functions for vector and matrix operations, calls to functions in the CALFEM finite element library or commands for workspace operations. An example of a command line for a matrix operation is

$$C = A + B'$$

where two matrices  $A$  and  $B'$  are added together and the result is stored in matrix  $C$ . The matrix  $B'$  is the transpose of  $B$ . An example of a call to the element library is

$$Ke = \text{spring1e}(k)$$

where the two-by-two element stiffness matrix  $K^e$  is computed for a spring element with spring stiffness  $k$ , and is stored in the variable  $Ke$ . The input argument is given within parentheses ( ) after the name of the function. Some functions have multiple input arguments and/or multiple output arguments. For example

$$[\text{lambda}, X] = \text{eigen}(K, M)$$

computes the eigenvalues and eigenvectors to a pair of matrices  $K$  and  $M$ . The output variables - the eigenvalues stored in the vector **lambda** and the corresponding eigenvectors stored in the matrix **X** - are surrounded by brackets [ ] and separated by commas. The input arguments are given inside the parentheses and also separated by commas.

The statement

$$\text{help } \textit{function}$$

provides information about purpose and syntax for the specified function.

---

The available functions are organized in groups as follows. Each group is described in a separate chapter.

| Groups of functions             |   |
|---------------------------------|---|
| <b>General purpose commands</b> | for managing variables, workspace, output etc     |
| <b>Matrix functions</b>         | for matrix handling                               |
| <b>Element functions</b>        | for computing element matrices and element forces |
| <b>System functions</b>         | for setting up and solving systems of equations   |
| <b>Statement functions</b>      | for algorithm definitions                         |
| <b>Graphics functions</b>       | for plotting                                      |



---

## 2 General purpose functions

The general purpose functions are used for managing variables and workspace, control of output etc. The functions listed here are a subset of the general purpose functions described in the MATLAB manual. The functions can be divided into the following groups

| Managing commands and functions |  |
|---------------------------------|--|
| help                            | Online documentation                           |
| type                            | List .m-file                                   |
| what                            | Directory listing of .m-, .mat- and .mex-files |
| ...                             | Continuation                                   |
| %                               | Write a comment line                           |

| Managing variables and the workspace |   |
|--------------------------------------|---|
| clear                                | Remove variables from workspace                   |
| disp                                 | Display variables in workspace on display screen  |
| load                                 | Retrieve variable from disk and load in workspace |
| save                                 | Save matrix bank variable on disk                 |
| who,<br>whos                         | List directory of variables in workspace          |

| Working with files and controlling the command window |   |
|---|---|
| diary   | Save session in a named file                    |
| echo  | Control output on the display screen            |
| format  | Control the output display format               |
| quit  | Stop execution and exit from the CALFEM program |

## **clear**

---

### **Purpose:**

Remove variables from workspace.

### **Syntax:**

```
clear  
clear name1 name2 name3 ...
```

### **Description:**

`clear` removes all variables from workspace.

`clear name1 name2 name3 ...` removes specified variables from workspace.

### **Note:**

This is a MATLAB built-in function. For more information about the `clear` function, type `help clear`.

**Purpose:**

Save session in a disk file.

**Syntax:**

```
diary filename  
diary off  
diary on
```

**Description:**

**diary** *filename* writes a copy of all subsequent keyboard input and most of the resulting output (but not graphs) on the named file. If the file *filename* already exists, the output is appended to the end of that file.

**diary off** stops storage of the output.

**diary on** turns it back on again, using the current filename or default filename **diary** if none has yet been specified.

The **diary** function may be used to store the current session for later runs. To make this possible, finish each command line with semicolon ';' to avoid the storage of intermediate results on the named diary file.

**Note:**

This is a MATLAB built-in function. For more information about the **diary** function, type **help diary**.

## **disp**

---

### **Purpose:**

Display a variable in matrix bank on display screen.

### **Syntax:**

`disp(A)`

### **Description:**

`disp(A)` displays the matrix *A* on the display screen.

### **Note:**

This is a MATLAB built-in function. For more information about the **disp** function, type `help disp`.

**Purpose:**

Control output on the display screen.

**Syntax:**

echo on  
echo off  
echo

**Description:**

echo on turns on echoing of commands inside Script-files.

echo off turns off echoing.

echo by itself, toggles the echo state.

**Note:**

This is a MATLAB built-in function. For more information about the `echo` function, type `help echo`.

**Purpose:**

Control the output display format.

**Syntax:**

See the listing below.

**Description:**

**format** controls the output format. By default, MATLAB displays numbers in a short format with five decimal digits.

| Command               | Result                      | Example                |
|-----------------------|-----------------------------|------------------------|
| <b>format short</b>   | 5 digit scaled fixed point  | 3.1416                 |
| <b>format long</b>    | 15 digit scaled fixed point | 3.14159265358979       |
| <b>format short e</b> | 5 digit floating point      | 3.1416e+000            |
| <b>format long e</b>  | 16 digit floating point     | 3.141592653589793e+000 |

**Note:**

This is a MATLAB built-in function. For more information about the **format** function, type **help format**.

**Purpose:**

Display a description of purpose and syntax for a specific function.

**Syntax:**

`help function name`

**Description:**

`help` provides an online documentation for the specified function.

**Example:**

Typing

```
>> help spring1e
```

yields

```
Ke=spring1e(ep)
```

```
-----  
PURPOSE
```

```
  Compute element stiffness matrix for spring (analog) element.
```

```
INPUT:  ep = [k]; spring stiffness or analog quantity.
```

```
OUTPUT: Ke : stiffness matrix, dim(Ke)= 2 x 2  
-----
```

**Note:**

This is a MATLAB built-in function. For more information about the **help** function, type `help help`.

## load

---

### Purpose:

Retrieve variable from disk and load in workspace.

### Syntax:

```
load filename  
load filename.ext
```

### Description:

`load filename` retrieves the variables from the binary file *filename.mat*.

`load filename.ext` reads the ASCII file *filename.ext* with numeric data arranged in  $m$  rows and  $n$  columns. The result is an  $m$ -by- $n$  matrix residing in workspace with the name *filename*, i.e. with the extension stripped.

### Note:

This is a MATLAB built-in function. For more information about the `load` function, type `help load`.



**Purpose:**

Terminate CALFEM session.

**Syntax:**

`quit`

**Description:**

`quit filename` terminates the CALFEM without saving the workspace.

**Note:**

This is a MATLAB built-in function. For more information about the `quit` function, type `help quit`.

**Purpose:**

Save workspace variables on disk.

**Syntax:**

```
save filename  
save filename variables  
save filename variables -ascii
```

**Description:**

`save filename` writes all variables residing in workspace in a binary file named *filename.mat*

`save filename variables` writes named variables, separated by blanks, in a binary file named *filename.mat*

`save filename variables -ascii` writes named variables in an ASCII file named *filename*.

**Note:**

This is a MATLAB built-in function. For more information about the `save` function, type `help save`.

**Purpose:**

List file.

**Syntax:**

`type filename`

**Description:**

`type filename` lists the specified file. Use path names in the usual way for your operating system. If a filename extension is not given, .m is added by default. This makes it convenient to list the contents of .m-files on the screen.

**Note:**

This is a MATLAB built-in function. For more information about the `type` function, type `help type`.

## what

---

### Purpose:

Directory listing of .m-files, .mat-files and .mex-files.

### Syntax:

`what`  
`what dirname`

### Description:

`what` lists the .m-files, .mat-files and .mex-files in the current directory.

`what dirname` lists the files in directory *dirname* in the MATLAB search path. The syntax of the path depends on your operating system.

### Note:

This is a MATLAB built-in function. For more information about the `what` function, type `help what`.

**Purpose:**

List directory of variables in matrix bank.

**Syntax:**

who  
whos

**Description:**

who lists the variables currently in memory.

whos lists the current variables and their size.

**Examples:**

who

Your variables are:

```
A  B  C
K  M  X
k  lambda
```

whos

| name   | size     | elements | bytes | density | complex |
|--------|----------|----------|-------|---------|---------|
| A      | 3-by-3   | 9        | 72    | Full    | No      |
| B      | 3-by-3   | 9        | 72    | Full    | No      |
| C      | 3-by-3   | 9        | 72    | Full    | No      |
| K      | 20-by-20 | 400      | 3200  | Full    | No      |
| M      | 20-by-20 | 400      | 3200  | Full    | No      |
| X      | 20-by-20 | 400      | 3200  | Full    | No      |
| k      | 1-by-1   | 1        | 8     | Full    | No      |
| lambda | 20-by-1  | 20       | 160   | Full    | No      |

Grand total is 1248 elements using 9984 bytes

**Note:**

These are MATLAB built-in functions. For more information about the functions, type `help who` or `help whos`.

...

---

**Purpose:**

Continuation.

**Syntax:**

...

**Description:**

An expression can be continued on the next line by using ... .

**Note:**

This is a MATLAB built-in function.

**Purpose:**

Write a comment line.

**Syntax:**

`% arbitrary text`

**Description:**

An arbitrary text can be written after the symbol %.

**Note:**

This is a MATLAB built-in character.





---

### 3 Matrix functions

The group of matrix functions comprises functions for vector and matrix operations and also functions for sparse matrix handling. MATLAB has two storage modes, full and sparse. Only nonzero entries and their indices are stored for sparse matrices. Sparse matrices are not created automatically. But once initiated, sparsity propagates. Operations on sparse matrices produce sparse matrices and operations on a mixture of sparse and full matrices also normally produce sparse matrices.

The following functions are described in this chapter:

| Vector and matrix operations |   |
|------------------------------|---|
| [ ] ( ) =                    | Special characters                          |
| ' . , ;                      | Special characters                          |
| :                            | Create vectors and do matrix subscripting   |
| + - * /                      | Matrix arithmetic                           |
| abs                          | Absolute value                              |
| det                          | Matrix determinant                          |
| diag                         | Diagonal matrices and diagonals of a matrix |
| inv                          | Matrix inverse                              |
| length                       | Vector length                               |
| max                          | Maximum element(s) of a matrix              |
| min                          | Minimum element(s) of a matrix              |
| ones                         | Generate a matrix of all ones               |
| size                         | Matrix dimensions                           |
| sqrt                         | Square root                                 |
| sum                          | Sum of the elements of a matrix             |
| zeros                        | Generate a zero matrix                      |

| Sparse matrix handling |                                      |
|------------------------|--------------------------------------|
| full                   | Convert sparse matrix to full matrix |
| sparse                 | Create sparse matrix                 |
| spy                    | Visualize sparsity structure         |

[ ] ( ) = ' . , ;

---

**Purpose:**

Special characters.

**Syntax:**

[ ] ( ) = ' . , ;

**Description:**

- [ ] Brackets are used to form vectors and matrices.
- ( ) Parentheses are used to indicate precedence in arithmetic expressions and to specify an element of a matrix.
- = Used in assignment statements.
- ' Matrix transpose.  $X'$  is the transpose of  $X$ . If  $X$  is complex, the apostrophe sign performs complex conjugate as well. Do  $X.'$  if only the transpose of the complex matrix is desired
- . Decimal point. 314/100, 3.14 and 0.314e1 are all the same.
- , Comma. Used to separate matrix subscripts and function arguments.
- ; Semicolon. Used inside brackets to end rows. Used after an expression to suppress printing or to separate statements.

**Examples:**

By the statement

$$a = 2$$

the scalar  $a$  is assigned a value of 2. An element in a matrix may be assigned a value according to

$$A(2,5) = 3$$

The statement

$$D = [ 1 \ 2; 3 \ 4]$$

results in matrix

$$D = \begin{bmatrix} 1 & 2 \\ 3 & 4 \end{bmatrix}$$

stored in the matrix bank. To copy the contents of the matrix  $D$  to a matrix  $E$ , use

$$E = D$$

The character  $'$  is used in the following statement to store the transpose of the matrix  $A$  in a new matrix  $F$

$$F = A'$$

**Note:**

These are MATLAB built-in characters.

**Purpose:**

Create vectors and do matrix subscripting.

**Description:**

The colon operator uses the following rules to create regularly spaced vectors:

$j : k$  is the same as  $[j, j + 1, \dots, k]$

$j : i : k$  is the same as  $[j, j + i, j + 2i, \dots, k]$

The colon notation may also be used to pick out selected rows, columns, and elements of vectors and matrices:

$A(:, j)$  is the  $j$ :th column of  $A$

$A(i, :)$  is the  $i$ :th row of  $A$

**Examples:**

The colon ':' used with integers

$$d = 1 : 4$$

results in a row vector

$$d = [1 \ 2 \ 3 \ 4]$$

stored in the workspace.

The colon notation may be used to display selected rows and columns of a matrix on the terminal. For example, if we have created a 3-times-4 matrix  $D$  by the statement

$$D = [d ; 2 * d ; 3 * d]$$

resulting in

$$D = \begin{bmatrix} 1 & 2 & 3 & 4 \\ 2 & 4 & 6 & 8 \\ 3 & 6 & 9 & 12 \end{bmatrix}$$

columns three and four are displayed by entering

$$D(:, 3 : 4)$$

resulting in

$$D(:, 3 : 4) = \begin{bmatrix} 3 & 4 \\ 6 & 8 \\ 9 & 12 \end{bmatrix}$$

In order to copy parts of the  $D$  matrix into another matrix the colon notation is used as

$$E(3 : 4, 2 : 3) = D(1 : 2, 3 : 4)$$

:

---

Assuming the matrix **E** was a zero matrix before the statement is executed, the result will be

$$\mathbf{E} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 3 & 4 & 0 \\ 0 & 6 & 8 & 0 \end{bmatrix}$$

**Note:**

This is a MATLAB built-in character.

**Purpose:**

Matrix arithmetic.

**Syntax:**

$A + B$

$A - B$

$A * B$

$A/s$

**Description:**

Matrix operations are defined by the rules of linear algebra.

**Examples:**

An example of a sequence of matrix-to-matrix operations is

$$D = A + B - C$$

A matrix-to-vector multiplication followed by a vector-to-vector subtraction may be defined by the statement

$$b = c - A * x$$

and finally, to scale a matrix by a scalar  $s$  we may use

$$B = A/s$$

**Note:**

These are MATLAB built-in operators.

**Purpose:**

Absolute value.

**Syntax:**

`B=abs(A)`

**Description:**

`B=abs(A)` computes the absolute values of the elements of matrix **A** and stores them in matrix **B**.

**Examples:**

Assume the matrix

$$C = \begin{bmatrix} -7 & 4 \\ -3 & -8 \end{bmatrix}$$

The statement `D=abs(C)` results in a matrix

$$D = \begin{bmatrix} 7 & 4 \\ 3 & 8 \end{bmatrix}$$

stored in the workspace.

**Note:**

This is a MATLAB built-in function. For more information about the **abs** function, type `help abs`.

**Purpose:**

Matrix determinant.

**Syntax:**

`a=det(A)`

**Description:**

`a=det(A)` computes the determinant of the matrix `A` and stores it in the scalar `a`.

**Note:**

This is a MATLAB built-in function. For more information about the `det` function, type `help det`.

## diag

---

### Purpose:

Diagonal matrices and diagonals of a matrix.

### Syntax:

```
M=diag(v)
v=diag(M)
```

### Description:

For a vector  $\mathbf{v}$  with  $n$  components, the statement  $\mathbf{M}=\text{diag}(\mathbf{v})$  results in an  $n \times n$  matrix  $\mathbf{M}$  with the elements of  $\mathbf{v}$  as the main diagonal.

For a  $n \times n$  matrix  $\mathbf{M}$ , the statement  $\mathbf{v}=\text{diag}(\mathbf{M})$  results in a column vector  $\mathbf{v}$  with  $n$  components formed by the main diagonal in  $\mathbf{M}$ .

### Note:

This is a MATLAB built-in function. For more information about the **diag** function, type `help diag`.



**Purpose:**

Convert sparse matrices to full storage class.

**Syntax:**

`A=full(S)`

**Description:**

`A=full(S)` converts the storage of a matrix from sparse to full. If `A` is already full, `full(A)` returns `A`.

**Note:**

This is a MATLAB built-in function. For more information about the `full` function, type `help full`.

## **inv**

---

### **Purpose:**

Matrix inverse.

### **Syntax:**

`B=inv(A)`

### **Description:**

`B=inv(A)` computes the inverse of the square matrix `A` and stores the result in the matrix `B`.

### **Note:**

This is a MATLAB built-in function. For more information about the `inv` function, type `help inv`.

**Purpose:**

Vector length.

**Syntax:**

`n=length(x)`

**Description:**

`n=length(x)` returns the dimension of the vector `x`.

**Note:**

This is a MATLAB built-in function. For more information about the **length** function, type `help length`.

**Purpose:**

Maximum element(s) of a matrix.

**Syntax:**

`b=max(A)`

**Description:**

For a vector **a**, the statement `b=max(a)` assigns the scalar **b** the maximum element of the vector **a**.

For a matrix **A**, the statement `b=max(A)` returns a row vector **b** containing the maximum elements found in each column vector in **A**.

The maximum element found in a matrix may thus be determined by `c=max(max(A))`.

**Examples:**

Assume the matrix **B** is defined as

$$B = \begin{bmatrix} -7 & 4 \\ -3 & -8 \end{bmatrix}$$

The statement `d=max(B)` results in a row vector

$$d = \begin{bmatrix} -3 & 4 \end{bmatrix}$$

The maximum element in the matrix **B** may be found by `e=max(d)` which results in the scalar **e** = 4.

**Note:**

This is a MATLAB built-in function. For more information about the `max` function, type `help max`.

**Purpose:**

Minimum element(s) of a matrix.

**Syntax:**

`b=min(A)`

**Description:**

For a vector **a**, the statement `b=min(a)` assigns the scalar **b** the minimum element of the vector **a**.

For a matrix **A**, the statement `b=min(A)` returns a row vector **b** containing the minimum elements found in each column vector in **A**.

The minimum element found in a matrix may thus be determined by `c=min(min(A))`.

**Examples:**

Assume the matrix **B** is defined as

$$B = \begin{bmatrix} -7 & 4 \\ -3 & -8 \end{bmatrix}$$

The statement `d=min(B)` results in a row vector

$$d = \begin{bmatrix} -7 & -8 \end{bmatrix}$$

The minimum element in the matrix **B** is then found by `e=min(d)`, which results in the scalar **e** = -8.

**Note:**

This is a MATLAB built-in function. For more information about the `min` function, type `help min`.

**Purpose:**

Generate a matrix of all ones.

**Syntax:**

`A=ones(m,n)`

**Description:**

`A=ones(m,n)` results in an `m`-times-`n` matrix `A` with all ones.

**Note:**

This is a MATLAB built-in function. For more information about the `ones` function, type `help ones`.

**Purpose:**

Matrix dimensions.

**Syntax:**

```
d=size(A)
[m,n]=size(A)
```

**Description:**

`d=size(A)` returns a vector with two integer components, `d=[m,n]`, from the matrix `A` with dimensions `m` times `n`.

`[m,n]=size(A)` returns the dimensions `m` and `n` of the  $m \times n$  matrix `A`.

**Note:**

This is a MATLAB built-in function. For more information about the `size` function, type `help size`.

**Purpose:**

Create sparse matrices.

**Syntax:**

```
S=sparse(A)  
S=sparse(m,n)
```

**Description:**

`S=sparse(A)` converts a full matrix to sparse form by extracting all nonzero matrix elements. If `S` is already sparse, `sparse(S)` returns `S`.

`S=sparse(m,n)` generates an m-times-n sparse zero matrix.

**Note:**

This is a MATLAB built-in function. For more information about the `sparse` function, type `help sparse`.



**Purpose:**

Visualize matrix sparsity structure.

**Syntax:**

`spy(S)`

**Description:**

`spy(S)` plots the sparsity structure of any matrix `S`. `S` is usually a sparse matrix, but the function also accepts full matrices and the nonzero matrix elements are plotted.

**Note:**

This is a MATLAB built-in function. For more information about the `spy` function, type `help spy`.

## **sqrt**

---

### **Purpose:**

Square root.

### **Syntax:**

`B=sqrt(A)`

### **Description:**

`B=sqrt(A)` computes the square root of the elements in matrix `A` and stores the result in matrix `B`.

### **Note:**

This is a MATLAB built-in function. For more information about the `sqrt` function, type `help sqrt`.

**Purpose:**

Sum of the elements of a matrix.

**Syntax:**

`b=sum(A)`

**Description:**

For a vector **a**, the statement `b=sum(a)` results in a scalar **a** containing the sum of all elements of **a**.

For a matrix **A**, the statement `b=sum(A)` returns a row vector **b** containing the sum of the elements found in each column vector of **A**.

The sum of all elements of a matrix is determined by `c=sum(sum(A))`.

**Note:**

This is a MATLAB built-in function. For more information about the `sum` function, type `help sum`.

**Purpose:**

Generate a zero matrix.

**Syntax:**

`A=zeros(m,n)`

**Description:**

`A=zeros(m,n)` results in an `m`-times-`n` matrix `A` of zeros.

**Note:**

This is a MATLAB built-in function. For more information about the **zeros** function, type `help zeros`.

---

## 4 Element functions

### 4.1 Introduction

The group of element functions contains functions for computation of element matrices and element forces for different element types. The element functions have been divided into the following groups

|                       |
|-----------------------|
| <b>Spring element</b> |
| <b>Bar elements</b>   |
| <b>Beam elements</b>  |

For each element type there is a function for computation of the element stiffness matrix  $\mathbf{K}^e$ . For most of the elements, an element load vector  $\mathbf{f}^e$  can also be computed. These functions are identified by their last letter -e.

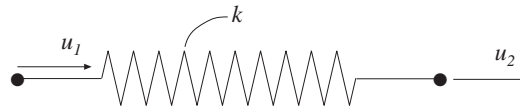
Using the function **assem**, the element stiffness matrices and element load vectors are assembled into a global stiffness matrix  $\mathbf{K}$  and a load vector  $\mathbf{f}$ . Unknown nodal values of temperatures or displacements  $\mathbf{a}$  are computed by solving the system of equations  $\mathbf{Ka} = \mathbf{f}$  using the function **solveq**. A vector of nodal values of temperatures or displacements for a specific element is formed by the function **extract**.

When the element nodal values have been computed, the element flux or element stresses can be calculated using functions specific to the element type concerned. These functions are identified by their last letter -s.

For some elements, a function for computing the internal force vector is also available. These functions are identified by their last letter -f.

## 4.2 Spring element

The spring element, shown below, can be used for the analysis of one-dimensional spring systems and for a variety of analogous physical problems.



Quantities corresponding to the variables of the spring are listed in Table 1.

| Problem type       | Spring stiffness           | Nodal displacement | Element force | Spring force |
|--------------------|----------------------------|--------------------|---------------|--------------|
| Spring             | $k$                        | $u$                | $P$           | $N$          |
| Bar                | $\frac{EA}{L}$             | $u$                | $P$           | $N$          |
| Thermal conduction | $\frac{\lambda A}{L}$      | $T$                | $\bar{H}$     | $H$          |
| Diffusion          | $\frac{DA}{L}$             | $c$                | $\bar{H}$     | $H$          |
| Electrical circuit | $\frac{1}{R}$              | $U$                | $\bar{I}$     | $I$          |
| Groundwater flow   | $\frac{kA}{L}$             | $\phi$             | $\bar{H}$     | $H$          |
| Pipe network       | $\frac{\pi D^4}{128\mu L}$ | $p$                | $\bar{H}$     | $H$          |

Table 1: *Analogous quantities*

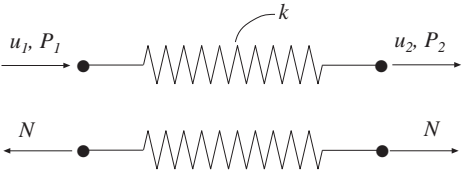
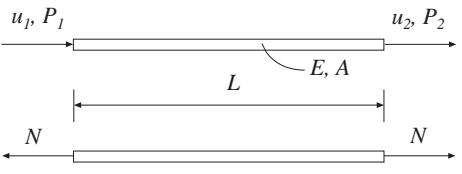
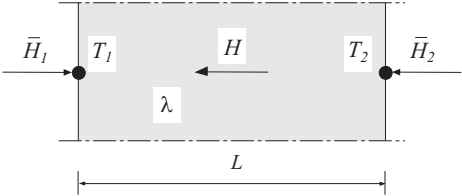
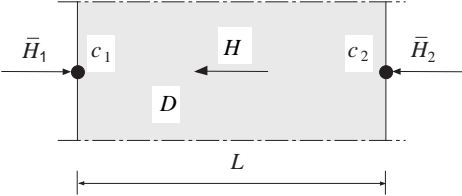
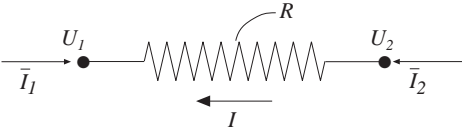
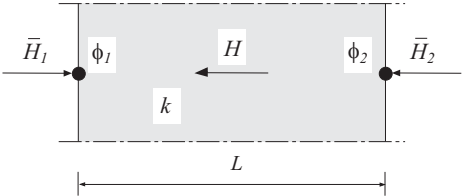
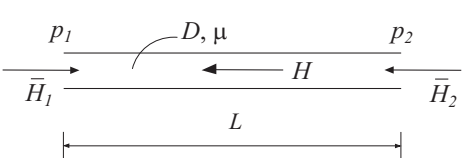
| Interpretations of the spring element |   |  |   |
|---------------------------------------|---|--|---|
| Problem type                          | Quantities  | Designations                                   |   |
| Spring                                |    | $k$<br>$u$<br>$P$<br>$N$                       | spring stiffness<br>displacement<br>element force<br>spring force   |
| Bar                                   |    | $L$<br>$E$<br>$A$<br>$u$<br>$P$<br>$N$         | length<br>modulus of elasticity<br>area of cross section<br>displacement<br>element force<br>normal force |
| Thermal conduction                    |    | $L$<br>$\lambda$<br>$T$<br>$\bar{H}$<br>$H$    | length<br>thermal conductivity<br>temperature<br>element heat flow<br>internal heat flow                  |
| Diffusion                             |  | $L$<br>$D$<br>$c$<br>$\bar{H}$<br>$H$          | length<br>diffusivity<br>nodal concentration<br>nodal mass flow<br>element mass flow                      |
| Electrical circuit                    |  | $R$<br>$U$<br>$\bar{I}$<br>$I$                 | resistance<br>potential<br>element current<br>internal current  |
| Ground-water flow                     |  | $L$<br>$k$<br>$\phi$<br>$\bar{H}$<br>$H$       | length<br>permeability<br>piezometric head<br>element water flow<br>internal water flow                   |
| Pipe network (laminar flow)           |  | $L$<br>$D$<br>$\mu$<br>$p$<br>$\bar{H}$<br>$H$ | length<br>pipe diameter<br>viscosity<br>pressure<br>element fluid flow<br>internal fluid flow             |

Table 2: *Quantities used in different types of problems*

---

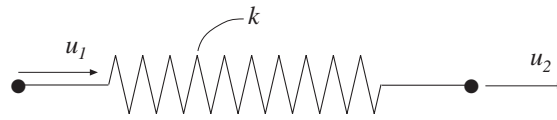
The following functions are available for the spring element:

| <b>Spring functions</b> |                        |
|-------------------------|------------------------|
| <b>spring1e</b>         | Compute element matrix |
| <b>spring1s</b>         | Compute spring force   |



**Purpose:**

Compute element stiffness matrix for a spring element.

**Syntax:**

$\mathbf{K}e = \text{spring1e}(\text{ep})$

**Description:**

`spring1e` provides the element stiffness matrix  $\mathbf{K}e$  for a spring element.

The input variable

$$\text{ep} = [k]$$

supplies the spring stiffness  $k$  or the analog quantity defined in Table 1.

**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in  $\mathbf{K}e$ , is computed according to

$$\mathbf{K}^e = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

where  $k$  is defined by `ep`.

**Purpose:**

Compute spring force in a spring element.

**Syntax:**

```
es=spring1s(ep,ed)
```

**Description:**

**spring1s** computes the spring force **es** in a spring element.

The input variable **ep** is defined in **spring1e** and the element nodal displacements **ed** are obtained by the function **extract**.

The output variable

$$\mathbf{es} = [ N ]$$

contains the spring force  $N$ , or the analog quantity.

**Theory:**

The spring force  $N$ , or analog quantity, is computed according to

$$N = k ( u_2 - u_1 )$$

---

### 4.3 Bar elements

Bar elements are available for one, two, and three dimensional analysis.

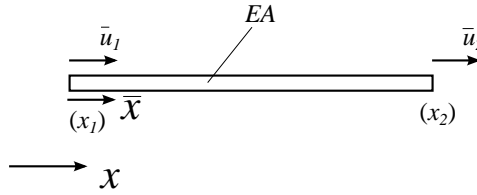
| One dimensional bar elements |   |
|------------------------------|---|
| bar1e                        | Compute element matrix                                      |
| bar1s                        | Compute normal force  |
| bar1we                       | Compute element matrix for bar element with elastic support |
| bar1ws                       | Compute normal force for bar element with elastic support   |

| Two dimensional bar elements |  |
|------------------------------|--|
| bar2e                        | Compute element matrix   |
| bar2s                        | Compute normal force   |
| bar2ge                       | Compute element matrix for geometric nonlinear element               |
| bar2gs                       | Compute normal force and axial force for geometric nonlinear element |

| Three dimensional bar elements |                        |
|--------------------------------|------------------------|
| bar3e                          | Compute element matrix |
| bar3s                          | Compute normal force   |

**Purpose:**

Compute element stiffness matrix for a one dimensional bar element.

**Syntax:**

```
Ke=bar1e(ex,ep)
[Ke,fe]=bar1e(ex,ep,eq)
```

**Description:**

**bar1e** provides the element stiffness matrix **Ke** for a one dimensional bar element. The input variables

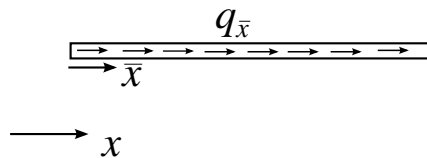
$$\mathbf{ex} = [x_1 \ x_2] \quad \mathbf{ep} = [E \ A]$$

supply the element nodal coordinates  $x_1$  and  $x_2$ , the modulus of elasticity  $E$ , and the cross section area  $A$ .

The element load vector **fe** can also be computed if uniformly distributed load is applied to the element. The optional input variable

$$\mathbf{eq} = [q_{\bar{x}}]$$

then contains the distributed load per unit length,  $q_{\bar{x}}$ .

**Theory:**

The element stiffness matrix  $\bar{\mathbf{K}}^e$ , stored in **Ke**, is computed according to

$$\bar{\mathbf{K}}^e = \frac{D_{EA}}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

where the axial stiffness  $D_{EA}$  and the length  $L$  are given by

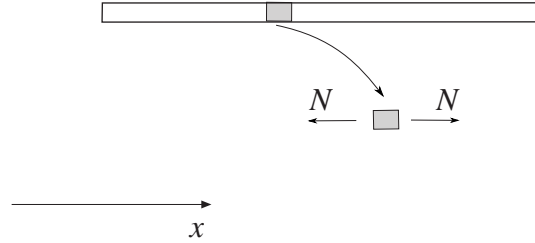
$$D_{EA} = EA; \quad L = x_2 - x_1$$

The element load vector  $\bar{\mathbf{f}}_l^e$ , stored in **fe**, is computed according to

$$\bar{\mathbf{f}}_l^e = \frac{q_{\bar{x}}L}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

**Purpose:**

Compute normal force in a one dimensional bar element.

**Syntax:**

```
es=bar1s(ex,ep,ed)
es=bar1s(ex,ep,ed,eq)
[es,edi]=bar1s(ex,ep,ed,eq,n)
[es,edi,eci]=bar1s(ex,ep,ed,eq,n)
```

**Description:**

**bar1s** computes the normal force in the one dimensional bar element **bar1e**.

The input variables **ex** and **ep** are defined in **bar1e** and the element nodal displacements, stored in **ed**, are obtained by the function **extract**. If distributed load is applied to the element, the variable **eq** must be included. The number of evaluation points for normal force and displacement are determined by **n**. If **n** is omitted, only the ends of the bar are evaluated.

The output variables

$$\mathbf{es} = \begin{bmatrix} N(0) \\ N(\bar{x}_2) \\ \vdots \\ N(\bar{x}_{n-1}) \\ N(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} u(0) \\ u(\bar{x}_2) \\ \vdots \\ u(\bar{x}_{n-1}) \\ u(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the normal force, the displacement, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the bar element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \end{bmatrix}$$

The transpose of  $\bar{\mathbf{a}}^e$  is stored in **ed**.

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}\bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}^{-1} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix}$$

$$u_p(\bar{x}) = -\frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

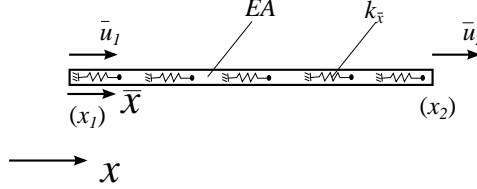
$$N_p(\bar{x}) = -q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EA}$ ,  $L$ , and  $q_{\bar{x}}$  are defined in bar1e and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a one dimensional bar element with elastic support.

**Syntax:**

```
Ke=bar1we(ex,ep)
[Ke,fe]=bar1we(ex,ep,eq)
```

**Description:**

bar1we provides the element stiffness matrix **Ke** for a one dimensional bar element with elastic support. The input variables

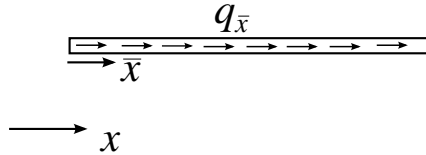
$$\mathbf{ex} = [x_1 \ x_2] \quad \mathbf{ep} = [E \ A \ k_{\bar{x}}]$$

supply the element nodal coordinates  $x_1$  and  $x_2$ , the modulus of elasticity  $E$ , the cross section area  $A$  and the stiffness of the axial springs  $k_{\bar{x}}$ .

The element load vector **fe** can also be computed if uniformly distributed load is applied to the element. The optional input variable

$$\mathbf{eq} = [q_{\bar{x}}]$$

then contains the distributed load per unit length,  $q_{\bar{x}}$ .

**Theory:**

The element stiffness matrix  $\bar{\mathbf{K}}^e$ , stored in **Ke**, is computed according to

$$\bar{\mathbf{K}}^e = \bar{\mathbf{K}}_0^e + \bar{\mathbf{K}}_s^e$$

$$\bar{\mathbf{K}}_0^e = \frac{D_{EA}}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\bar{\mathbf{K}}_s^e = k_{\bar{x}} L \begin{bmatrix} \frac{1}{3} & \frac{1}{6} \\ \frac{1}{6} & \frac{1}{3} \end{bmatrix}$$

where the axial stiffness  $D_{EA}$  and the length  $L$  are given by

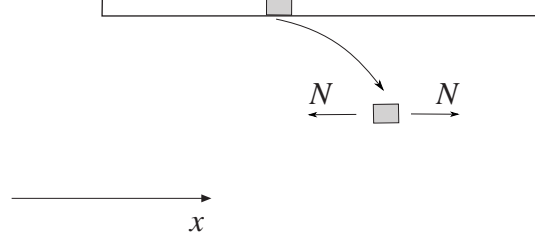
$$D_{EA} = EA; \quad L = x_2 - x_1$$

The element load vector  $\bar{\mathbf{f}}_l^e$ , stored in **fe**, is computed according to

$$\bar{\mathbf{f}}_l^e = \frac{q_{\bar{x}} L}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

**Purpose:**

Compute normal force in a one dimensional bar element with elastic support.

**Syntax:**

```
es=bar1ws(ex,ep,ed)
es=bar1ws(ex,ep,ed,eq)
[es,edi]=bar1ws(ex,ep,ed,eq,n)
[es,edi,eci]=bar1ws(ex,ep,ed,eq,n)
```

**Description:**

**bar1ws** computes the normal force in the one dimensional bar element **bar1we**.

The input variables **ex** and **ep** are defined in **bar1we** and the element nodal displacements, stored in **ed**, are obtained by the function **extract**. If distributed load is applied to the element, the variable **eq** must be included. The number of evaluation points for normal force and displacement are determined by **n**. If **n** is omitted, only the ends of the bar are evaluated.

The output variables

$$\mathbf{es} = \begin{bmatrix} N(0) \\ N(\bar{x}_2) \\ \vdots \\ N(\bar{x}_{n-1}) \\ N(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} u(0) \\ u(\bar{x}_2) \\ \vdots \\ u(\bar{x}_{n-1}) \\ u(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the normal force, the displacement, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the bar element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \end{bmatrix}$$

The transpose of  $\bar{\mathbf{a}}^e$  is stored in **ed**.

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}\bar{\mathbf{a}}^e + N_p(\bar{x})$$



where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}^{-1} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix}$$

$$u_p(\bar{x}) = \frac{k_{\bar{x}}}{D_{EA}} \begin{bmatrix} \frac{\bar{x}^2 - L\bar{x}}{2} & \frac{\bar{x}^3 - L^2\bar{x}}{6} \end{bmatrix} \mathbf{C}^{-1} \bar{\mathbf{a}}^e - \frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

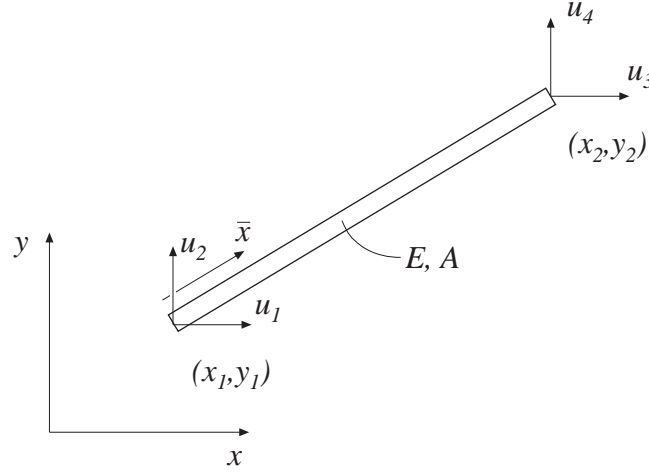
$$N_p(\bar{x}) = k_{\bar{x}} \begin{bmatrix} \frac{2\bar{x} - L}{2} & \frac{3\bar{x}^2 - L^2}{6} \end{bmatrix} \mathbf{C}^{-1} \bar{\mathbf{a}}^e - q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EA}$ ,  $L$ ,  $k_{\bar{x}}$  and  $q_{\bar{x}}$  are defined in bar1we and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a two dimensional bar element.

**Syntax:**

```
Ke=bar2e(ex,ey,ep)
[Ke,fe]=bar2e(ex,ey,ep,eq)
```

**Description:**

bar2e provides the global element stiffness matrix **Ke** for a two dimensional bar element.

The input variables

$$\begin{aligned} \text{ex} &= \begin{bmatrix} x_1 & x_2 \end{bmatrix} \\ \text{ey} &= \begin{bmatrix} y_1 & y_2 \end{bmatrix} \end{aligned} \quad \text{ep} = \begin{bmatrix} E & A \end{bmatrix}$$

supply the element nodal coordinates  $x_1$ ,  $y_1$ ,  $x_2$ , and  $y_2$ , the modulus of elasticity  $E$ , and the cross section area  $A$ .

The element load vector **fe** can also be computed if uniformly distributed axial load is applied to the element. The optional input variable

$$\text{eq} = \begin{bmatrix} q_{\bar{x}} \end{bmatrix}$$

then contains the distributed load per unit length,  $q_{\bar{x}}$ .

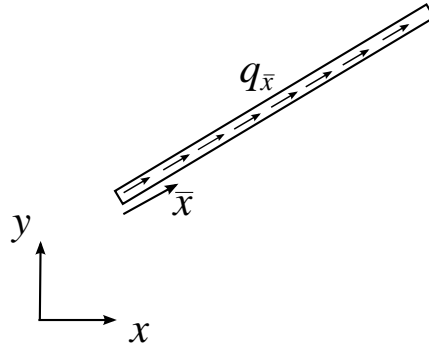
**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in **Ke**, is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \frac{D_{EA}}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad \mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & 0 & 0 \\ 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} \end{bmatrix}$$



where the axial stiffness  $D_{EA}$  and the length  $L$  are given by

$$D_{EA} = EA; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

and the transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = \frac{y_2 - y_1}{L}$$

The element load vector  $\mathbf{f}_l^e$ , stored in `fe`, is computed according to

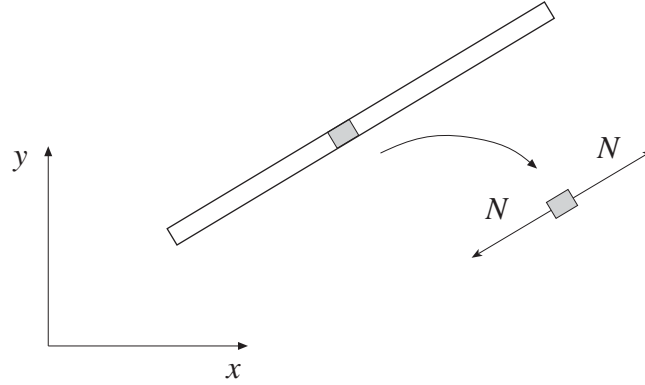
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = \frac{q_{\bar{x}} L}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

**Purpose:**

Compute normal force in a two dimensional bar element.

**Syntax:**

```

es=bar2s(ex,ey,ep,ed)
es=bar2s(ex,ey,ep,ed,eq)
[es,edi]=bar2s(ex,ey,ep,ed,eq,n)
[es,edi,eci]=bar2s(ex,ey,ep,ed,eq,n)

```

**Description:**

**bar2s** computes the normal force in the two dimensional bar element **bar2e**.

The input variables **ex**, **ey**, and **ep** are defined in **bar2e** and the element nodal displacements, stored in **ed**, are obtained by the function **extract**. If distributed loads are applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the bar are evaluated.

The output variables

$$\text{es} = \begin{bmatrix} N(0) \\ N(\bar{x}_2) \\ \vdots \\ N(\bar{x}_{n-1}) \\ N(L) \end{bmatrix} \quad \text{edi} = \begin{bmatrix} u(0) \\ u(\bar{x}_2) \\ \vdots \\ u(\bar{x}_{n-1}) \\ u(L) \end{bmatrix} \quad \text{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the normal force, the displacement, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the bar element.

**Theory:**

The nodal displacements in global coordinates

$$\mathbf{a}^e = [u_1 \ u_2 \ u_3 \ u_4]^T$$

are also shown in **bar2e**. The transpose of  $\mathbf{a}^e$  is stored in **ed**.

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \mathbf{G}\mathbf{a}^e$$

where the transformation matrix  $\mathbf{G}$  is defined in bar2e.

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}\bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}^{-1} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix}$$

$$u_p(\bar{x}) = -\frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

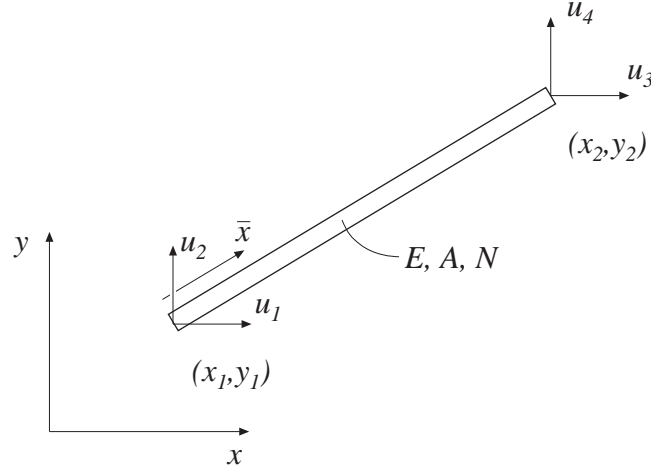
$$N_p(\bar{x}) = -q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

where  $D_{EA}$ ,  $L$ ,  $q_{\bar{x}}$  are defined in bar2e and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a two dimensional geometric nonlinear bar.

**Syntax:**

$\mathbf{K}_e = \text{bar2ge}(\text{ex}, \text{ey}, \text{ep}, \text{Qx})$

**Description:**

bar2ge provides the element stiffness matrix  $\mathbf{K}_e$  for a two dimensional geometric nonlinear bar element.

The input variables

$$\begin{aligned} \text{ex} &= \begin{bmatrix} x_1 & x_2 \end{bmatrix} \\ \text{ey} &= \begin{bmatrix} y_1 & y_2 \end{bmatrix} \end{aligned} \quad \text{ep} = \begin{bmatrix} E & A \end{bmatrix}$$

supply the element nodal coordinates  $x_1$ ,  $y_1$ ,  $x_2$ , and  $y_2$ , the modulus of elasticity  $E$ , and the cross section area  $A$ . The input variable

$$\text{Qx} = \begin{bmatrix} Q_{\bar{x}} \end{bmatrix}$$

contains the value of the axial force, which is positive in tension.

**Theory:**

The global element stiffness matrix  $\mathbf{K}^e$ , stored in  $\mathbf{K}_e$ , is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \frac{D_{EA}}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} + \frac{Q_{\bar{x}}}{L} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

$$\mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & 0 & 0 \\ n_{x\bar{y}} & n_{y\bar{y}} & 0 & 0 \\ 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} \\ 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} \end{bmatrix}$$

where the axial stiffness  $D_{EA}$  and the length  $L$  are given by

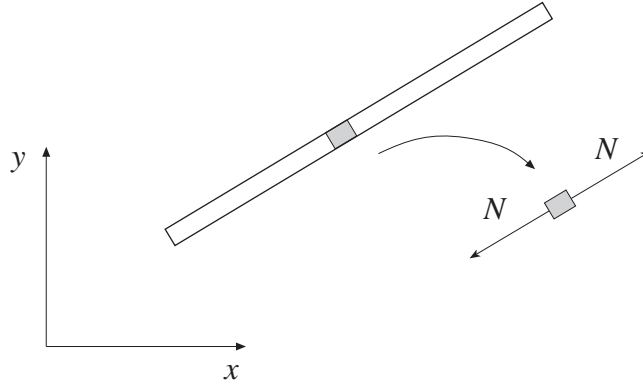
$$D_{EA} = EA; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

and the transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = n_{y\bar{y}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = -n_{x\bar{y}} = \frac{y_2 - y_1}{L}$$

**Purpose:**

Compute axial force and normal force in a two dimensional bar element.

**Syntax:**

```
[es,Qx]=bar2gs(ex,ey,ep,ed)
[es,Qx]=bar2gs(ex,ey,ep,ed,eq)
[es,Qx,edi]=bar2gs(ex,ey,ep,ed,eq,n)
[es,Qx,edi,eci]=bar2gs(ex,ey,ep,ed,eq,n)
```

**Description:**

**bar2gs** computes the normal force in the two dimensional bar elements **bar2g**.

The input variables **ex**, **ey**, and **ep** are defined in **bar2ge** and the element nodal displacements, stored in **ed**, are obtained by the function **extract**. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the bar are evaluated.

The output variable **Qx** contains the axial force  $Q_{\bar{x}}$  and the output variables

$$\mathbf{es} = \begin{bmatrix} N(0) \\ N(\bar{x}_2) \\ \vdots \\ N(\bar{x}_{n-1}) \\ N(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} u(0) \\ u(\bar{x}_2) \\ \vdots \\ u(\bar{x}_{n-1}) \\ u(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the normal force, the displacement, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the bar element.

**Theory:**

The nodal displacements in global coordinates are given by

$$\mathbf{a}^e = [u_1 \ u_2 \ u_3 \ u_4]^T$$

The transpose of  $\mathbf{a}^e$  is stored in **ed**. The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \mathbf{G}\mathbf{a}^e$$



where the transformation matrix  $\mathbf{G}$  is defined in **bar2ge**. The displacements associated with bar action are determined as

$$\bar{\mathbf{a}}_{\text{bar}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_3 \end{bmatrix}$$

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}_{\text{bar}}^e$$

$$N(\bar{x}) = D_{EA}\mathbf{B}\bar{\mathbf{a}}_{\text{bar}}^e$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}^{-1} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix}$$

where  $D_{EA}$  and  $L$  are defined in **bar2ge** and

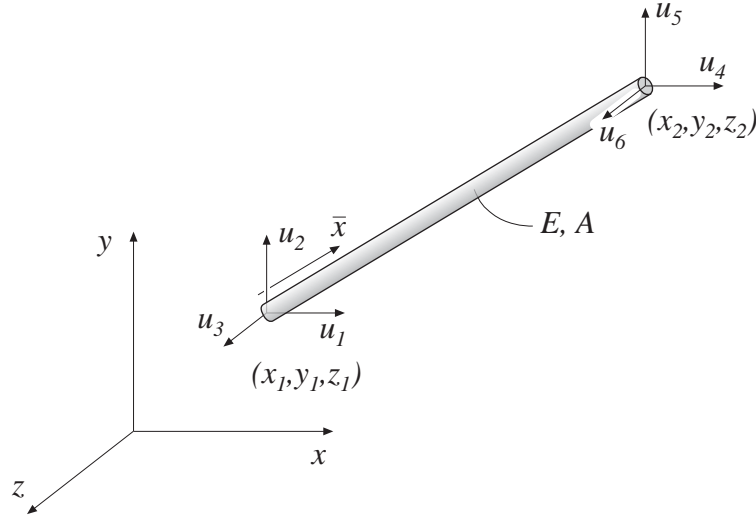
$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

An updated value of the axial force is computed as

$$Q_{\bar{x}} = N(0)$$

**Purpose:**

Compute element stiffness matrix for a three dimensional bar element.

**Syntax:**

```
Ke=bar3e(ex,ey,ez,ep)
[Ke,fe]=bar3e(ex,ey,ez,ep,eq)
```

**Description:**

bar3e provides the global element stiffness matrix  $\mathbf{K}_e$  for a three dimensional bar element.

The input variables

$$\begin{aligned} \mathbf{ex} &= \begin{bmatrix} x_1 & x_2 \end{bmatrix} \\ \mathbf{ey} &= \begin{bmatrix} y_1 & y_2 \end{bmatrix} \\ \mathbf{ez} &= \begin{bmatrix} z_1 & z_2 \end{bmatrix} \end{aligned} \quad \mathbf{ep} = \begin{bmatrix} E & A \end{bmatrix}$$

supply the element nodal coordinates  $x_1$ ,  $y_1$ ,  $z_1$ ,  $x_2$ ,  $y_2$ , and  $z_2$ , the modulus of elasticity  $E$ , and the cross section area  $A$ .

The element load vector  $\mathbf{fe}$  can also be computed if uniformly distributed axial load is applied to the element. The optional input variable

$$\mathbf{eq} = \begin{bmatrix} q_{\bar{x}} \end{bmatrix}$$

then contains the distributed load per unit length,  $q_{\bar{x}}$ .

**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in  $\mathbf{K}_e$ , is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \frac{D_{EA}}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad \mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} \end{bmatrix}$$

where the axial stiffness  $D_{EA}$  and the length  $L$  are given by

$$D_{EA} = EA; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2 + (z_2 - z_1)^2}$$

and the transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = \frac{y_2 - y_1}{L} \quad n_{z\bar{x}} = \frac{z_2 - z_1}{L}$$

The element load vector  $\mathbf{f}_l^e$ , stored in  $\mathbf{fe}$ , is computed according to

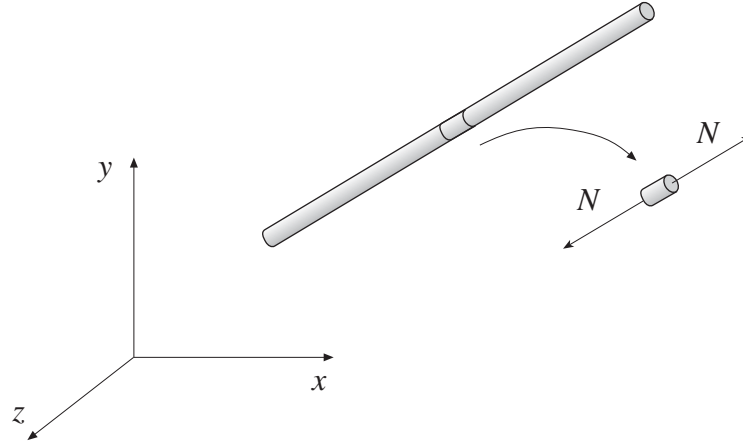
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = \frac{q_{\bar{x}} L}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

**Purpose:**

Compute normal force in a three dimensional bar element.

**Syntax:**

```
es=bar3s(ex,ey,e,ed)
es=bar3s(ex,ey,e,ed,eq)
[es,edi]=bar3s(ex,ey,e,ed,eq,n)
[es,edi,eci]=bar3s(ex,ey,e,ed,eq,n)
```

**Description:**

**bar3s** computes the normal force in a three dimensional bar element **bar3e**.

The input variables **ex**, **ey**, and **ep** are defined in **bar3e** and the element nodal displacements, stored in **ed**, are obtained by the function **extract**. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the bar are evaluated.

The output variables

$$\text{es} = \begin{bmatrix} N(0) \\ N(\bar{x}_2) \\ \vdots \\ N(\bar{x}_{n-1}) \\ N(L) \end{bmatrix} \quad
 \text{edi} = \begin{bmatrix} u(0) \\ u(\bar{x}_2) \\ \vdots \\ u(\bar{x}_{n-1}) \\ u(L) \end{bmatrix} \quad
 \text{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the normal force, the displacement, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the bar element.

**Theory:**

The nodal displacements in global coordinates are given by

$$\mathbf{a}^e = [u_1 \ u_2 \ u_3 \ u_4 \ u_5 \ u_6]^T$$

The transpose of  $\mathbf{a}^e$  is stored in **ed**.

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \mathbf{G}\mathbf{a}^e$$

where the transformation matrix  $\mathbf{G}$  is defined in **bar3e**.

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}\bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}^{-1} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix}$$

$$u_p(\bar{x}) = -\frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

$$N_p(\bar{x}) = -q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

where  $D_{EA}$ ,  $L$ ,  $q_{\bar{x}}$  are defined in **bar3e** and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

---

## 4.4 Beam elements

Beam elements are available for one, two, and three dimensional linear static analysis. Two dimensional beam elements for nonlinear geometric analysis are also available.

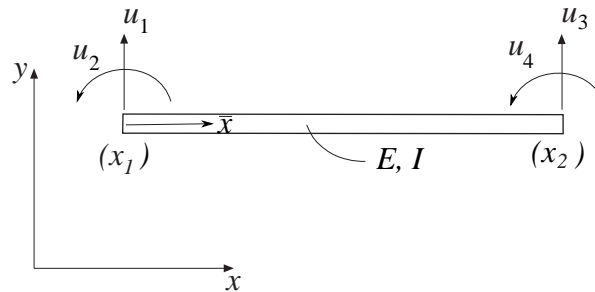
| 1D beam elements |   |
|------------------|---|
| beam1e           | Compute element matrices  |
| beam1s           | Compute section forces  |
| beam1we          | Compute element matrices for beam element on elastic foundation |
| beam1ws          | Compute section forces for beam element on elastic foundation   |

| 2D beam elements |   |
|------------------|---|
| beam2e           | Compute element matrices  |
| beam2s           | Compute section forces  |
| beam2we          | Compute element matrices for beam element on elastic foundation |
| beam2ws          | Compute section forces for beam element on elastic foundation   |
| beam2ge          | Compute element matrices for geometric nonlinear beam element   |
| beam2gs          | Compute section forces for geometric nonlinear beam element     |

| 3D beam elements |                          |
|------------------|--------------------------|
| beam3e           | Compute element matrices |
| beam3s           | Compute section forces   |

**Purpose:**

Compute element stiffness matrix for a one dimensional beam element.

**Syntax:**

```
Ke=beam1e(ex,ep)
[Ke,fe]=beam1e(ex,ep,eq)
```

**Description:**

beam1e provides the global element stiffness matrix **Ke** for a one dimensional beam element.

The input variables

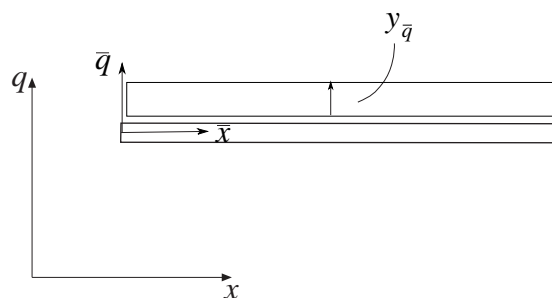
$$\mathbf{ex} = [x_1 \ x_2] \quad \mathbf{ep} = [E \ I]$$

supply the element nodal coordinates  $x_1$  and  $x_2$ , the modulus of elasticity  $E$  and the moment of inertia  $I$ .

The element load vector **fe** can also be computed if uniformly distributed load is applied to the element. The optional input variable

$$\mathbf{eq} = [q_{\bar{y}}]$$

then contains the distributed load per unit length,  $q_{\bar{y}}$ .



**Theory:**

The element stiffness matrix  $\bar{\mathbf{K}}^e$ , stored in  $\mathbf{Ke}$ , is computed according to

$$\bar{\mathbf{K}}^e = \frac{D_{EI}}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

where the bending stiffness  $D_{EI}$  and the length  $L$  are given by

$$D_{EI} = EI; \quad L = x_2 - x_1$$

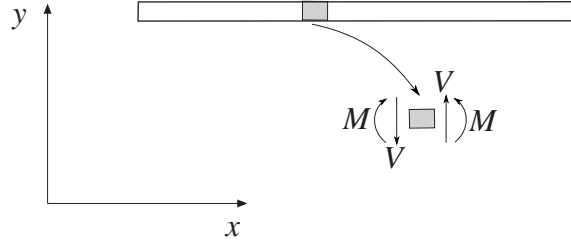
The element loads  $\bar{\mathbf{f}}_l^e$  stored in the variable  $\mathbf{fe}$  are computed according to

$$\bar{\mathbf{f}}_l^e = q_{\bar{y}} \begin{bmatrix} \frac{L}{2} \\ \frac{L^2}{12} \\ \frac{L}{2} \\ -\frac{L^2}{12} \end{bmatrix}$$



**Purpose:**

Compute section forces in a one dimensional beam element.

**Syntax:**

```
es=beam1s(ex,ep,ed)
es=beam1s(ex,ep,ed,eq)
[es,edi,eci]=beam1s(ex,ep,ed,eq,n))
```

**Description:**

**beam1s** computes the section forces and displacements in local directions along the beam element **beam1e**.

The input variables **ex**, **ep** and **eq** are defined in **beam1e**, and the element displacements, stored in **ed**, are obtained by the function **extract**. If distributed loads are applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variables

$$\mathbf{es} = \begin{bmatrix} V(0) & M(0) \\ V(\bar{x}_2) & M(\bar{x}_2) \\ \vdots & \vdots \\ V(\bar{x}_{n-1}) & M(\bar{x}_{n-1}) \\ V(L) & M(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} v(0) \\ v(\bar{x}_2) \\ \vdots \\ v(\bar{x}_{n-1}) \\ v(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix}$$

where the transpose of  $\bar{\mathbf{a}}^e$  is stored in **ed**.

The displacement  $v(\bar{x})$ , the bending moment  $M(\bar{x})$  and the shear force  $V(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + v_p(\bar{x})$$

$$M(\bar{x}) = D_{EI}\mathbf{B}\bar{\mathbf{a}}^e + M_p(\bar{x})$$

$$V(\bar{x}) = -D_{EI}\frac{d\mathbf{B}}{dx}\bar{\mathbf{a}}^e + V_p(\bar{x})$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}^{-1}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}^{-1}$$

$$\frac{d\mathbf{B}}{dx} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}^{-1}$$

$$v_p(\bar{x}) = \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_p(\bar{x}) = q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

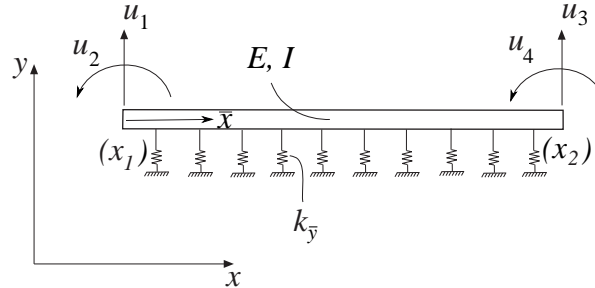
$$V_p(\bar{x}) = -q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in **beam1e** and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a one dimensional beam element on elastic support.

**Syntax:**

```
Ke=beam1we(ex,ep)
[Ke,fe]=beam1we(ex,ep,eq)
```

**Description:**

beam1we provides the global element stiffness matrix **Ke** for a one dimensional beam element with elastic support.

The input variables

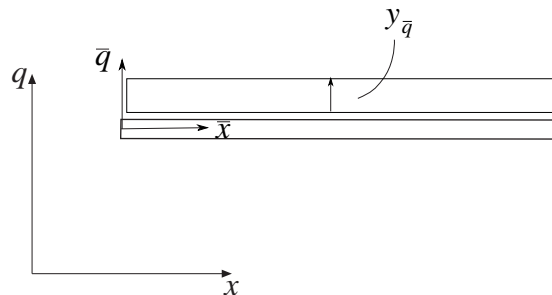
$$\mathbf{ex} = [x_1 \ x_2] \quad \mathbf{ep} = [E \ I \ k_{\bar{y}}]$$

supply the element nodal coordinates  $x_1$  and  $x_2$ , the modulus of elasticity  $E$ , the moment of inertia  $I$ , and the spring stiffness in the transverse direction  $k_{\bar{y}}$ .

The element load vector **fe** can also be computed if uniformly distributed load is applied to the element. The optional input variable

$$\mathbf{eq} = [q_{\bar{y}}]$$

then contains the distributed load per unit length,  $q_{\bar{y}}$ .



**Theory:**

The element stiffness matrix  $\bar{\mathbf{K}}^e$ , stored in  $\mathbf{Ke}$ , is computed according to

$$\bar{\mathbf{K}}^e = \bar{\mathbf{K}}_0^e + \bar{\mathbf{K}}_s^e$$

$$\bar{\mathbf{K}}_0^e = \frac{D_{EI}}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

$$\bar{\mathbf{K}}_s^e = \frac{k_y L}{420} \begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix}$$

where the bending stiffness  $D_{EI}$  and the length  $L$  are given by

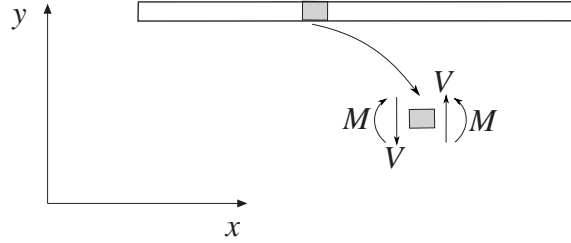
$$D_{EI} = EI; \quad L = x_2 - x_1$$

The element loads  $\bar{\mathbf{f}}_l^e$  stored in the variable  $\mathbf{fe}$  are computed according to

$$\bar{\mathbf{f}}_l^e = q_y \begin{bmatrix} \frac{L}{2} \\ L^2 \\ \frac{12}{L} \\ \frac{L}{2} \\ L^2 \\ -\frac{12}{L} \end{bmatrix}$$

**Purpose:**

Compute section forces in a one dimensional beam element with elastic support.

**Syntax:**

```
es=beam1ws(ex,ep,ed)
es=beam1ws(ex,ep,ed,eq)
[es,edi,eci]=beam1ws(ex,ep,ed,eq,n))
```

**Description:**

**beam1ws** computes the section forces and displacements in local directions along the beam element **beam1we**.

The input variables **ex**, **ep** and **eq** are defined in **beam1we**, and the element displacements, stored in **ed**, are obtained by the function **extract**. If distributed loads are applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variables

$$\mathbf{es} = \begin{bmatrix} V(0) & M(0) \\ V(\bar{x}_2) & M(\bar{x}_2) \\ \vdots & \vdots \\ V(\bar{x}_{n-1}) & M(\bar{x}_{n-1}) \\ V(L) & M(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} v(0) \\ v(\bar{x}_2) \\ \vdots \\ v(\bar{x}_{n-1}) \\ v(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \end{bmatrix}$$

where the transpose of  $\bar{\mathbf{a}}^e$  is stored in **ed**.

The displacement  $v(\bar{x})$ , the bending moment  $M(\bar{x})$  and the shear force  $V(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}\bar{\mathbf{a}}^e + v_p(\bar{x})$$

$$M(\bar{x}) = D_{EI}\mathbf{B}\bar{\mathbf{a}}^e + M_p(\bar{x})$$

$$V(\bar{x}) = -D_{EI}\frac{d\mathbf{B}}{dx}\bar{\mathbf{a}}^e + V_p(\bar{x})$$

where

$$\mathbf{N} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}^{-1}$$

$$\mathbf{B} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}^{-1}$$

$$\frac{d\mathbf{B}}{dx} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}^{-1}$$

$$v_p(\bar{x}) = -\frac{k_{\bar{y}}}{D_{EI}} \begin{bmatrix} \frac{\bar{x}^4 - 2L\bar{x}^3 + L^2\bar{x}^2}{24} \\ \frac{\bar{x}^5 - 3L^2\bar{x}^3 + 2L^3\bar{x}^2}{120} \\ \frac{\bar{x}^6 - 4L^3\bar{x}^3 + 3L^4\bar{x}^2}{360} \\ \frac{\bar{x}^7 - 5L^4\bar{x}^3 + 4L^5\bar{x}^2}{840} \end{bmatrix}^T \mathbf{C}^{-1}\bar{\mathbf{a}}^e + \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_p(\bar{x}) = -k_{\bar{y}} \begin{bmatrix} \frac{6\bar{x}^2 - 6L\bar{x} + L^2}{12} \\ \frac{10\bar{x}^3 - 9L^2\bar{x} + 2L^3}{60} \\ \frac{5\bar{x}^4 - 4L^3\bar{x} + L^4}{60} \\ \frac{21\bar{x}^5 - 15L^4\bar{x} + 4L^5}{420} \end{bmatrix}^T \mathbf{C}^{-1}\bar{\mathbf{a}}^e + q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

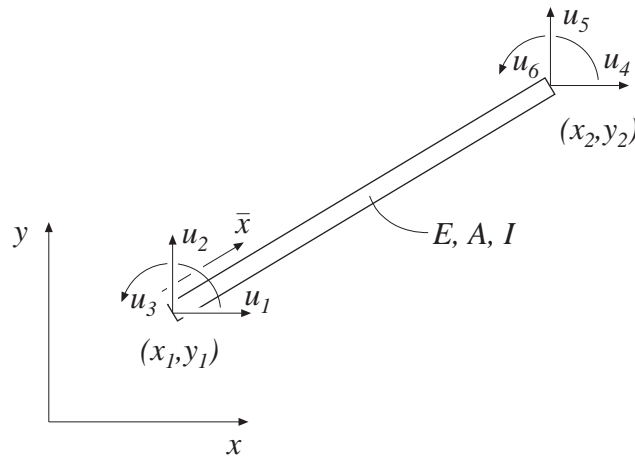
$$V_p(\bar{x}) = k_{\bar{y}} \begin{bmatrix} \frac{2\bar{x} - L}{2} \\ \frac{10\bar{x}^2 - 3L^2}{20} \\ \frac{5\bar{x}^3 - L^3}{15} \\ \frac{7\bar{x}^4 - L^4}{28} \end{bmatrix}^T \mathbf{C}^{-1}\bar{\mathbf{a}}^e - q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI}$ ,  $k_{\bar{y}}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in beam1we and

$$\mathbf{C}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a two dimensional beam element.

**Syntax:**

```
Ke=beam2e(ex,ey,ep)
[Ke,fe]=beam2e(ex,ey,ep,eq)
```

beam2e provides the global element stiffness matrix **Ke** for a two dimensional beam element.

The input variables

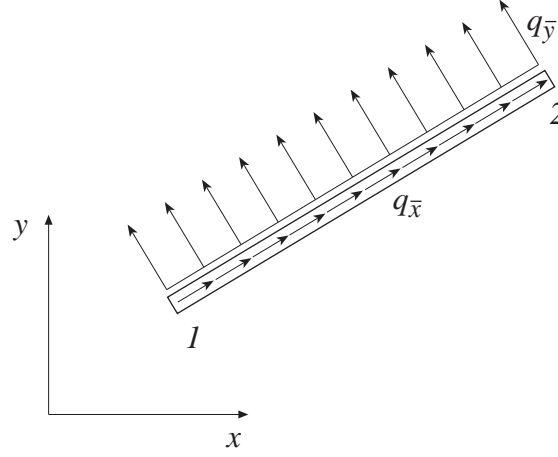
$$\begin{aligned} \mathbf{ex} &= \begin{bmatrix} x_1 & x_2 \\ y_1 & y_2 \end{bmatrix} & \mathbf{ep} &= \begin{bmatrix} E & A & I \end{bmatrix} \end{aligned}$$

supply the element nodal coordinates  $x_1$ ,  $y_1$ ,  $x_2$ , and  $y_2$ , the modulus of elasticity  $E$ , the cross section area  $A$ , and the moment of inertia  $I$ .

The element load vector **fe** can also be computed if a uniformly distributed transverse load is applied to the element. The optional input variable

$$\mathbf{eq} = \begin{bmatrix} q_{\bar{x}} & q_{\bar{y}} \end{bmatrix}$$

then contains the distributed loads per unit length,  $q_{\bar{x}}$  and  $q_{\bar{y}}$ .

**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in  $\mathbf{Ke}$ , is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \begin{bmatrix} \frac{D_{EA}}{L} & 0 & 0 & -\frac{D_{EA}}{L} & 0 & 0 \\ 0 & \frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} & 0 & -\frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} \\ -\frac{D_{EA}}{L} & 0 & 0 & \frac{D_{EA}}{L} & 0 & 0 \\ 0 & -\frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} & 0 & \frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} \end{bmatrix}$$

$$\mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & 0 & 0 & 0 & 0 \\ n_{x\bar{y}} & n_{y\bar{y}} & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & 0 \\ 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

where the axial stiffness  $D_{EA}$ , the bending stiffness  $D_{EI}$  and the length  $L$  are given

by

$$D_{EA} = EA; \quad D_{EI} = EI; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

The transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = n_{y\bar{y}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = -n_{x\bar{y}} = \frac{y_2 - y_1}{L}$$



The element loads  $\mathbf{f}_l^e$  stored in the variable  $\mathbf{fe}$  are computed according to

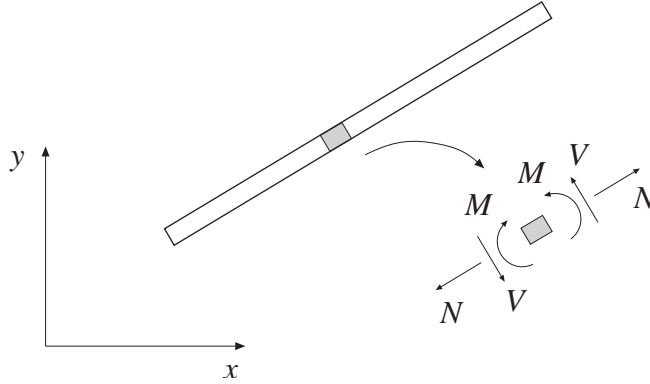
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = \begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_y L}{2} \\ \frac{q_y L^2}{12} \\ \frac{q_x L}{2} \\ \frac{q_y L}{2} \\ -\frac{q_y L^2}{12} \end{bmatrix}$$

**Purpose:**

Compute section forces in a two dimensional beam element.

**Syntax:**

```
[es]=beam2s(ex,ey,ep,ed)
[es]=beam2s(ex,ey,ep,ed,eq)
[es,edi]=beam2s(ex,ey,ep,ed,eq,n)
[es,edi,eci]=beam2s(ex,ey,ep,ed,eq,n)
```

**Description:**

**beam2s** computes the section forces and displacements in local directions along the beam element **beam2e**.

The input variables **ex**, **ey**, **ep**, and **eq** are defined in **beam2e**.

The element displacements, stored in **ed**, are obtained by the function **extract**. If a distributed load is applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variables

$$\mathbf{es} = \begin{bmatrix} N(0) & V(0) & M(0) \\ N(\bar{x}_2) & V(\bar{x}_2) & M(\bar{x}_2) \\ \vdots & \vdots & \vdots \\ N(\bar{x}_{n-1}) & V(\bar{x}_{n-1}) & M(\bar{x}_{n-1}) \\ N(L) & V(L) & M(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} u(0) & v(0) \\ u(\bar{x}_2) & v(\bar{x}_2) \\ \vdots & \vdots \\ u(\bar{x}_{n-1}) & v(\bar{x}_{n-1}) \\ u(L) & v(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix} = \mathbf{Ga}^e$$

where  $\mathbf{G}$  is described in **beam2e** and the transpose of  $\mathbf{a}^e$  is stored in **ed**. The displacements associated with bar action and beam action are determined as

$$\bar{\mathbf{a}}_{\text{bar}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_4 \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{beam}}^e = \begin{bmatrix} \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix}$$

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}_{\text{bar}} \bar{\mathbf{a}}_{\text{bar}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA} \mathbf{B}_{\text{bar}} \bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N}_{\text{bar}} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B}_{\text{bar}} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

$$u_p(\bar{x}) = -\frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

$$N_p(\bar{x}) = -q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EA}$ ,  $L$ , and  $q_{\bar{x}}$  are defined in **beam2e** and

$$\mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

The displacement  $v(\bar{x})$ , the bending moment  $M(\bar{x})$  and the shear force  $V(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam}}^e + v_p(\bar{x})$$

$$M(\bar{x}) = D_{EI} \mathbf{B}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam}}^e + M_p(\bar{x})$$

$$V(\bar{x}) = -D_{EI} \frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} \bar{\mathbf{a}}_{\text{beam}}^e + V_p(\bar{x})$$

where

$$\mathbf{N}_{\text{beam}} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\mathbf{B}_{\text{beam}} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$v_p(\bar{x}) = \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_p(\bar{x}) = q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

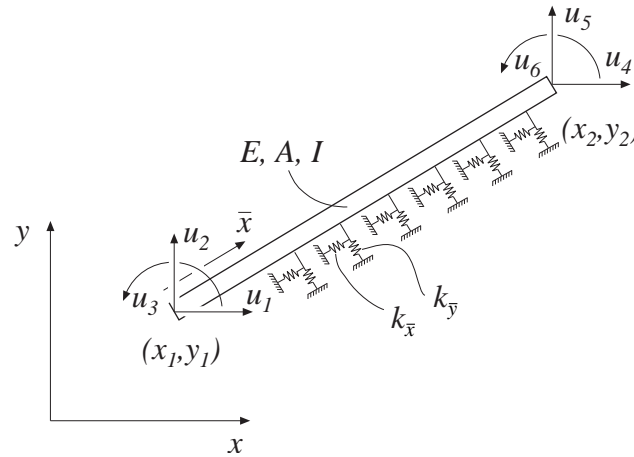
$$V_p(\bar{x}) = -q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in beam2e and

$$\mathbf{C}_{\text{beam}}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a two dimensional beam element on elastic support.

**Syntax:**

```
Ke=beam2we(ex,ey,ep)
[Ke,fe]=beam2we(ex,ey,ep,eq)
```

**Description:**

beam2we provides the global element stiffness matrix **Ke** for a two dimensional beam element with elastic support.

The input variables

$$\mathbf{ex} = [x_1 \ x_2] \quad \mathbf{ey} = [y_1 \ y_2] \quad \mathbf{ep} = [E \ A \ I \ k_{\bar{x}} \ k_{\bar{y}}]$$

supply the element nodal coordinates  $x_1$ ,  $x_2$ ,  $y_1$ , and  $y_2$ , the modulus of elasticity  $E$ , the cross section area  $A$ , the moment of inertia  $I$ , the spring stiffness in the axial direction  $k_{\bar{x}}$ , and the spring stiffness in the transverse direction  $k_{\bar{y}}$ .

The element load vector **fe** can also be computed if uniformly distributed loads are applied to the element. The optional input variable

$$\mathbf{eq} = [q_{\bar{x}} \ q_{\bar{y}}]$$

then contains the distributed load per unit length,  $q_{\bar{x}}$  and  $q_{\bar{y}}$ .

**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in  $\mathbf{Ke}$ , is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \bar{\mathbf{K}}_0^e + \bar{\mathbf{K}}_s^e$$

$$\bar{\mathbf{K}}_0^e = \begin{bmatrix} \frac{D_{EA}}{L} & 0 & 0 & -\frac{D_{EA}}{L} & 0 & 0 \\ 0 & \frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} & 0 & -\frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} \\ -\frac{D_{EA}}{L} & 0 & 0 & \frac{D_{EA}}{L} & 0 & 0 \\ 0 & -\frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} & 0 & \frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} \end{bmatrix}$$

$$\bar{\mathbf{K}}_s^e = \frac{L}{420} \begin{bmatrix} 140k_{\bar{x}} & 0 & 0 & 70k_{\bar{x}} & 0 & 0 \\ 0 & 156k_{\bar{y}} & 22k_{\bar{y}}L & 0 & 54k_{\bar{y}} & -13k_{\bar{y}}L \\ 0 & 22k_{\bar{y}}L & 4k_{\bar{y}}L^2 & 0 & 13k_{\bar{y}}L & -3k_{\bar{y}}L^2 \\ 70k_{\bar{x}} & 0 & 0 & 140k_{\bar{x}} & 0 & 0 \\ 0 & 54k_{\bar{y}} & 13k_{\bar{y}}L & 0 & 156k_{\bar{y}} & -22k_{\bar{y}}L \\ 0 & -13k_{\bar{y}}L & -3k_{\bar{y}}L^2 & 0 & -22k_{\bar{y}}L & 4k_{\bar{y}}L^2 \end{bmatrix}$$

$$\mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & 0 & 0 & 0 & 0 \\ n_{x\bar{y}} & n_{y\bar{y}} & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & 0 \\ 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

where the axial stiffness  $D_{EA}$ , the bending stiffness  $D_{EI}$  and the length  $L$  are given by

$$D_{EA} = EA; \quad D_{EI} = EI; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

The transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = n_{y\bar{y}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = -n_{x\bar{y}} = \frac{y_2 - y_1}{L}$$

The element loads  $\mathbf{f}_l^e$  stored in the variable  $\mathbf{fe}$  are computed according to

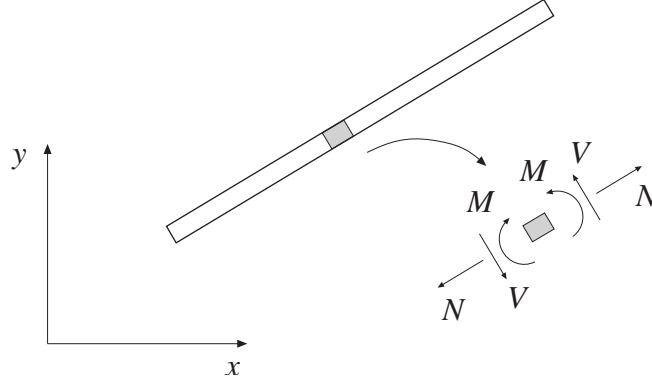
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = \begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_y L}{2} \\ \frac{q_y L^2}{12} \\ \frac{q_x L}{2} \\ \frac{q_y L}{2} \\ -\frac{q_y L^2}{12} \end{bmatrix}$$

**Purpose:**

Compute section forces in a two dimensional beam element with elastic support.

**Syntax:**

```
es=beam2ws(ex,ey,ep,ed)
es=beam2ws(ex,ey,ep,ed,eq)
[es,edi,eci]=beam2ws(ex,ey,ep,ed,eq,n)
```

**Description:**

**beam2ws** computes the section forces and displacements in local directions along the beam element **beam2we**.

The input variables **ex**, **ey**, **ep** and **eq** are defined in **beam2we**, and the element displacements, stored in **ed**, are obtained by the function **extract**. If distributed loads are applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variables

$$\text{es} = \begin{bmatrix} N(0) & V(0) & M(0) \\ N(\bar{x}_2) & V(\bar{x}_2) & M(\bar{x}_2) \\ \vdots & \vdots & \vdots \\ N(\bar{x}_{n-1}) & V(\bar{x}_{n-1}) & M(\bar{x}_{n-1}) \\ N(L) & V(L) & M(L) \end{bmatrix} \quad
 \text{edi} = \begin{bmatrix} u(0) & v(0) \\ u(\bar{x}_2) & v(\bar{x}_2) \\ \vdots & \vdots \\ u(\bar{x}_{n-1}) & v(\bar{x}_{n-1}) \\ u(L) & v(L) \end{bmatrix} \quad
 \text{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

**Theory:**



The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix} = \mathbf{G}\mathbf{a}^e$$

where  $\mathbf{G}$  is described in `beam2we` and the transpose of  $\mathbf{a}^e$  is stored in `ed`. The displacements associated with bar action and beam action are determined as

$$\bar{\mathbf{a}}_{\text{bar}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_4 \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{beam}}^e = \begin{bmatrix} \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix}$$

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}_{\text{bar}}\bar{\mathbf{a}}_{\text{bar}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}_{\text{bar}}\bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N}_{\text{bar}} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B}_{\text{bar}} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

$$u_p(\bar{x}) = \frac{k_{\bar{x}}}{D_{EA}} \begin{bmatrix} \frac{\bar{x}^2 - L\bar{x}}{2} & \frac{\bar{x}^3 - L^2\bar{x}}{6} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1}\bar{\mathbf{a}}_{\text{bar}}^e - \frac{q_{\bar{x}}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

$$N_p(\bar{x}) = k_{\bar{x}} \begin{bmatrix} \frac{2\bar{x} - L}{2} & \frac{3\bar{x}^2 - L^2}{6} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1}\bar{\mathbf{a}}_{\text{bar}}^e - q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EA}$ ,  $k_{\bar{x}}$ ,  $L$ , and  $q_{\bar{x}}$  are defined in `beam2we` and

$$\mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

The displacement  $v(\bar{x})$ , the bending moment  $M(\bar{x})$  and the shear force  $V(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}_{\text{beam}}\bar{\mathbf{a}}_{\text{beam}}^e + v_p(\bar{x})$$

$$M(\bar{x}) = D_{EI}\mathbf{B}_{\text{beam}}\bar{\mathbf{a}}_{\text{beam}}^e + M_p(\bar{x})$$

$$V(\bar{x}) = -D_{EI} \frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} \bar{\mathbf{a}}_{\text{beam}}^e + V_p(\bar{x})$$

where

$$\mathbf{N}_{\text{beam}} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\mathbf{B}_{\text{beam}} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\frac{d\mathbf{B}_{\text{beam}}}{dx} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$v_p(\bar{x}) = -\frac{k_{\bar{y}}}{D_{EI}} \begin{bmatrix} \frac{\bar{x}^4 - 2L\bar{x}^3 + L^2\bar{x}^2}{24} \\ \frac{\bar{x}^5 - 3L^2\bar{x}^3 + 2L^3\bar{x}^2}{120} \\ \frac{\bar{x}^6 - 4L^3\bar{x}^3 + 3L^4\bar{x}^2}{360} \\ \frac{\bar{x}^7 - 5L^4\bar{x}^3 + 4L^5\bar{x}^2}{840} \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e + \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_p(\bar{x}) = -k_{\bar{y}} \begin{bmatrix} \frac{6\bar{x}^2 - 6L\bar{x} + L^2}{12} \\ \frac{10\bar{x}^3 - 9L^2\bar{x} + 2L^3}{60} \\ \frac{5\bar{x}^4 - 4L^3\bar{x} + L^4}{60} \\ \frac{21\bar{x}^5 - 15L^4\bar{x} + 4L^5}{420} \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e + q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

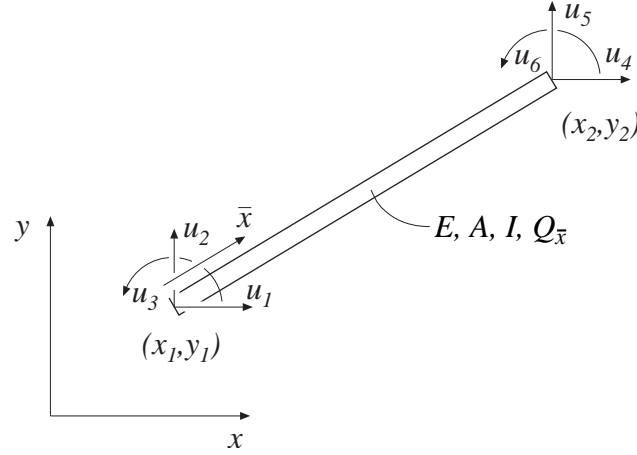
$$V_p(\bar{x}) = k_{\bar{y}} \begin{bmatrix} \frac{2\bar{x} - L}{2} \\ \frac{10\bar{x}^2 - 3L^2}{20} \\ \frac{5\bar{x}^3 - L^3}{15} \\ \frac{7\bar{x}^4 - L^4}{28} \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e - q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI}$ ,  $k_{\bar{y}}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in beam2we and

$$\mathbf{C}_{\text{beam}}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

**Purpose:**

Compute element stiffness matrix for a two dimensional nonlinear beam element with respect to geometrical nonlinearity.

**Syntax:**

```
Ke=beam2ge(ex,ey,ep,Qx)
[Ke,fe]=beam2ge(ex,ey,ep,Qx,eq)
```

**Description:**

beam2ge provides the global element stiffness matrix **Ke** for a two dimensional beam element with respect to geometrical nonlinearity.

The input variables

$$\begin{aligned} \mathbf{ex} &= \begin{bmatrix} x_1 & x_2 \end{bmatrix} & \mathbf{ep} &= \begin{bmatrix} E & A & I \end{bmatrix} \\ \mathbf{ey} &= \begin{bmatrix} y_1 & y_2 \end{bmatrix} \end{aligned}$$

supply the element nodal coordinates  $x_1$ ,  $y_1$ ,  $x_2$ , and  $y_2$ , the modulus of elasticity  $E$ , the cross section area  $A$ , and the moment of inertia  $I$  and

$$\mathbf{Qx} = \begin{bmatrix} Q_{\bar{x}} \end{bmatrix}$$

contains the value of the predefined axial force  $Q_{\bar{x}}$ , which is positive in tension.

The element load vector **fe** can also be computed if a uniformly distributed transverse load is applied to the element. The optional input variable

$$\mathbf{eq} = \begin{bmatrix} q_{\bar{y}} \end{bmatrix}$$

then contains the distributed transverse load per unit length,  $q_{\bar{y}}$ . Note that **eq** is a scalar and not a vector as in beam2e.

**Theory:**

The element stiffness matrix  $\mathbf{K}^e$ , stored in the variable **Ke**, is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where  $\bar{\mathbf{K}}^e$  is given by

$$\bar{\mathbf{K}}^e = \bar{\mathbf{K}}_0^e + \bar{\mathbf{K}}_\sigma^e$$

with

$$\bar{\mathbf{K}}_0^e = \begin{bmatrix} \frac{D_{EA}}{L} & 0 & 0 & -\frac{D_{EA}}{L} & 0 & 0 \\ 0 & \frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} & 0 & -\frac{12D_{EI}}{L^3} & \frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} \\ -\frac{D_{EA}}{L} & 0 & 0 & \frac{D_{EA}}{L} & 0 & 0 \\ 0 & -\frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} & 0 & \frac{12D_{EI}}{L^3} & -\frac{6D_{EI}}{L^2} \\ 0 & \frac{6D_{EI}}{L^2} & \frac{2D_{EI}}{L} & 0 & -\frac{6D_{EI}}{L^2} & \frac{4D_{EI}}{L} \end{bmatrix}$$

$$\bar{\mathbf{K}}_\sigma^e = Q_{\bar{x}} \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{6}{5L} & \frac{1}{10} & 0 & -\frac{6}{5L} & \frac{1}{10} \\ 0 & \frac{1}{10} & \frac{2L}{15} & 0 & -\frac{1}{10} & -\frac{L}{30} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -\frac{6}{5L} & -\frac{1}{10} & 0 & \frac{6}{5L} & -\frac{1}{10} \\ 0 & \frac{1}{10} & -\frac{L}{30} & 0 & -\frac{1}{10} & \frac{2L}{15} \end{bmatrix}$$

$$\mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & 0 & 0 & 0 & 0 \\ n_{x\bar{y}} & n_{y\bar{y}} & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & 0 \\ 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

where the axial stiffness  $D_{EA}$ , the bending stiffness  $D_{EI}$  and the length  $L$  are given by

$$D_{EA} = EA; \quad D_{EI} = EI; \quad L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

The transformation matrix  $\mathbf{G}$  contains the direction cosines

$$n_{x\bar{x}} = n_{y\bar{y}} = \frac{x_2 - x_1}{L} \quad n_{y\bar{x}} = -n_{x\bar{y}} = \frac{y_2 - y_1}{L}$$

The element loads  $\mathbf{f}_l^e$  stored in  $\mathbf{fe}$  are computed according to

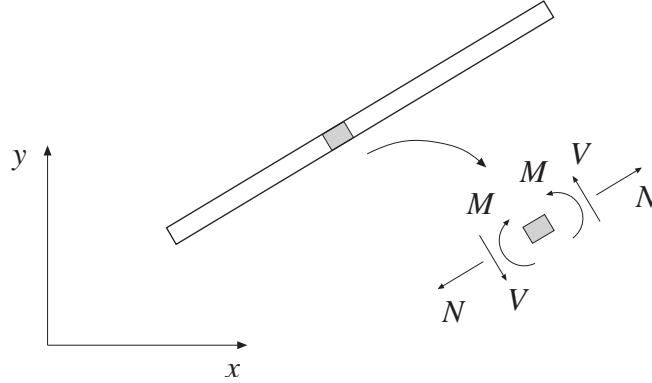
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = q_{\bar{y}} \begin{bmatrix} 0 & \frac{L}{2} & \frac{L^2}{12} & 0 & \frac{L}{2} & -\frac{L^2}{12} \end{bmatrix}^T$$

**Purpose:**

Compute section forces in a two dimensional nonlinear beam element with geometrical nonlinearity.

**Syntax:**

```
[es,Qx]=beam2gs(ex,ey,ep,ed,Qx)
[es,Qx]=beam2gs(ex,ey,ep,ed,Qx,eq)
[es,Qx,edi]=beam2gs(ex,ey,ep,ed,Qx,eq,n)
[es,Qx,edi,eci]=beam2gs(ex,ey,ep,ed,Qx,eq,n)
```

**Description:**

**beam2gs** computes the section forces and displacements in local directions along the geometric nonlinear beam element **beam2ge**.

The input variables **ex**, **ey**, **ep**, **Qx**, and **eq** are described in **beam2ge**. The element displacements, stored in **ed**, are obtained by the function **extract**. If a distributed transversal load is applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variable **Qx** contains  $Q_{\bar{x}}$  and the output variables

$$\mathbf{es} = \begin{bmatrix} N(0) & V(0) & M(0) \\ N(\bar{x}_2) & V(\bar{x}_2) & M(\bar{x}_2) \\ \vdots & \vdots & \vdots \\ N(\bar{x}_{n-1}) & V(\bar{x}_{n-1}) & M(\bar{x}_{n-1}) \\ N(L) & V(L) & M(L) \end{bmatrix} \quad \mathbf{edi} = \begin{bmatrix} u(0) & v(0) \\ u(\bar{x}_2) & v(\bar{x}_2) \\ \vdots & \vdots \\ u(\bar{x}_{n-1}) & v(\bar{x}_{n-1}) \\ u(L) & v(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

**Theory:**

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix} = \mathbf{G} \mathbf{a}^e$$

where  $\mathbf{G}$  is described in **beam2ge** and the transpose of  $\mathbf{a}^e$  is stored in **ed**. The displacements associated with bar action and beam action are determined as

$$\bar{\mathbf{a}}_{\text{bar}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_4 \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{beam}}^e = \begin{bmatrix} \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_5 \\ \bar{u}_6 \end{bmatrix}$$

The displacement  $u(\bar{x})$  is computed from

$$u(\bar{x}) = \mathbf{N}_{\text{bar}} \bar{\mathbf{a}}_{\text{bar}}^e$$

where

$$\mathbf{N}_{\text{bar}} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

where  $L$  is defined in **beam2ge** and

$$\mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

The displacement  $v(\bar{x})$ , the rotation  $\theta(\bar{x})$ , the bending moment  $M(\bar{x})$  and the shear force  $V(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam}}^e + v_p(\bar{x})$$

$$\theta(\bar{x}) = \frac{d\mathbf{N}_{\text{beam}}}{dx} \bar{\mathbf{a}}_{\text{beam}}^e + \theta_p(\bar{x})$$

$$M(\bar{x}) = D_{EI} \mathbf{B}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam}}^e + M_p(\bar{x})$$

$$V(\bar{x}) = -D_{EI} \frac{d\mathbf{B}_{\text{beam}}}{dx} \bar{\mathbf{a}}_{\text{beam}}^e + V_p(\bar{x})$$

where

$$\mathbf{N}_{\text{beam}} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\frac{d\mathbf{N}_{\text{beam}}}{dx} = \begin{bmatrix} 0 & 1 & 2\bar{x} & 3\bar{x}^2 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\mathbf{B}_{\text{beam}} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\frac{d\mathbf{B}_{\text{beam}}}{dx} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$v_p(\bar{x}) = -\frac{Q_{\bar{x}}}{D_{EI}} \begin{bmatrix} 0 \\ 0 \\ \left(\frac{\bar{x}^4}{12} - \frac{L\bar{x}^3}{6} + \frac{L^2\bar{x}^2}{12}\right) \\ \left(\frac{\bar{x}^5}{20} - \frac{3L^2\bar{x}^3}{20} + \frac{L^3\bar{x}^2}{10}\right) \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e + \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$\theta_p(\bar{x}) = -\frac{Q_{\bar{x}}}{D_{EI}} \begin{bmatrix} 0 \\ 0 \\ \left(\frac{\bar{x}^3}{3} - \frac{L\bar{x}^2}{2} + \frac{L^2\bar{x}}{6}\right) \\ \left(\frac{\bar{x}^4}{4} - \frac{9L^2\bar{x}^2}{20} + \frac{L^3\bar{x}}{5}\right) \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e + \frac{q_{\bar{y}}}{D_{EI}} \left( \frac{\bar{x}^3}{6} - \frac{L\bar{x}^2}{4} + \frac{L^2\bar{x}}{12} \right)$$

$$M_p(\bar{x}) = -Q_{\bar{x}} \begin{bmatrix} 0 \\ 0 \\ (\bar{x}^2 - L\bar{x} + \frac{L^2}{6}) \\ (\bar{x}^3 - \frac{9L^2\bar{x}}{10} + \frac{L^3}{5}) \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e + q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

$$V_p(\bar{x}) = Q_{\bar{x}} \begin{bmatrix} 0 \\ 0 \\ (2\bar{x} - L) \\ (3\bar{x}^2 - \frac{9L^2}{10}) \end{bmatrix}^T \mathbf{C}_{\text{beam}}^{-1} \bar{\mathbf{a}}_{\text{beam}}^e - q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in `beam2ge` and

$$\mathbf{C}_{\text{beam}}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

An updated value of the axial force is computed as

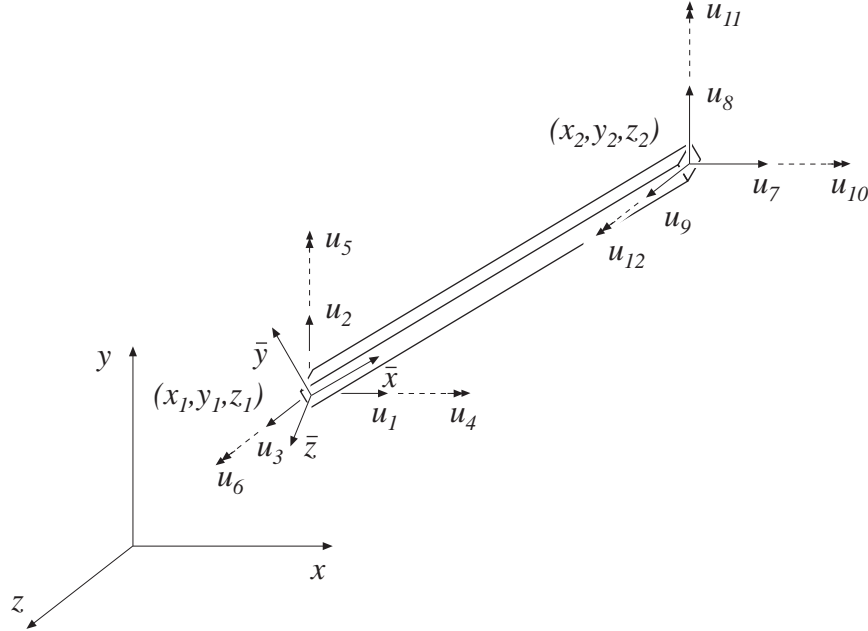
$$Q_{\bar{x}} = D_{EA} \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} \bar{\mathbf{a}}_{\text{bar}}^e$$

The normal force  $N(\bar{x})$  is then computed as

$$N(\bar{x}) = Q_{\bar{x}} + \theta(\bar{x})V(\bar{x})$$

**Purpose:**

Compute element stiffness matrix for a three dimensional beam element.

**Syntax:**

```
Ke=beam3e(ex,ey,ez,eo,ep)
[Ke,fe]=beam3e(ex,ey,ez,eo,ep,eq)
```

**Description:**

beam3e provides the global element stiffness matrix **Ke** for a three dimensional beam element.

The input variables

$$\begin{aligned} \mathbf{ex} &= \begin{bmatrix} x_1 & x_2 \end{bmatrix} \\ \mathbf{ey} &= \begin{bmatrix} y_1 & y_2 \end{bmatrix} & \mathbf{eo} &= \begin{bmatrix} x_{\bar{z}} & y_{\bar{z}} & z_{\bar{z}} \end{bmatrix} \\ \mathbf{ez} &= \begin{bmatrix} z_1 & z_2 \end{bmatrix} \end{aligned}$$

supply the element nodal coordinates  $x_1, y_1$ , etc. as well as the direction of the local beam coordinate system  $(\bar{x}, \bar{y}, \bar{z})$ . By giving a global vector  $(x_{\bar{z}}, y_{\bar{z}}, z_{\bar{z}})$  parallel with the positive local  $\bar{z}$  axis of the beam, the local beam coordinate system is defined. The variable

$$\mathbf{ep} = \begin{bmatrix} E & G & A & I_{\bar{y}} & I_{\bar{z}} & K_v \end{bmatrix}$$

supplies the modulus of elasticity  $E$ , the shear modulus  $G$ , the cross section area  $A$ , the moment of inertia with respect to the  $\bar{y}$  axis  $I_{\bar{y}}$ , the moment of inertia with respect to the  $\bar{z}$  axis  $I_{\bar{z}}$ , and St. Venant torsion constant  $K_v$ .

The element load vector **fe** can also be computed if uniformly distributed loads are applied to the element. The optional input variable

$$\mathbf{eq} = \begin{bmatrix} q_{\bar{x}} & q_{\bar{y}} & q_{\bar{z}} & q_{\bar{\omega}} \end{bmatrix}$$



then contains the distributed loads. The positive directions of  $q_{\bar{x}}$ ,  $q_{\bar{y}}$ , and  $q_{\bar{z}}$  follow the local beam coordinate system. The distributed torque  $q_{\bar{\omega}}$  is positive if directed in the local  $\bar{x}$ -direction, i.e. from local  $\bar{y}$  to local  $\bar{z}$ . All the loads are per unit length.

### Theory:

The element stiffness matrix  $\mathbf{K}^e$  is computed according to

$$\mathbf{K}^e = \mathbf{G}^T \bar{\mathbf{K}}^e \mathbf{G}$$

where

$$\bar{\mathbf{K}}^e = \begin{bmatrix} \frac{D_{EA}}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{D_{EA}}{L} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{12D_{EI\bar{z}}}{L^3} & 0 & 0 & 0 & \frac{6D_{EI\bar{z}}}{L^2} & 0 & -\frac{12D_{EI\bar{z}}}{L^3} & 0 & 0 & 0 & \frac{6D_{EI\bar{z}}}{L^2} \\ 0 & 0 & \frac{12D_{EI\bar{y}}}{L^3} & 0 & -\frac{6D_{EI\bar{y}}}{L^2} & 0 & 0 & 0 & -\frac{12D_{EI\bar{y}}}{L^3} & 0 & -\frac{6D_{EI\bar{y}}}{L^2} & 0 \\ 0 & 0 & 0 & \frac{D_{GK}}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{D_{GK}}{L} & 0 & 0 \\ 0 & 0 & -\frac{6D_{EI\bar{y}}}{L^2} & 0 & \frac{4D_{EI\bar{y}}}{L} & 0 & 0 & 0 & \frac{6D_{EI\bar{y}}}{L^2} & 0 & \frac{2D_{EI\bar{y}}}{L} & 0 \\ 0 & \frac{6D_{EI\bar{z}}}{L^2} & 0 & 0 & 0 & \frac{4D_{EI\bar{z}}}{L} & 0 & -\frac{6D_{EI\bar{z}}}{L^2} & 0 & 0 & 0 & \frac{2D_{EI\bar{z}}}{L} \\ -\frac{D_{EA}}{L} & 0 & 0 & 0 & 0 & 0 & \frac{D_{EA}}{L} & 0 & 0 & 0 & 0 & 0 \\ 0 & -\frac{12D_{EI\bar{z}}}{L^3} & 0 & 0 & 0 & -\frac{6D_{EI\bar{z}}}{L^2} & 0 & \frac{12D_{EI\bar{z}}}{L^3} & 0 & 0 & 0 & -\frac{6D_{EI\bar{z}}}{L^2} \\ 0 & 0 & -\frac{12D_{EI\bar{y}}}{L^3} & 0 & \frac{6D_{EI\bar{y}}}{L^2} & 0 & 0 & 0 & \frac{12D_{EI\bar{y}}}{L^3} & 0 & \frac{6D_{EI\bar{y}}}{L^2} & 0 \\ 0 & 0 & 0 & -\frac{D_{GK}}{L} & 0 & 0 & 0 & 0 & 0 & \frac{D_{GK}}{L} & 0 & 0 \\ 0 & 0 & -\frac{6D_{EI\bar{y}}}{L^2} & 0 & \frac{2D_{EI\bar{y}}}{L} & 0 & 0 & 0 & \frac{6D_{EI\bar{y}}}{L^2} & 0 & \frac{4D_{EI\bar{y}}}{L} & 0 \\ 0 & \frac{6D_{EI\bar{z}}}{L^2} & 0 & 0 & 0 & \frac{2D_{EI\bar{z}}}{L} & 0 & -\frac{6D_{EI\bar{z}}}{L^2} & 0 & 0 & 0 & \frac{4D_{EI\bar{z}}}{L} \end{bmatrix}$$

where

$$\mathbf{G} = \begin{bmatrix} n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ n_{x\bar{y}} & n_{y\bar{y}} & n_{z\bar{y}} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ n_{x\bar{z}} & n_{y\bar{z}} & n_{z\bar{z}} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & n_{z\bar{y}} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & n_{x\bar{z}} & n_{y\bar{z}} & n_{z\bar{z}} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & n_{z\bar{y}} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{z}} & n_{y\bar{z}} & n_{z\bar{z}} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{x}} & n_{y\bar{x}} & n_{z\bar{x}} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{y}} & n_{y\bar{y}} & n_{z\bar{y}} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & n_{x\bar{z}} & n_{y\bar{z}} & n_{z\bar{z}} \end{bmatrix}$$

where the axial stiffness  $D_{EA}$ , the bending stiffness  $D_{EI\bar{z}}$ , the bending stiffness  $D_{EI\bar{y}}$ , and the St. Venant torsion stiffness  $D_{GK}$  are given by

$$D_{EA} = EA; \quad D_{EI\bar{z}} = EI_{\bar{z}}; \quad D_{EI\bar{y}} = EI_{\bar{y}}; \quad D_{GK} = GK_v$$

The length  $L$  is given by

$$L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2 + (z_2 - z_1)^2}$$

The transformation matrix  $\mathbf{G}$  contains direction cosines computed as

$$\begin{aligned} n_{x\bar{x}} &= \frac{x_2 - x_1}{L} & n_{y\bar{x}} &= \frac{y_2 - y_1}{L} & n_{z\bar{x}} &= \frac{z_2 - z_1}{L} \\ n_{x\bar{z}} &= \frac{x_{\bar{z}}}{L_{\bar{z}}} & n_{y\bar{z}} &= \frac{y_{\bar{z}}}{L_{\bar{z}}} & n_{z\bar{z}} &= \frac{z_{\bar{z}}}{L_{\bar{z}}} \\ n_{x\bar{y}} &= n_{y\bar{z}}n_{z\bar{x}} - n_{z\bar{z}}n_{y\bar{x}} & n_{y\bar{y}} &= n_{z\bar{z}}n_{x\bar{x}} - n_{x\bar{z}}n_{z\bar{x}} & n_{z\bar{y}} &= n_{x\bar{z}}n_{y\bar{x}} - n_{y\bar{z}}n_{x\bar{x}} \end{aligned}$$

where

$$L_{\bar{z}} = \sqrt{x_{\bar{z}}^2 + y_{\bar{z}}^2 + z_{\bar{z}}^2}$$

The element load vector  $\mathbf{f}_l^e$ , stored in  $\mathbf{fe}$ , is computed according to

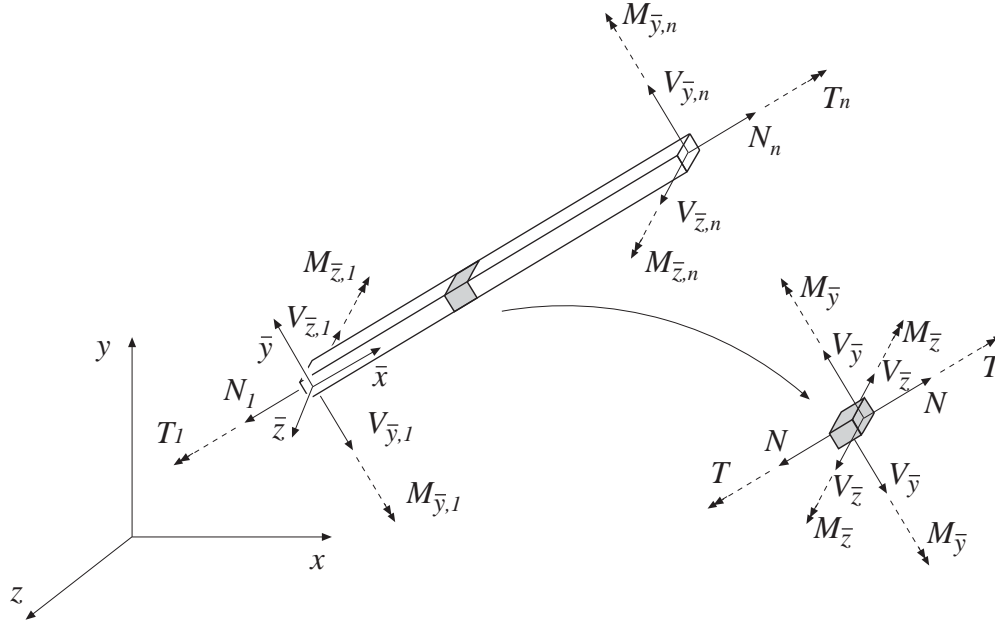
$$\mathbf{f}_l^e = \mathbf{G}^T \bar{\mathbf{f}}_l^e$$

where

$$\bar{\mathbf{f}}_l^e = \begin{bmatrix} \frac{q_{\bar{x}}L}{2} \\ \frac{q_{\bar{y}}L}{2} \\ \frac{q_{\bar{z}}L}{2} \\ \frac{q_{\bar{\omega}}L}{2} \\ -\frac{q_{\bar{z}}L^2}{12} \\ \frac{q_{\bar{y}}L^2}{12} \\ \frac{q_{\bar{x}}L}{2} \\ \frac{q_{\bar{y}}L}{2} \\ \frac{q_{\bar{z}}L}{2} \\ \frac{q_{\bar{\omega}}L}{2} \\ \frac{q_{\bar{z}}L^2}{12} \\ -\frac{q_{\bar{y}}L^2}{12} \end{bmatrix}$$

**Purpose:**

Compute section forces in a three dimensional beam element .

**Syntax:**

```
[es]=beam3s(ex,ey,e,ez,eo,ep,ed)
[es]=beam3s(ex,ey,e,ez,eo,ep,ed,eq)
[es,edi]=beam3s(ex,ey,e,ez,eo,ep,ed,eq,n)
[es,edi,eci]=beam3s(ex,ey,e,ez,eo,ep,ed,eq,n)
```

**Description:**

**beam3s** computes the section forces and displacements in local directions along the beam element **beam3e**.

The input variables **ex**, **ey**, **ez**, **eo**, **ep**, and **eq** are defined in **beam3e**.

The element displacements, stored in **ed**, are obtained by the function **extract**. If a distributed load is applied to the element, the variable **eq** must be included. The number of evaluation points for section forces and displacements are determined by **n**. If **n** is omitted, only the ends of the beam are evaluated.

The output variables

$$es = \begin{bmatrix} N(0) & V_{\bar{y}}(0) & V_{\bar{z}}(0) & T(0) & M_{\bar{y}}(0) & M_{\bar{z}}(0) \\ N(\bar{x}_2) & V_{\bar{y}}(\bar{x}_2) & V_{\bar{z}}(\bar{x}_2) & T(\bar{x}_2) & M_{\bar{y}}(\bar{x}_2) & M_{\bar{z}}(\bar{x}_2) \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ N(\bar{x}_{n-1}) & V_{\bar{y}}(\bar{x}_{n-1}) & V_{\bar{z}}(\bar{x}_{n-1}) & T(\bar{x}_{n-1}) & M_{\bar{y}}(\bar{x}_{n-1}) & M_{\bar{z}}(\bar{x}_{n-1}) \\ N(L) & V_{\bar{y}}(L) & V_{\bar{z}}(L) & T(\bar{x}_{n-1}) & M_{\bar{y}}(L) & M_{\bar{z}}(L) \end{bmatrix}$$

$$\mathbf{edi} = \begin{bmatrix} u(0) & v(0) & w(0) & \varphi(0) \\ u(\bar{x}_2) & v(\bar{x}_2) & w(\bar{x}_2) & \varphi(\bar{x}_2) \\ \vdots & \vdots & \vdots & \vdots \\ u(\bar{x}_{n-1}) & v(\bar{x}_{n-1}) & w(\bar{x}_{n-1}) & \varphi(\bar{x}_{n-1}) \\ u(L) & v(L) & w(L) & \varphi(L) \end{bmatrix} \quad \mathbf{eci} = \begin{bmatrix} 0 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_{n-1} \\ L \end{bmatrix}$$

contain the section forces, the displacements, and the evaluation points on the local  $\bar{x}$ -axis.  $L$  is the length of the beam element.

### Theory:

The nodal displacements in local coordinates are given by

$$\bar{\mathbf{a}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_2 \\ \bar{u}_3 \\ \bar{u}_4 \\ \bar{u}_5 \\ \bar{u}_6 \\ \bar{u}_7 \\ \bar{u}_8 \\ \bar{u}_9 \\ \bar{u}_{10} \\ \bar{u}_{11} \\ \bar{u}_{12} \end{bmatrix} = \mathbf{G}\mathbf{a}^e$$

where  $\mathbf{G}$  is described in **beam3e** and the transpose of  $\mathbf{a}^e$  is stored in **ed**. The displacements associated with bar action, beam action in the  $\bar{x}\bar{y}$ -plane, beam action in the  $\bar{x}\bar{z}$ -plane, and torsion are determined as

$$\bar{\mathbf{a}}_{\text{bar}}^e = \begin{bmatrix} \bar{u}_1 \\ \bar{u}_7 \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{beam},\bar{z}}^e = \begin{bmatrix} \bar{u}_2 \\ \bar{u}_6 \\ \bar{u}_8 \\ \bar{u}_{12} \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{beam},\bar{y}}^e = \begin{bmatrix} \bar{u}_3 \\ -\bar{u}_5 \\ \bar{u}_9 \\ -\bar{u}_{11} \end{bmatrix}; \quad \bar{\mathbf{a}}_{\text{torsion}}^e = \begin{bmatrix} \bar{u}_4 \\ \bar{u}_{10} \end{bmatrix}$$

The displacement  $u(\bar{x})$  and the normal force  $N(\bar{x})$  are computed from

$$u(\bar{x}) = \mathbf{N}_{\text{bar}}\bar{\mathbf{a}}_{\text{bar}}^e + u_p(\bar{x})$$

$$N(\bar{x}) = D_{EA}\mathbf{B}_{\text{bar}}\bar{\mathbf{a}}^e + N_p(\bar{x})$$

where

$$\mathbf{N}_{\text{bar}} = \begin{bmatrix} 1 & \bar{x} \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 - \frac{\bar{x}}{L} & \frac{\bar{x}}{L} \end{bmatrix}$$

$$\mathbf{B}_{\text{bar}} = \begin{bmatrix} 0 & 1 \end{bmatrix} \mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

$$u_p(\bar{x}) = -\frac{q\bar{x}}{D_{EA}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

$$N_p(\bar{x}) = -q_{\bar{x}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EA}$ ,  $L$ , and  $q_{\bar{x}}$  are defined in **beam3e** and

$$\mathbf{C}_{\text{bar}}^{-1} = \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

The displacement  $v(\bar{x})$ , the bending moment  $M_{\bar{z}}(\bar{x})$  and the shear force  $V_{\bar{y}}(\bar{x})$  are computed from

$$v(\bar{x}) = \mathbf{N}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam},\bar{z}}^e + v_p(\bar{x})$$

$$M_{\bar{z}}(\bar{x}) = D_{EI_{\bar{z}}} \mathbf{B}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam},\bar{z}}^e + M_{\bar{z},p}(\bar{x})$$

$$V_{\bar{y}}(\bar{x}) = -D_{EI_{\bar{z}}} \frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} \bar{\mathbf{a}}_{\text{beam},\bar{z}}^e + V_{\bar{y},p}(\bar{x})$$

where

$$\mathbf{N}_{\text{beam}} = \begin{bmatrix} 1 & \bar{x} & \bar{x}^2 & \bar{x}^3 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\mathbf{B}_{\text{beam}} = \begin{bmatrix} 0 & 0 & 2 & 6\bar{x} \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$\frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} = \begin{bmatrix} 0 & 0 & 0 & 6 \end{bmatrix} \mathbf{C}_{\text{beam}}^{-1}$$

$$v_p(\bar{x}) = \frac{q_{\bar{y}}}{D_{EI_{\bar{z}}}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_{\bar{z},p}(\bar{x}) = q_{\bar{y}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

$$V_{\bar{y},p}(\bar{x}) = -q_{\bar{y}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI_{\bar{z}}}$ ,  $L$ , and  $q_{\bar{y}}$  are defined in **beam3e** and

$$\mathbf{C}_{\text{beam}}^{-1} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ -\frac{3}{L^2} & -\frac{2}{L} & \frac{3}{L^2} & -\frac{1}{L} \\ \frac{2}{L^3} & \frac{1}{L^2} & -\frac{2}{L^3} & \frac{1}{L^2} \end{bmatrix}$$

The displacement  $w(\bar{x})$ , the bending moment  $M_{\bar{y}}(\bar{x})$  and the shear force  $V_{\bar{z}}(\bar{x})$  are computed from

$$w(\bar{x}) = \mathbf{N}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam},\bar{y}}^e + w_p(\bar{x})$$

$$M_{\bar{y}}(\bar{x}) = -D_{EI_{\bar{y}}} \mathbf{B}_{\text{beam}} \bar{\mathbf{a}}_{\text{beam},\bar{y}}^e + M_{\bar{y},p}(\bar{x})$$

$$V_{\bar{z}}(\bar{x}) = -D_{EI_{\bar{y}}} \frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}} \bar{\mathbf{a}}_{\text{beam},\bar{y}}^e + V_{\bar{z},p}(\bar{x})$$

where

$$w_p(\bar{x}) = \frac{q_{\bar{z}}}{D_{EI_{\bar{y}}}} \left( \frac{\bar{x}^4}{24} - \frac{L\bar{x}^3}{12} + \frac{L^2\bar{x}^2}{24} \right)$$

$$M_{\bar{y},p}(\bar{x}) = -q_{\bar{z}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} + \frac{L^2}{12} \right)$$

$$V_{\bar{z},p}(\bar{x}) = -q_{\bar{z}} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{EI_{\bar{y}}}$ ,  $L$ , and  $q_{\bar{z}}$  are defined in beam3e and  $\mathbf{N}_{\text{beam}}$ ,  $\mathbf{B}_{\text{beam}}$ , and  $\frac{d\mathbf{B}_{\text{beam}}}{d\bar{x}}$  are given above.

The displacement  $\varphi(\bar{x})$  and the torque  $T(\bar{x})$  are computed from

$$\varphi(\bar{x}) = \mathbf{N}_{\text{torsion}} \bar{\mathbf{a}}_{\text{torsion}}^e + \varphi_p(\bar{x})$$

$$T(\bar{x}) = D_{GK} \mathbf{B}_{\text{torsion}} \bar{\mathbf{a}}^e + T_p(\bar{x})$$

where

$$\mathbf{N}_{\text{torsion}} = \mathbf{N}_{\text{bar}}$$

$$\mathbf{B}_{\text{torsion}} = \mathbf{B}_{\text{bar}}$$

$$\varphi_p(\bar{x}) = -\frac{q_{\omega}}{D_{GK}} \left( \frac{\bar{x}^2}{2} - \frac{L\bar{x}}{2} \right)$$

$$T_p(\bar{x}) = -q_{\omega} \left( \bar{x} - \frac{L}{2} \right)$$

in which  $D_{GK}$ ,  $L$ , and  $q_{\omega}$  are defined in beam3e.

---

## 5 System functions

### 5.1 Introduction

The group of system functions comprises functions for the setting up, solving, and elimination of systems of equations. The functions are

|                                |
|--------------------------------|
| <b>Static system functions</b> |
|--------------------------------|

Static system functions concern the linear system of equations

$$\mathbf{K}\mathbf{a} = \mathbf{f}$$

where  $\mathbf{K}$  is the global stiffness matrix and  $\mathbf{f}$  is the global load vector. Often used static system functions are `assem` and `solveq`. The function `assem` assembles the global stiffness matrix and `solveq` computes the global displacement vector  $\mathbf{a}$  considering the boundary conditions. It should be noted that  $\mathbf{K}$ ,  $\mathbf{f}$ , and  $\mathbf{a}$  also represent analogous quantities in systems others than structural mechanical systems. For example, in a heat flow problem  $\mathbf{K}$  represents the conductivity matrix,  $\mathbf{f}$  the heat flow, and  $\mathbf{a}$  the temperature.





---

## 5.2 Static system functions

The group of static system functions comprises functions for setting up and solving the global system of equations. It also contains a function for eigenvalue analysis, a function for static condensation, a function for extraction of element displacements from the global displacement vector and a function for extraction of element coordinates.

The following functions are available for static analysis:

| Static system functions |  |
|-------------------------|--|
| <code>assem</code>      | Assemble element matrices                                    |
| <code>coordxtr</code>   | Extract element coordinates from a global coordinate matrix. |
| <code>eigen</code>      | Solve a generalized eigenvalue problem                       |
| <code>extract_ed</code> | Extract values from a global vector                          |
| <code>insert</code>     | Assemble element internal force vector                       |
| <code>red</code>        | Reduce the size of a square matrix                           |
| <code>solveq</code>     | Solve a system of equations                                  |
| <code>statcon</code>    | Perform static condensation                                  |

**Purpose:**

Assemble element matrices.

$$\begin{array}{c}
 \begin{array}{cc} i & j \\ \left[ \begin{array}{cc} k_{ii}^e & k_{ij}^e \\ k_{ji}^e & k_{jj}^e \end{array} \right] & \begin{array}{c} i \\ j \end{array} \\ \mathbf{K}^e \\ i = dof_i \\ j = dof_j \end{array} \quad \longrightarrow \quad \begin{array}{c} \begin{array}{cc} & i & j \\ \left[ \begin{array}{cccc} k_{11} & k_{12} & \vdots & \vdots \\ k_{21} & \vdots & \vdots & \vdots \\ \dots & k_{ii} + k_{ii}^e & k_{ij} + k_{ij}^e & \dots \\ \dots & k_{ji} + k_{ji}^e & k_{jj} + k_{jj}^e & \dots \end{array} \right] & \begin{array}{c} i \\ j \end{array} \\ & \vdots & \vdots & k_{nn} \\ & \vdots & \vdots & \end{array} \\ \mathbf{K} \end{array}
 \end{array}$$

**Syntax:**

$\mathbf{K} = \text{assem}(\text{edof}, \mathbf{K}, \mathbf{K}^e)$

$[\mathbf{K}, \mathbf{f}] = \text{assem}(\text{edof}, \mathbf{K}, \mathbf{K}^e, \mathbf{f}, \mathbf{f}^e)$

**Description:**

**assem** adds the element stiffness matrix  $\mathbf{K}^e$ , stored in **Ke**, to the structure stiffness matrix  $\mathbf{K}$ , stored in **K**, according to the topology matrix **edof**.

The element topology matrix **edof** is defined as

$$\text{edof} = [\text{el} \quad \underbrace{dof_1 \quad dof_2 \quad \dots \quad dof_{ned}}_{\text{global dof}}]$$

where the first column contains the element number, and the columns 2 to  $(ned + 1)$  contain the corresponding global degrees of freedom ( $ned$  = number of element degrees of freedom).

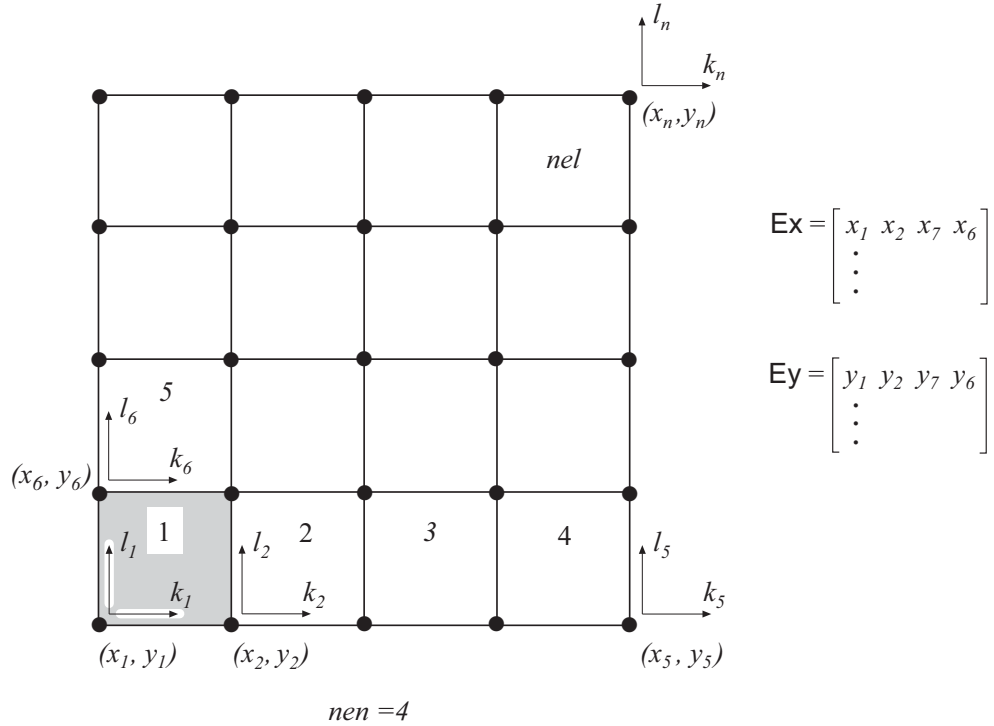
In the case where the matrix  $\mathbf{K}^e$  is identical for several elements, assembling of these can be carried out simultaneously. Each row in **Edof** then represents one element, i.e.  $nel$  is the total number of considered elements.

$$\text{Edof} = \left[ \begin{array}{cccccc} \text{el}_1 & dof_1 & dof_2 & . & . & dof_{ned} \\ \text{el}_2 & dof_1 & dof_2 & . & . & dof_{ned} \\ \vdots & \vdots & \vdots & & & \vdots \\ \text{el}_{nel} & dof_1 & dof_2 & . & . & dof_{ned} \end{array} \right] \left. \vphantom{\begin{array}{cccccc} \text{el}_1 & dof_1 & dof_2 & . & . & dof_{ned} \\ \text{el}_2 & dof_1 & dof_2 & . & . & dof_{ned} \\ \vdots & \vdots & \vdots & & & \vdots \\ \text{el}_{nel} & dof_1 & dof_2 & . & . & dof_{ned} \end{array}} \right\} \text{one row for each element}$$

If **fe** and **f** are given in the function, the element load vector  $\mathbf{f}^e$  is also added to the global load vector **f**.

**Purpose:**

Extract element coordinates from a global coordinate matrix.

**Syntax:**

$[\mathbf{Ex}, \mathbf{Ey}, \mathbf{Ez}] = \text{coordxtr}(\mathbf{Edof}, \mathbf{Coord}, \mathbf{Dof}, nen)$

**Description:**

`coordxtr` extracts element nodal coordinates from the global coordinate matrix **Coord** for elements with equal numbers of element nodes and dof's.

Input variables are the element topology matrix **Edof**, defined in **assem**, the global coordinate matrix **Coord**, the global topology matrix **Dof**, and the number of element nodes **nen** in each element.

$$\mathbf{Coord} = \begin{bmatrix} x_1 & y_1 & [z_1] \\ x_2 & y_2 & [z_2] \\ x_3 & y_3 & [z_3] \\ \vdots & \vdots & \vdots \\ x_n & y_n & [z_n] \end{bmatrix} \quad \mathbf{Dof} = \begin{bmatrix} k_1 & l_1 & \dots & m_1 \\ k_2 & l_2 & \dots & m_2 \\ k_3 & l_3 & \dots & m_3 \\ \vdots & \vdots & \dots & \vdots \\ k_n & l_n & \dots & m_n \end{bmatrix} \quad nen = [nen]$$

The nodal coordinates defined in row  $i$  of **Coord** correspond to the degrees of freedom of row  $i$  in **Dof**. The components  $k_i, l_i$  and  $m_i$  define the degrees of freedom of node  $i$ , and  $n$  is the number of global nodes for the considered part of the FE-model.

The output variables **Ex**, **Ey**, and **Ez** are matrices defined according to

$$\mathbf{Ex} = \begin{bmatrix} x_1^1 & x_2^1 & x_3^1 & \dots & x_{nen}^1 \\ x_1^2 & x_2^2 & x_3^2 & \dots & x_{nen}^2 \\ \vdots & \vdots & \vdots & \vdots & \vdots \\ x_1^{nel} & x_2^{nel} & x_3^{nel} & \dots & x_{nen}^{nel} \end{bmatrix}$$

where row  $i$  gives the  $x$ -coordinates of the element defined in row  $i$  of **Edof**, and where  $nel$  is the number of considered elements.

The element coordinate data extracted by the function **coordxtr** can be used for plotting purposes and to create input data for the element stiffness functions.

**Purpose:**

Solve the generalized eigenvalue problem.

**Syntax:**

```
L=eigen(K,M)
L=eigen(K,M,b)
[L,X]=eigen(K,M)
[L,X]=eigen(K,M,b)
```

**Description:**

`eigen` solves the eigenvalue problem

$$| K - \lambda M | = 0$$

where  $K$  and  $M$  are square matrices. The eigenvalues  $\lambda$  are stored in the vector  $L$  and the corresponding eigenvectors in the matrix  $X$ .

If certain rows and columns in matrices  $K$  and  $M$  are to be eliminated in computing the eigenvalues,  $b$  must be given in the function. The rows (and columns) that are to be eliminated are described in the vector  $b$  defined as

$$b = \begin{bmatrix} dof_1 \\ dof_2 \\ \vdots \\ dof_{nb} \end{bmatrix}$$

The computed eigenvalues are given in order ranging from the smallest to the largest. The eigenvectors are normalized in order that

$$X^T M X = I$$

where  $I$  is the identity matrix.

**Purpose:**

Extract element nodal quantities from a global solution vector.

$$\begin{bmatrix} \vdots \\ a_i \\ a_j \\ \vdots \\ a_m \\ a_n \\ \vdots \end{bmatrix} \xrightarrow{\quad} \begin{bmatrix} a_i \\ a_j \\ a_m \\ a_n \end{bmatrix} = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} \quad \begin{array}{l} \mathbf{edof} = [eln \ i \ j \ m \ n] \\ \mathbf{ed} = [u_1 \ u_2 \ u_3 \ u_4] \end{array}$$

**Syntax:**

`ed=extract_ed(edof,a)`

**Description:**

`extract_ed` extracts element displacements or corresponding quantities  $\mathbf{a}^e$  from the global solution vector  $\mathbf{a}$ , stored in `a`.

Input variables are the element topology matrix `edof`, defined in `assem`, and the global solution vector `a`.

The output variable

$$\mathbf{ed} = (\mathbf{a}^e)^T$$

contains the element displacement vector.

If `Edof` contains more than one element, `Ed` will be a matrix

$$\mathbf{Ed} = \begin{bmatrix} (\mathbf{a}^e)_1^T \\ (\mathbf{a}^e)_2^T \\ \vdots \\ (\mathbf{a}^e)_{nel}^T \end{bmatrix}$$

where row  $i$  gives the element displacements for the element defined in row  $i$  of `Edof`, and  $nel$  is the total number of considered elements.

**Example:**

For the two dimensional beam element, the `extract` function will extract six nodal displacements for each element given in `Edof`, and create a matrix `Ed` of size  $(nel \times 6)$ .

$$\mathbf{Ed} = \begin{bmatrix} u_1 & u_2 & u_3 & u_4 & u_5 & u_6 \\ u_1 & u_2 & u_3 & u_4 & u_5 & u_6 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ u_1 & u_2 & u_3 & u_4 & u_5 & u_6 \end{bmatrix}$$

**Purpose:**

Assemble internal element forces in a global force vector.

$$\begin{array}{ccc}
 \begin{bmatrix} f_i^e \\ f_j^e \end{bmatrix} & \xrightarrow{\quad} & \begin{bmatrix} f_1 \\ f_2 \\ \vdots \\ f_i + f_i^e \\ f_j + f_j^e \\ \vdots \\ f_n \end{bmatrix} \\
 \mathbf{f}^e & & \mathbf{f} \\
 i = dof_i & & \\
 j = dof_j & & 
 \end{array}$$

**Syntax:**

`f=insert(edof,f,ef)`

**Description:**

`insert` adds the internal element load vector  $\mathbf{f}_i^e$ , stored in `ef`, to the global internal force vector  $\mathbf{f}$ , stored in `f`, according to the topology matrix `edof`. The function is for use in nonlinear analysis.

The element topology matrix `edof` is defined in `assem`. The vector `f` is the global internal force vector, and the vector `ef` is the internal element force vector computed from the element stresses, see for example `planif`.



**Purpose:**

Reduce the size of a square matrix by omitting rows and columns.

**Syntax:**

$B = \text{red}(A, b)$

**Description:**

$B = \text{red}(A, b)$  reduces the square matrix  $A$  to a smaller matrix  $B$  by omitting rows and columns of  $A$ . The indices for rows and columns to be omitted are specified by the column vector  $b$ .

**Examples:**

Assume that the matrix  $A$  is defined as

$$A = \begin{bmatrix} 1 & 2 & 3 & 4 \\ 5 & 6 & 7 & 8 \\ 9 & 10 & 11 & 12 \\ 13 & 14 & 15 & 16 \end{bmatrix}$$

and  $b$  as

$$b = \begin{bmatrix} 2 \\ 4 \end{bmatrix}$$

The statement  $B = \text{red}(A, b)$  results in the matrix

$$B = \begin{bmatrix} 1 & 3 \\ 9 & 11 \end{bmatrix}$$

**Purpose:**

Solve equation system.

**Syntax:**

```
a=solveq(K,f)
a=solveq(K,f,bc)
[a,r]=solveq(K,f,bc)
```

**Description:**

solveq solves the equation system

$$\mathbf{K} \mathbf{a} = \mathbf{f}$$

where  $\mathbf{K}$  is a matrix and  $\mathbf{a}$  and  $\mathbf{f}$  are vectors.

The matrix  $\mathbf{K}$  and the vector  $\mathbf{f}$  must be predefined. The solution of the system of equations is stored in a vector  $\mathbf{a}$  which is created by the function.

If some values of  $\mathbf{a}$  are to be prescribed, the row number and the corresponding values are given in the boundary condition matrix

$$\mathbf{bc} = \begin{bmatrix} dof_1 & u_1 \\ dof_2 & u_2 \\ \vdots & \vdots \\ dof_{nbc} & u_{nbc} \end{bmatrix}$$

where the first column contains the row numbers and the second column the corresponding prescribed values.

If  $\mathbf{r}$  is given in the function, support forces are computed according to

$$\mathbf{r} = \mathbf{K} \mathbf{a} - \mathbf{f}$$

**Purpose:**

Reduce system of equations by static condensation.

**Syntax:**

`[K1,f1]=statcon(K,f,b)`

**Description:**

`statcon` reduces a system of equations

$$\mathbf{K} \mathbf{a} = \mathbf{f}$$

by static condensation.

The degrees of freedom to be eliminated are supplied to the function by the vector

$$\mathbf{b} = \begin{bmatrix} dof_1 \\ dof_2 \\ \vdots \\ dof_{nb} \end{bmatrix}$$

where each row in  $\mathbf{b}$  contains one degree of freedom to be eliminated.

The elimination gives the reduced system of equations

$$\mathbf{K}_1 \mathbf{a}_1 = \mathbf{f}_1$$

where  $\mathbf{K}_1$  and  $\mathbf{f}_1$  are stored in `K1` and `f1` respectively.



---

## 6 Statements and macros

Statements describe algorithmic actions that can be executed. There are two different types of control statements, conditional and repetitive. The first group defines conditional jumps whereas the latter one defines repetition until a conditional statement is fulfilled. Macros are used to define new functions to the MATLAB or CALFEM structure, or to store a sequence of statements in an `.m`-file.

| Control statements |                           |
|--------------------|---------------------------|
| <code>if</code>    | Conditional jump          |
| <code>for</code>   | Initiate a loop           |
| <code>while</code> | Define a conditional loop |

| Macros          |                                |
|-----------------|--------------------------------|
| <i>function</i> | Define a new function          |
| <i>script</i>   | Store a sequence of statements |

**Purpose:**

Conditional jump.

**Syntax:**

```
if logical expression
:
elseif logical expression
:
else
:
end
```

**Description:**

if initiates a conditional jump. If *logical expression* produces the value *True* the statements following if are executed. If *logical expression* produces the value *False* the next conditional statement **elseif** is checked.

**elseif** works like if. One or more of the conditional statement **elseif** can be added after the initial conditional statement if.

If **else** is present, the statements following **else** are executed if the *logical expressions* in all if and **elseif** statements produce the value *False*. The if loop is closed by **end** to define the loop sequence.

The following relation operators can be used

|    |                          |
|----|--------------------------|
| == | equal                    |
| >= | greater than or equal to |
| >  | greater than             |
| <= | less than or equal to    |
| <  | less than                |
| ~= | not equal                |

**Note:**

This is MATLAB built-in language.

**Purpose:**

Initiate a loop.

**Syntax:**

```
for i = start : inc : stop
:
end
```

**Description:**

for initiates a loop which terminates when  $i > \text{stop}$ . The for loop is closed by end to define the loop sequence.

**Examples:**

|                     |                              |
|---------------------|------------------------------|
| for i = 1 : 10      | i takes values from 1 to 10. |
| for i = 1 : 2 : 10  | i equals 1, 3, 5, 7, 9.      |
| for i = 20 : -1 : 1 | i equals 20, 19 ... 2, 1.    |

**Note:**

This is MATLAB built-in language.

## while

---

### Purpose:

Define a conditional loop.

### Syntax:

```
while logical expression
:
end
```

### Description:

**while** initiates a conditional loop which terminates when *logical expression* equals *False*. The **while** loop is closed by **end** to define the loop sequence.

The different relation operators that can be used can be found under the **if** command.

### Examples:

A loop continuing until a equals b

```
while a~=b
:
end
```

### Note:

This is MATLAB built-in language.



**Purpose:**

Define a new function.

**Syntax:**

`function[ out1 , out2 , ... ]=name( in1 , in2 , ... )`

**Description:**

*name* is replaced by the name of the function. The input variables `in1`, `in2`, ... can be scalars, vectors or matrices, and the same holds for the output variables `out1`, `out2`, ... .

**Example:**

To define the CALFEM function `spring1e` a file named `spring1e.m` is created. The file contains the following statements:

```
function [Ke]=spring1e(k)
% Define the stiffness matrix
% for a one dimensional spring
% with spring stiffness k
Ke=[ k, -k; -k, k ]
```

i.e. the function `spring1e` is defined to return a stiffness matrix.

**Note:**

This is MATLAB built-in language.

**Purpose:**

Execute a stored sequence of statements.

**Syntax:**

*name*

**Description:**

*name* is replaced by the name of the script.

**Example:**

The statements below are stored in a file named **spring.m** and executed by typing **spring** in the MATLAB command window.

```
% Stiffness matrix for a one dimensional  
% spring with stiffness k=10  
k=10;  
[Ke]=spring1e(k);
```

**Note:**

This is MATLAB built-in language.

---

## 7 Graphics functions

The group of graphics functions comprises functions for element based graphics. Mesh plots, displacements, section forces, flows, iso lines and principal stresses can be displayed. The functions are divided into two dimensional, and general graphics functions.

| Two dimensional graphics functions |  |
|------------------------------------|--|
| disbeam2                           | Draw displacements for beam element                |
| eldraw2                            | Draw undeformed finite element mesh                |
| eldisp2                            | Draw deformed finite element mesh                  |
| elflux2                            | Plot flux vectors                                  |
| eliso2                             | Draw isolines for nodal quantities                 |
| elprinc2                           | Plot principal stresses                            |
| scalfact2                          | Determine scale factor                             |
| scalgraph2                         | Draw graphic scale                                 |
| secforce2                          | Draw section force diagram for bar or beam element |

| General graphics functions in MATLAB |                                   |
|--------------------------------------|-----------------------------------|
| axis                                 | Axis scaling and appearance       |
| clf                                  | Clear current figure              |
| figure                               | Create figures                    |
| fill                                 | Draw filled 2D polygons           |
| grid                                 | Grid lines                        |
| hold                                 | Hold current graph                |
| plot                                 | Plot lines and points in 2D space |
| print                                | Print graph or save graph to file |
| text                                 | Add text to current plot          |
| title                                | Titles for 2D and 3D plots        |
| xlabel,                              | Axis labels for 2D and 3D plots   |
| ylabel,                              |                                   |
| zlabel                               |                                   |

### Purpose:

Draw the displacements for a two dimensional beam element.

### Syntax:

```
[sfac]=dispbeam2(ex,ey,edi)
[sfac]=dispbeam2(ex,ey,edi,plotpar)
dispbeam2(ex,ey,edi,plotpar,sfac)
```

### Description:

Input variables are the coordinate matrices **ex** and **ey**, see e.g. **beam2e**, and the element displacements **edi** obtained by e.g. **beam2s**.

The variable **plotpar** sets plot parameters for linetype, linecolour and node marker.

```
plotpar=[ linetype  linecolor  nodemark ]
```

|                     |             |                      |         |
|---------------------|-------------|----------------------|---------|
| <i>linetype</i> = 1 | solid line  | <i>linecolor</i> = 1 | black   |
| 2                   | dashed line | 2                    | blue    |
| 3                   | dotted line | 3                    | magenta |
|                     |             | 4                    | red     |

|                     |         |
|---------------------|---------|
| <i>nodemark</i> = 1 | circle  |
| 2                   | star    |
| 0                   | no mark |

Default is dashed black lines with circles at nodes.

The scale factor **sfac** is a scalar that the element displacements are multiplied with to get a suitable geometrical representation. If **sfac** is omitted in the input list the scale factor is set automatically.

**Purpose:**

Draw the undeformed mesh for a two dimensional structure.

**Syntax:**

```
eldraw2(Ex,Ey)
eldraw2(Ex,Ey,plotpar)
eldraw2(Ex,Ey,plotpar,elnum)
```

**Description:**

eldraw2 displays the undeformed mesh for a two dimensional structure.

Input variables are the coordinate matrices **Ex** and **Ey** formed by the function **co-ordxtr**.

The variable **plotpar** sets plot parameters for **linetype**, **linecolor** and node marker.

**plotpar** = [ *linetype linecolor nodemark* ]

|                     |             |                      |         |
|---------------------|-------------|----------------------|---------|
| <i>linetype</i> = 1 | solid line  | <i>linecolor</i> = 1 | black   |
| 2                   | dashed line | 2                    | blue    |
| 3                   | dotted line | 3                    | magenta |
|                     |             | 4                    | red     |

|                     |         |
|---------------------|---------|
| <i>nodemark</i> = 1 | circle  |
| 2                   | star    |
| 0                   | no mark |

Default is solid black lines with circles at nodes.

Element numbers can be displayed at the center of the element if a column vector **elnum** with the element numbers is supplied. This column vector can be derived from the element topology matrix **Edof**,

**elnum**=**Edof**(:,1)

i.e. the first column of the topology matrix.

**Limitations:**

Supported elements are bar, beam, triangular three node, and quadrilateral four node elements.

### Purpose:

Draw the deformed mesh for a two dimensional structure.

### Syntax:

```
[sfac]=eldisp2(Ex,Ey,Ed)
[sfac]=eldisp2(Ex,Ey,Ed,plotpar)
eldisp2(Ex,Ey,Ed,plotpar,sfac)
```

### Description:

`eldisp2` displays the deformed mesh for a two dimensional structure.

Input variables are the coordinate matrices `Ex` and `Ey` formed by the function `co-ordxtr`, and the element displacements `Ed` formed by the function `extract`.

The variable `plotpar` sets plot parameters for `linetype`, `linecolor` and node marker.

```
plotpar=[ linetype linecolor nodemark ]
```

|                     |             |                      |         |
|---------------------|-------------|----------------------|---------|
| <i>linetype</i> = 1 | solid line  | <i>linecolor</i> = 1 | black   |
| 2                   | dashed line | 2                    | blue    |
| 3                   | dotted line | 3                    | magenta |
|                     |             | 4                    | red     |

|                     |         |
|---------------------|---------|
| <i>nodemark</i> = 1 | circle  |
| 2                   | star    |
| 0                   | no mark |

Default is dashed black lines with circles at nodes.

The scale factor `sfac` is a scalar that the element displacements are multiplied with to get a suitable geometrical representation. The scale factor is set automatically if it is omitted in the input list.

### Limitations:

Supported elements are bar, beam, triangular three node, and quadrilateral four node elements.

**Purpose:**

Determine scale factor for drawing computational results.

**Syntax:**

```
[sfac]=scalfact2(ex,ey,ed)  
[sfac]=scalfact2(ex,ey,ed,rat)
```

**Description:**

**scalfact2** determines a scale factor **sfac** for drawing computational results, such as displacements, section forces or flux.

Input variables are the coordinate matrices **ex** and **ey** and the matrix **ed** containing the quantity to be displayed. The scalar **rat** defines the ratio between the geometric representation of the largest quantity to be displayed and the element size. If **rat** is not specified, 0.2 is used.

**Purpose:**

Draw a Graphic scale.

**Syntax:**

```
scalgraph2(sfac,magnitude)
scalgraph2(sfac,magnitude,plotpar)
```

**Description:**

`scalgraph2` draws a graphic scale to visualise the magnitude of displayed computational results. The input variable `sfac` is a scale factor determined by the function `scalfact2` and the variable

$$\text{magnitude} = [ S \ x \ y ]$$

specifies the value corresponding the length of the graphic scale  $S$ , and  $(x, y)$  are the coordinates of the starting point. If no coordinates are given the starting point will be  $(0, -0.5)$ .

The variable `plotpar` sets the the graphic scale color.

```
plotpar=[color ]
```

```
color = 1  black
        2  blue
        3  magenta
        4  red
```



**Purpose:**

Draw the section force diagrams of a two dimensional bar or beam element in its global position.

**Syntax:**

```
secforce2(ex,ey,es,plotpar,sfac)
secforce2(ex,ey,es,plotpar,sfac,eci)
[sfac]=secforce2(ex,ey,es)
[sfac]=secforce2(ex,ey,es,plotpar)
```

**Description:**

The input variables **ex** and **ey** are defined in **bar2e** or **beam2e** and the input variable

$$\mathbf{es} = \begin{bmatrix} S_1 \\ S_2 \\ \vdots \\ S_n \end{bmatrix}$$

consists of a column matrix that contains section forces. The values in **es** are computed in e.g. **bar2s** or **beam2s**.

The variable **plotpar** sets plot parameters for the diagram.

**plotpar**=[ *linecolor* *elementcolor* ]

|                      |         |                         |         |
|----------------------|---------|-------------------------|---------|
| <i>linecolor</i> = 1 | black   | <i>elementcolor</i> = 1 | black   |
| 2                    | blue    | 2                       | blue    |
| 3                    | magenta | 3                       | magenta |
| 4                    | red     | 4                       | red     |

The scale factor **sfac** is a scalar that the section forces are multiplied with to get a suitable graphical representation. If **sfac** is omitted in the input list the scale factor is set automatically.

The input variable

$$\mathbf{eci} = \begin{bmatrix} \bar{x}_1 \\ \bar{x}_2 \\ \vdots \\ \bar{x}_n \end{bmatrix}$$

specifies the local  $\bar{x}$ -coordinates of the quantities in **es**. If **eci** is not given, uniform distance is assumed.

### Purpose:

Plot axis scaling and appearance.

### Syntax:

```
axis([xmin xmax ymin ymax])  
axis([xmin xmax ymin ymax zmin zmax])  
axis auto  
axis square  
axis equal  
axis off  
axis on
```

### Description:

`axis([xmin xmax ymin ymax])` sets scaling for the x- and y-axes on the current 2D plot.

`axis([xmin xmax ymin ymax zmin zmax])` sets the scaling for the x-, y- and z-axes on the current 3D plot.

`axis auto` returns the axis scaling to its default automatic mode where, for each plot,  $xmin = \min(x)$ ,  $xmax = \max(x)$ , etc.

`axis square` makes the current axis box square in shape.

`axis equal` changes the current axis box size so that equal tick mark increments on the x- and y-axes are equal in size. This makes `plot(sin(x),cos(x))` look like a circle, instead of an oval.

`axis normal` restores the current axis box to full size and removes any restrictions on the scaling of the units. This undoes the effects of `axis square` and `axis equal`.

`axis off` turns off all axis labeling and tick marks.

`axis on` turns axis labeling and tick marks back on.

### Note:

This is a MATLAB built-in function. For more information about the `axis` function, type `help axis`.

**Purpose:**

Clear current figure (graph window).

**Syntax:**

clf

**Description:**

clf deletes all objects (axes) from the current figure.

**Note:**

This is a MATLAB built-in function. For more information about the clf function, type `help clf`.

## figure

---

### Purpose:

Create figures (graph windows).

### Syntax:

figure(h)

### Description:

figure(h) makes the h'th figure the current figure for subsequent plotting functions. If *figure h* does not exist, a new one is created using the first available figure handle.

### Note:

This is a MATLAB built-in function. For more information about the **figure** function, type **help figure**.

**Purpose:**

Filled 2D polygons.

**Syntax:**

```
fill(x,y,c)
fill(X,Y,C)
```

**Description:**

`fill(x,y,c)` fills the 2D polygon defined by vectors `x` and `y` with the color specified by `c`. The vertices of the polygon are specified by pairs of components of `x` and `y`. If necessary, the polygon is closed by connecting the last vertex to the first.

If `c` is a vector of the same length as `x` and `y`, its elements are used to specify colors at the vertices. The color within the polygon is obtained by bilinear interpolation in the vertex colors.

If `X`, `Y` and `C` are matrices of the same size, `fill(X,Y,C)` draws one polygon per column with interpolated colors.

**Example:**

The solution of a heat conduction problem results in a vector `d` with nodal temperatures. The temperature distribution in a group of triangular 3 node (`nen=3`) or quadrilateral 4 node (`nen=4`) elements, with topology defined by `edof`, can be displayed by

```
[ex,ey]=coordxtr(edof,Coord,Dof,nen)
ed=extract(edof,d)
colormap(hot)
fill(ex',ey',ed')
```

**Note:**

This is a MATLAB built-in function. For more information about the `fill` function, type `help fill`.

## **grid**

---

### **Purpose:**

Grid lines for 2D and 3D plots.

### **Syntax:**

```
grid on  
grid off  
grid
```

### **Description:**

`grid on` adds grid lines on the current axes.

`grid off` takes them off.

`grid` by itself, toggles the grid state.

### **Note:**

This is a MATLAB built-in function. For more information about the **grid** function, type `help grid`.

**Purpose:**

Hold the current graph.

**Syntax:**

hold on  
hold off  
hold

**Description:**

hold on holds the current graph.

hold off returns to the default mode where plot functions erase previous plots.

hold by itself, toggles the hold state.

**Note:**

This is a MATLAB built-in function. For more information about the **hold** function, type `help hold`.

## **plot**

---

### **Purpose:**

Linear two dimensional plot.

### **Syntax:**

```
plot(x,y)
plot(x,y,'linetype')
```

### **Description:**

`plot(x,y)` plots vector `x` versus vector `y`. Straight lines are drawn between each pair of values.

Various line types, plot symbols and colors may be obtained with `plot(x,y,s)` where `s` is a 1, 2, or 3 character string made from the following characters:

|    |              |   |        |   |         |
|----|--------------|---|--------|---|---------|
| -  | solid line   | . | point  | y | yellow  |
| :  | dotted line  | o | circle | m | magenta |
| -. | dashdot line | x | x-mark | c | cyan    |
| -- | dashed line  | + | plus   | r | red     |
|    |              | * | star   | g | green   |
|    |              |   |        | b | blue    |
|    |              |   |        | w | white   |
|    |              |   |        | k | black   |

Default is solid blue lines.

### **Example:**

The statement

```
plot(x,y,'-',x,y,'ro')
```

plots the data twice, giving a solid blue line with red circles at the data points.

### **Note:**

This is a MATLAB built-in function. For more information about the `plot` function, type `help plot`.



**Purpose:**

Create hardcopy output of current figure window.

**Syntax:**

```
print [filename]
```

**Description:**

`print` with no arguments sends the contents of the current figure window to the default printer. `print filename` creates a PostScript file of the current figure window and writes it to the specified file.

**Note:**

This is a MATLAB built-in function. For more information about the `print` function, type `help print`.

**Purpose:**

Add text to current plot.

**Syntax:**

```
text(x,y,'string')
```

**Description:**

**text** adds the text in the quotes to location (x,y) on the current axes, where (x,y) is in units from the current plot. If **x** and **y** are vectors, **text** writes the text at all locations given. If '**string**' is an array with the same number of rows as the length of **x** and **y**, **text** marks each point with the corresponding row of the '**string**' array.

**Note:**

This is a MATLAB built-in function. For more information about the **text** function, type **help text**.

**Purpose:**

Titles for 2D and 3D plots.

**Syntax:**

```
title('text')
```

**Description:**

`title` adds the text string `'text'` at the top of the current plot.

**Note:**

This is a MATLAB built-in function. For more information about the **title** function, type `help title`.

**Purpose:**

$x$ -,  $y$ -, and  $z$ -axis labels for 2D and 3D plots.

**Syntax:**

```
xlabel('text')  
ylabel('text')  
zlabel('text')
```

**Description:**

`xlabel` adds text beside the  $x$ -axis on the current plot.

`ylabel` adds text beside the  $y$ -axis on the current plot.

`zlabel` adds text beside the  $z$ -axis on the current plot.

**Note:**

This is a MATLAB built-in function. For more information about the functions, type `help xlabel`, `help ylabel`, or `help zlabel`.

---

## 8 User's Manual, examples

### 8.1 Introduction

This set of examples is defined with the ambition to serve as a User's Manual. The examples, except the introductory ones, are written as `.m`-files (script files) and supplied together with the CALFEM functions.

The User's Manual examples are separated into two groups:

|                           |
|---------------------------|
| <b>Static analysis</b>    |
| <b>Nonlinear analysis</b> |

The static linear examples illustrate finite element analysis of different structures loaded by stationary loads. The examples of nonlinear analysis cover subjects such as geometrical and material nonlinearities.



---

## 8.2 Static analysis

This section illustrates some linear static finite element calculations. The examples deal with structural problems as well as field problems such as heat conduction.

| Static analysis |                                  |
|-----------------|----------------------------------|
| exs_spring      | Linear spring system             |
| exs_flw_temp1   | One-dimensional heat flow        |
| exs_bar2        | Plane truss                      |
| exs_bar2_l      | Plane truss analysed using loops |
| exs_beam1       | Simply supported beam            |
| exs_beam2       | Plane frame                      |
| exs_beambar2    | Plane frame stabilized with bars |

**Note:** The examples listed above are supplied as `.m`-files under the directory `examples`. The example files are named according to the table.

**Purpose:**

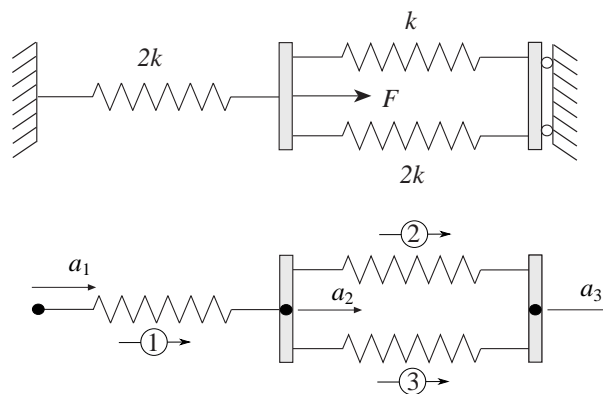
Show the basic steps in a finite element calculation.

**Description:**

The general procedure in linear finite element calculations is carried out for a simple structure. The steps are

- define the model
- generate element matrices
- assemble element matrices into the global system of equations
- solve the global system of equations
- evaluate element forces

Consider the system of three linear elastic springs, and the corresponding finite element model. The system of springs is fixed in its ends and loaded by a single load  $F$ .



The computation is initialized by defining the topology matrix **Edof**, containing element numbers and global element degrees of freedom,

```
>> Edof=[1  1  2;
          2  2  3;
          3  2  3];
```

the global stiffness matrix **K** ( $3 \times 3$ ) of zeros,

```
>> K=zeros(3,3)
```

```
K =
    0    0    0
    0    0    0
    0    0    0
```



and the load vector  $\mathbf{f}$  ( $3 \times 1$ ) with the load  $F = 100$  in position 2.

```
>> f=zeros(3,1); f(2)=100
```

```
f =
     0
    100
     0
```

Element stiffness matrices are generated by the function `spring1e`. The element property `ep` for the springs contains the spring stiffnesses  $k$  and  $2k$  respectively, where  $k = 1500$ .

```
>> k=1500; ep1=k; ep2=2*k;
>> Ke1=spring1e(ep1)
```

```
Ke1 =
      1500      -1500
     -1500       1500
```

```
>> Ke2=spring1e(ep2)
```

```
Ke2 =
      3000     -3000
     -3000       3000
```

The element stiffness matrices are assembled into the global stiffness matrix  $\mathbf{K}$  according to the topology.

```
>> K=assem(Edof(1,:),K,Ke2)
```

```
K =
      3000     -3000         0
     -3000       3000         0
         0         0         0
```

```
>> K=assem(Edof(2,:),K,Ke1)
```

```
K =
      3000     -3000         0
     -3000       4500     -1500
         0     -1500       1500
```

```
>> K=assem(Edof(3,:),K,Ke2)
```

```
K =
      3000      -3000         0
     -3000       7500     -4500
         0      -4500      4500
```

The global system of equations is solved considering the boundary conditions given in `bc`.

```
>> bc= [1 0; 3 0];
>> [a,r]=solveq(K,f,bc)
```

```
a =
      0
    0.0133
      0
```

```
r =
    -40.0000
         0
    -60.0000
```

Element forces are evaluated from the element displacements. These are obtained from the global displacements `a` using the function `extract`.

```
>> ed1=extract_ed(Edof(1,:),a)
```

```
ed1 =
      0      0.0133
```

```
>> ed2=extract_ed(Edof(2,:),a)
```

```
ed2 =
    0.0133      0
```

```
>> ed3=extract_ed(Edof(3,:),a)
```

```
ed3 =
    0.0133      0
```

The spring forces are evaluated using the function `spring1s`.

```
>> es1=spring1s(ep2,ed1)
```

```
es1 =  
      40
```

```
>> es2=spring1s(ep1,ed2)
```

```
es2 =  
     -20
```

```
>> es3=spring1s(ep2,ed3)
```

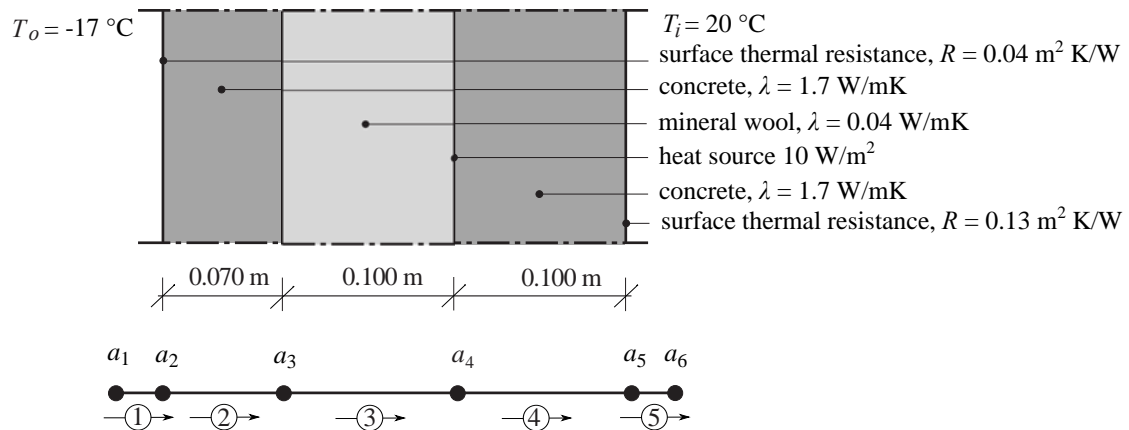
```
es3 =  
     -40
```

**Purpose:**

Analysis of one-dimensional heat flow.

**Description:**

Consider a wall built up of concrete and thermal insulation. The outdoor temperature is  $-17^\circ\text{C}$  and the temperature inside is  $20^\circ\text{C}$ . At the inside of the thermal insulation there is a heat source yielding  $10\text{ W/m}^2$ .



The wall is subdivided into five elements and the one-dimensional spring (analogy) element `spring1e` is used. Equivalent spring stiffnesses are  $k_i = \lambda A/L$  for thermal conductivity and  $k_i = A/R$  for thermal surface resistance. Corresponding spring stiffnesses per  $\text{m}^2$  of the wall are:

$$\begin{aligned} k_1 &= 1/0.04 &= 25.0 &\text{ W/K} \\ k_2 &= 1.7/0.070 &= 24.3 &\text{ W/K} \\ k_3 &= 0.040/0.100 &= 0.4 &\text{ W/K} \\ k_4 &= 1.7/0.100 &= 17.0 &\text{ W/K} \\ k_5 &= 1/0.13 &= 7.7 &\text{ W/K} \end{aligned}$$

A global system matrix  $\mathbf{K}$  and a heat flow vector  $\mathbf{f}$  are defined. The heat source inside the wall is considered by setting  $f_4 = 10$ . The element matrices  $\mathbf{K}_e$  are computed using `spring1e`, and the function `assem` assembles the global stiffness matrix.

The system of equations is solved using `solveq` with considerations to the boundary conditions in `bc`. The prescribed temperatures are  $a_1 = -17^\circ\text{C}$  and  $a_6 = 20^\circ\text{C}$ .

```
>> Edof=[1  1  2
          2  2  3;
          3  3  4;
          4  4  5;
          5  5  6];
```

```
>> K=zeros(6);
>> f=zeros(6,1); f(4)=10

f =

    0
    0
    0
   10
    0
    0

>> ep1=[25]; ep2=[24.3];
>> ep3=[0.4]; ep4=[17];
>> ep5=[7.7];

>> Ke1=spring1e(ep1); Ke2=spring1e(ep2);
>> Ke3=spring1e(ep3); Ke4=spring1e(ep4);
>> Ke5=spring1e(ep5);

>> K=assem(Edof(1,:),K,Ke1); K=assem(Edof(2,:),K,Ke2);
>> K=assem(Edof(3,:),K,Ke3); K=assem(Edof(4,:),K,Ke4);
>> K=assem(Edof(5,:),K,Ke5);

>> bc=[1 -17; 6 20];

>> [a,r]=solveq(K,f,bc)

a =

-17.0000
-16.4384
-15.8607
 19.2378
 19.4754
 20.0000

r =

-14.0394
 0.0000
-0.0000
 0
 0.0000
 4.0394
```

The temperature values  $a_i$  in the node points are given in the vector **a** and the boundary flows in the vector **r**.

After solving the system of equations, the heat flow through the wall is computed using `extract` and `spring1s`.

```
>> ed1=extract_ed(Edof(1,:),a);  
>> ed2=extract_ed(Edof(2,:),a);  
>> ed3=extract_ed(Edof(3,:),a);  
>> ed4=extract_ed(Edof(4,:),a);  
>> ed5=extract_ed(Edof(5,:),a);
```

```
>> q1=spring1s(ep1,ed1)
```

```
q1 =
```

```
14.0394
```

```
>> q2=spring1s(ep2,ed2)
```

```
q2 =
```

```
14.0394
```

```
>> q3=spring1s(ep3,ed3)
```

```
q3 =
```

```
14.0394
```

```
>> q4=spring1s(ep4,ed4)
```

```
q4 =
```

```
4.0394
```

```
>> q5=spring1s(ep5,ed5)
```

```
q5 =
```

```
4.0394
```

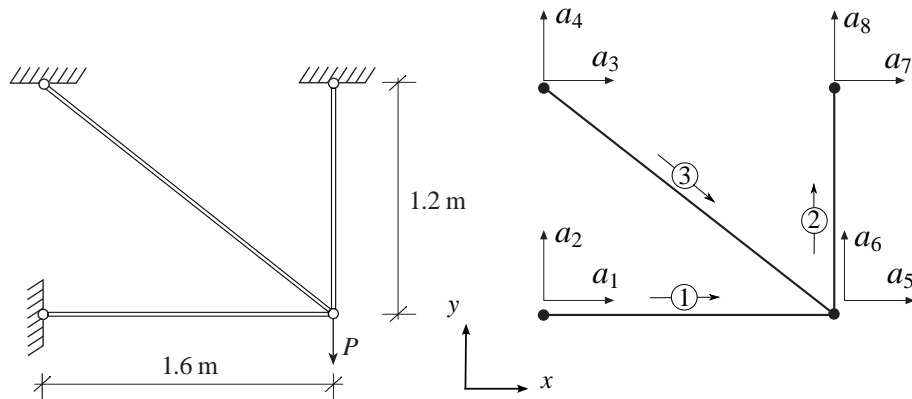
The heat flow through the wall is  $q = 14.0 \text{ W/m}^2$  in the part of the wall to the left of the heat source, and  $q = 4.0 \text{ W/m}^2$  in the part to the right of the heat source.

**Purpose:**

Analysis of a plane truss.

**Description:**

Consider a plane truss consisting of three bars with the properties  $E = 200 \text{ GPa}$ ,  $A_1 = 6.0 \cdot 10^{-4} \text{ m}^2$ ,  $A_2 = 3.0 \cdot 10^{-4} \text{ m}^2$  and  $A_3 = 10.0 \cdot 10^{-4} \text{ m}^2$ , and loaded by a single force  $P = 80 \text{ kN}$ . The corresponding finite element model consists of three elements and eight degrees of freedom.



The topology is defined by the matrix

```
>> Edof=[1  1  2  5  6;
          2  5  6  7  8;
          3  3  4  5  6];
```

The stiffness matrix  $K$  and the load vector  $f$ , are defined by

```
>> K=zeros(8);
    f=zeros(8,1); f(6)=-80e3;
```

The element property vectors  $ep1$ ,  $ep2$  and  $ep3$  are defined by

```
>> E=2.0e11;
>> A1=6.0e-4;    A2=3.0e-4;    A3=10.0e-4;
>> ep1=[E A1];   ep2=[E A2];   ep3=[E A3];
```

and the element coordinate vectors  $ex1$ ,  $ex2$ ,  $ex3$ ,  $ey1$ ,  $ey2$  and  $ey3$  by

```
>> ex1=[0 1.6];   ex2=[1.6 1.6];   ex3=[0 1.6];
>> ey1=[0 0];     ey2=[0 1.2];     ey3=[1.2 0];
```

The element stiffness matrices **Ke1**, **Ke2** and **Ke3** are computed using **bar2e**.

```
>> Ke1=bar2e(ex1,ey1,ep1)
```

Ke1 =

```
1.0e+007 *
    7.5000         0   -7.5000         0
         0         0         0         0
   -7.5000         0    7.5000         0
         0         0         0         0
```

```
>> Ke2=bar2e(ex2,ey2,ep2)
```

Ke2 =

```
1.0e+007 *
         0         0         0         0
         0    5.0000         0   -5.0000
         0         0         0         0
         0   -5.0000         0    5.0000
```

```
>> Ke3=bar2e(ex3,ey3,ep3)
```

Ke3 =

```
1.0e+007 *
    6.4000   -4.8000   -6.4000    4.8000
   -4.8000    3.6000    4.8000   -3.6000
   -6.4000    4.8000    6.4000   -4.8000
    4.8000   -3.6000   -4.8000    3.6000
```

Based on the topology information, the global stiffness matrix can be generated by assembling the element stiffness matrices

```
>> K=assem(Edof(1,:),K,Ke1);
>> K=assem(Edof(2,:),K,Ke2);
>> K=assem(Edof(3,:),K,Ke3)
```



K =

1.0e+008 \*

Columns 1 through 7

|         |   |         |         |         |         |   |
|---------|---|---------|---------|---------|---------|---|
| 0.7500  | 0 | 0       | 0       | -0.7500 | 0       | 0 |
| 0       | 0 | 0       | 0       | 0       | 0       | 0 |
| 0       | 0 | 0.6400  | -0.4800 | -0.6400 | 0.4800  | 0 |
| 0       | 0 | -0.4800 | 0.3600  | 0.4800  | -0.3600 | 0 |
| -0.7500 | 0 | -0.6400 | 0.4800  | 1.3900  | -0.4800 | 0 |
| 0       | 0 | 0.4800  | -0.3600 | -0.4800 | 0.8600  | 0 |
| 0       | 0 | 0       | 0       | 0       | 0       | 0 |
| 0       | 0 | 0       | 0       | 0       | -0.5000 | 0 |

Column 8

|         |
|---------|
| 0       |
| 0       |
| 0       |
| 0       |
| 0       |
| -0.5000 |
| 0       |
| 0.5000  |

Considering the prescribed displacements in **bc**, the system of equations is solved using the function **solveq**, yielding displacements **a** and support forces **r**.

```
>> bc= [1 0;2 0;3 0;4 0;7 0;8 0];
>> [a,r]=solveq(K,f,bc)
```

a =

1.0e-002 \*

|         |
|---------|
| 0       |
| 0       |
| 0       |
| 0       |
| -0.0398 |
| -0.1152 |
| 0       |
| 0       |

```

r =

1.0e+004 *

    2.9845
         0
   -2.9845
    2.2383
    0.0000
    0.0000
         0
    5.7617

```

The vertical displacement at the point of loading is 1.15 mm. The section forces `es1`, `es2` and `es3` are calculated using `bar2s` from element displacements `ed1`, `ed2` and `ed3` obtained using `extract`.

```

>> ed1=extract_ed(Edof(1,:),a);
>> es1=bar2s(ex1,ey1,ep1,ed1)

```

```

es1 =

1.0e+004 *

   -2.9845
   -2.9845

```

```

>> ed2=extract_ed(Edof(2,:),a);
>> es2=bar2s(ex2,ey2,ep2,ed2)

```

```

es2 =

1.0e+004 *

    5.7617
    5.7617

```

```

>> ed3=extract_ed(Edof(3,:),a);
>> es3=bar2s(ex3,ey3,ep3,ed3)

```

```

es3 =

1.0e+004 *

    3.7306
    3.7306

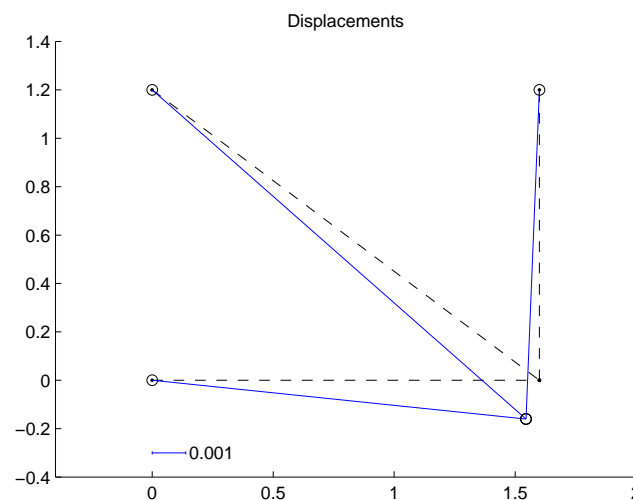
```

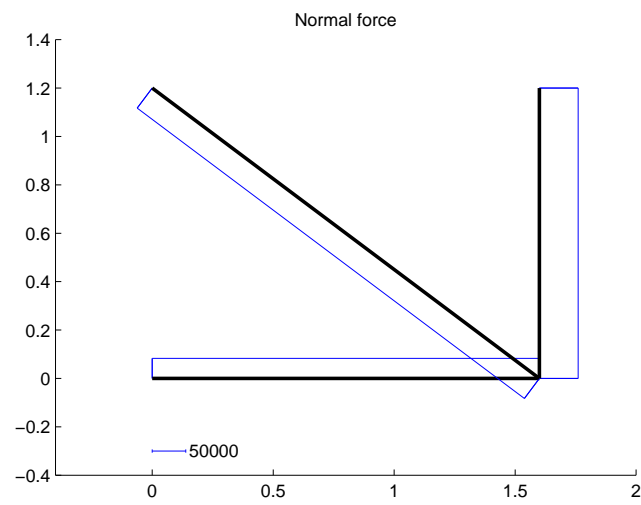
i.e., the normal forces are  $N_1 = -29.84$  kN,  $N_2 = 57.62$  kN and  $N_3 = 37.31$  kN.

A displacement diagram is displayed using the function `eldisp2` and normal force diagram using the function `secforce2`.

```
>> figure(1)
>> plotpar=[2 1 0];
>> eldraw2(ex1,ey1,plotpar);
>> eldraw2(ex2,ey2,plotpar);
>> eldraw2(ex3,ey3,plotpar);
>> sfac=scalfact2(ex1,ey1,ed1,0.1);
>> plotpar=[1 2 1];
>> eldisp2(ex1,ey1,ed1,plotpar,sfac);
>> eldisp2(ex2,ey2,ed2,plotpar,sfac);
>> eldisp2(ex3,ey3,ed3,plotpar,sfac);
>> axis([-0.4 2.0 -0.4 1.4]);
>> scalgraph2(sfac,[1e-3 0 -0.3]);
>> title('Displacements')

>> figure(2)
>> plotpar=[2 1];
>> sfac=scalfact2(ex1,ey1,N2(:,1),0.1);
>> secforce2(ex1,ey1,N1(:,1),plotpar,sfac);
>> secforce2(ex2,ey2,N2(:,1),plotpar,sfac);
>> secforce2(ex3,ey3,N3(:,1),plotpar,sfac);
>> axis([-0.4 2.0 -0.4 1.4]);
>> scalgraph2(sfac,[5e4 0 -0.3]);
>> title('Normal force')
```



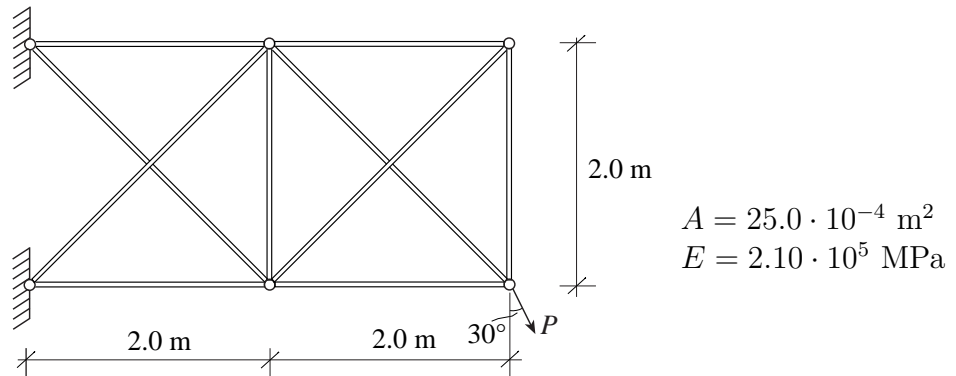


**Purpose:**

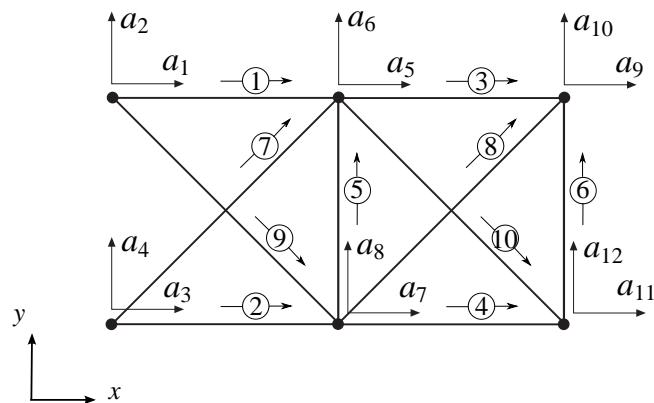
Analysis of a plane truss.

**Description:**

Consider a plane truss, loaded by a single force  $P = 0.5$  MN.



The corresponding finite element model consists of ten elements and twelve degrees of freedom.



The topology is defined by the matrix

```
>> Edof=[1  1  2  5  6;
          2  3  4  7  8;
          3  5  6  9 10;
          4  7  8 11 12;
          5  7  8  5  6;
          6 11 12  9 10;
          7  3  4  5  6;
          8  7  8  9 10;
          9  1  2  7  8;
          10 5  6 11 12];
```

A global stiffness matrix  $K$  and a load vector  $f$  are defined. The load  $P$  is divided into  $x$  and  $y$  components and inserted in the load vector  $f$ .

```
>> K=zeros(12);
>> f=zeros(12,1); f(11)=0.5e6*sin(pi/6); f(12)=-0.5e6*cos(pi/6);
```

The element matrices  $K_e$  are computed by the function `bar2e`. These matrices are then assembled in the global stiffness matrix using the function `assem`.

```
>> A=25.0e-4; E=2.1e11; ep=[E A];
```

```
>> Ex=[0 2;
      0 2;
      2 4;
      2 4;
      2 2;
      4 4;
      0 2;
      2 4;
      0 2;
      2 4];
```

```
>> Ey=[2 2;
      0 0;
      2 2;
      0 0;
      0 2;
      0 2;
      0 2;
      0 2;
      0 2;
      2 0;
      2 0];
```

All the element matrices are computed and assembled in the loop

```
>> for i=1:10
      Ke=bar2e(Ex(i,:),Ey(i,:),ep);
      K=assem(Edof(i,:),K,Ke);
    end;
```

The displacements in  $a$  and the support forces in  $r$  are computed by solving the system of equations considering the boundary conditions in  $bc$ .

```
>> bc=[1 0;2 0;3 0;4 0];
>> [a,r]=solveq(K,f,bc)
```

```
a =
```

```

    0
    0
    0
    0
  0.0024
 -0.0045
 -0.0016
 -0.0042
  0.0030
 -0.0107
 -0.0017
 -0.0113

```

```
r =
```

```

1.0e+005 *

 -8.6603
  2.4009
  6.1603
  1.9293
  0.0000
 -0.0000
 -0.0000
 -0.0000
  0.0000
  0.0000
  0.0000
  0.0000

```

The displacement at the point of loading is  $-1.7 \cdot 10^{-3}$  m in the x-direction and  $-11.3 \cdot 10^{-3}$  m in the y-direction. At the upper support the horizontal force is  $-0.866$  MN and the vertical  $0.240$  MN. At the lower support the forces are  $0.616$  MN and  $0.193$  MN, respectively.

Normal forces are evaluated from element displacements. These are obtained from the global displacements **a** using the function `extract_ed`. The normal forces are evaluated using the function `bar2s`.

```

ed=extract_ed(Edof,a);

>> for i=1:10
    es=bar2s(Ex(i,:),Ey(i,:),ep,ed(i,:));
    N(i,:)=es(1);
end

```

The obtained normal forces are

```
>> N
```

```
N =
```

```
1.0e+005 *  
  
 6.2594  
-4.2310  
 1.7064  
-0.1237  
-0.6945  
 1.7064  
-2.7284  
-2.4132  
 3.3953  
 3.7105
```

The largest normal force  $N = 0.626$  MN is obtained in element 1 and is equivalent to a normal stress  $\sigma = 250$  MPa.

To reduce the quantity of input data, the element coordinate matrices **Ex** and **Ey** can alternatively be created from a global coordinate matrix **Coord** and a global topology matrix **Coord** using the function **coor dxtr**, i.e.

```
>> Coord=[0 2;  
          0 0;  
          2 2;  
          2 0;  
          4 2;  
          4 0];
```

```
>> Dof=[ 1 2;  
        3 4;  
        5 6;  
        7 8;  
        9 10;  
       11 12];
```

```
>> [ex,ey]=coor dxtr(Edof,Coord,Dof,2);
```

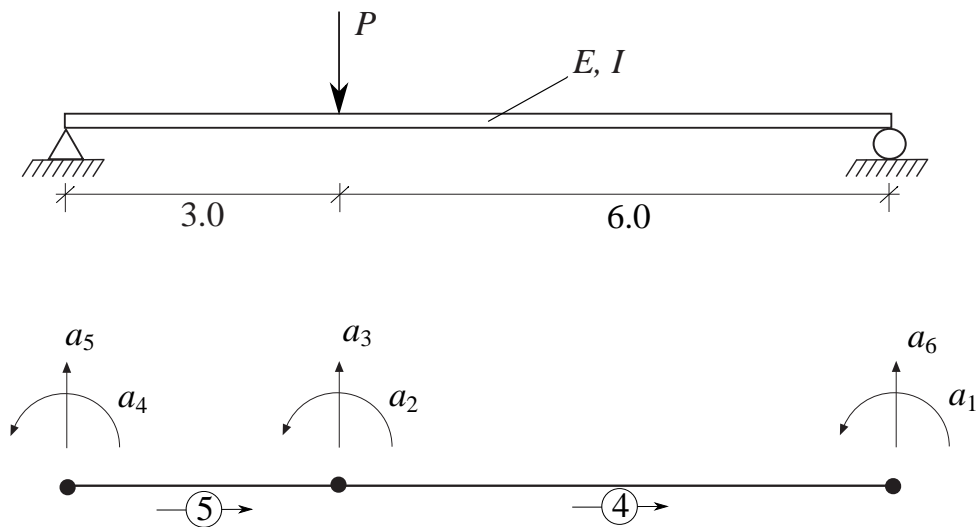


**Purpose:**

Analysis of a simply supported beam.

**Description:**

Consider a beam with the length 9.0 m. The beam is simply supported and loaded by a point load  $P = 10000$  N applied at a point 3.0 m from the left support. The corresponding computational model has six degrees of freedom and consists of two beam elements with four degrees of freedom. The beam has Young's modulus  $E = 210$  GPa and moment of inertia  $I = 2510 \cdot 10^{-8} \text{ m}^4$ .



The element topology is defined by the topology matrix

```
>> Edof=[1  1  2  3  4;
          2  3  4  5  6];
```

The system matrices, i.e. the stiffness matrix  $\mathbf{K}$  and the load vector  $\mathbf{f}$ , are defined by

```
>> K=zeros(6);    f=zeros(6,1);    f(3)=-10000;
```

The element property vector  $\mathbf{ep}$ , the element coordinate vectors  $\mathbf{ex1}$  and  $\mathbf{ex2}$ , and the element stiffness matrices  $\mathbf{Ke1}$  and  $\mathbf{Ke2}$ , are generated.

```
>> E=210e9;      I=2510e-8;      ep=[E A I];
>> ex1=[0 3];    ex2=[3 9];
```

```
>> Ke1=beam1e(ex1,ep)
```

```
Ke1 =
```

```
1.0e+06 *

    2.3427    3.5140   -2.3427    3.5140
    3.5140    7.0280   -3.5140    3.5140
   -2.3427   -3.5140    2.3427   -3.5140
    3.5140    3.5140   -3.5140    7.0280
```

```
>> Ke2=beam1e(ex2,ep)
```

```
Ke2 =
```

```
1.0e+06 *

    0.2928    0.8785   -0.2928    0.8785
    0.8785    3.5140   -0.8785    1.7570
   -0.2928   -0.8785    0.2928   -0.8785
    0.8785    1.7570   -0.8785    3.5140
```

Based on the topology information, the global stiffness matrix can be generated by assembling the element stiffness matrices

```
>> K=assem(Edof(1,:),K,Ke1);
>> K=assem(Edof(2,:),K,Ke2);
```

Finally, the solution can be calculated by defining the boundary conditions in `bc` and solving the system of equations. Displacements `a` and support forces `r` are computed by the function `solveq`.

```
>> bc=[1 0; 5 0];
[a,r]=solveq(K,f,bc)
```

The section forces `es` are calculated from element displacements `Ed`

```
>> Ed=extract_ed(Edof,a);

>> [es1,edi1]=beam1s(ex1,ep,Ed(1,:),eq,6)

>> [es2,edi2]=beam1s(ex2,ep,Ed(2,:),eq,11)
```

Results

```

a =
      0
    -0.0095
    -0.0228
    -0.0038
      0
    0.0076

r =
      1.0e+003 *
      6.6667
     -0.0000
     -0.0000
     -0.0000
      3.3333
      0

es1 =
      1.0e+004 *
     -0.6667    0.0000
     -0.6667    0.6667
     -0.6667    1.3333
     -0.6667    2.0000

edi1 =
      0
     -0.0093
     -0.0173
     -0.0228

es2 =
      1.0e+004 *
      0.3333    2.0000
      0.3333    1.6667
      0.3333    1.3333
      0.3333    1.0000
      0.3333    0.6667
      0.3333    0.3333
      0.3333   -0.0000

edi2 =
     -0.0228
     -0.0248
     -0.0236
     -0.0199
     -0.0143
     -0.0075
     -0.0000

```

A displacement diagram and section force diagrams are displayed using the function `plot`.

```

figure(1)
hold on;
plot([0 9],[0 0]);
c=plot([0,0:1:3,3:1:9,9],[0;edi1(:,1);edi2(:,1);0]);
set(c,'LineWidth',[2]);
axis([-1 10 -0.03 0.01]);
title('displacements')

```

```

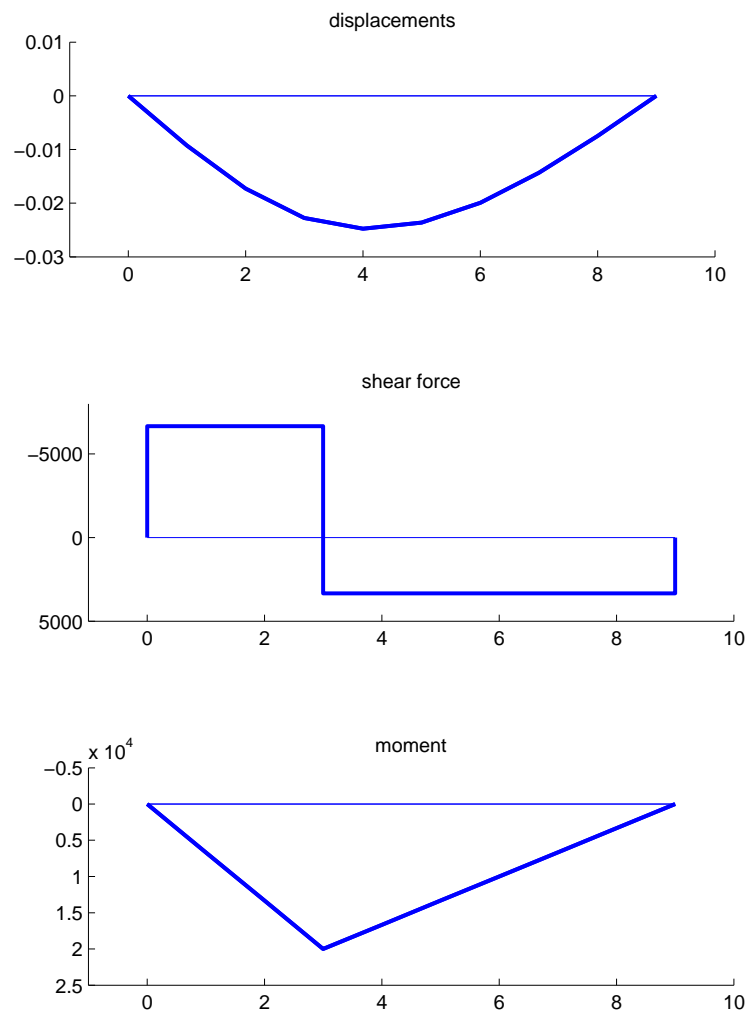
figure(2)
hold on;
plot([0 9],[0 0]);
c=plot([0,0:1:3,3:1:9,9],[0;es1(:,1);es2(:,1);0]);
set(c,'LineWidth',[2]);
axis([-1 10 -8000 5000]);
set(gca, 'YDir','reverse');
title('shear force')

```

```

figure(3)
hold on;
plot([0 9],[0 0]);
c=plot([0,0:1:3,3:1:9,9],[0;es1(:,2);es2(:,2);0]);
set(c,'LineWidth',[2]);
axis([-1 10 -5000 25000]);
set(gca, 'YDir','reverse');
title('moment')

```

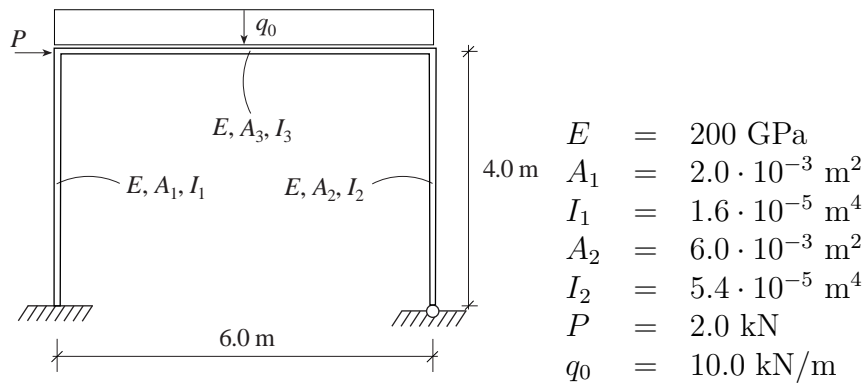


**Purpose:**

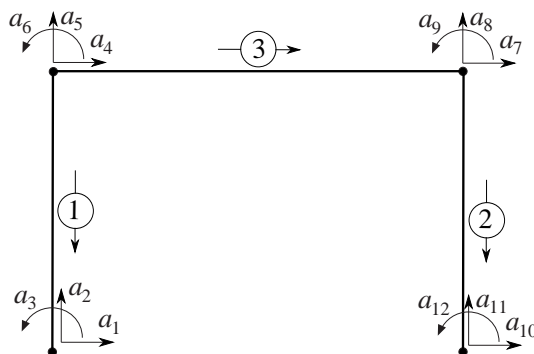
Analysis of a plane frame.

**Description:**

A frame consists of one horizontal and two vertical beams according to the figure.



The corresponding finite element model consists of three beam elements and twelve degrees of freedom.



A topology matrix **Edof**, a global stiffness matrix **K** and load vector **f** are defined. The element matrices **Ke** and **fe** are computed by the function **beam2e**. These matrices are then assembled in the global matrices using the function **assem**.

```
>> Edof=[1  4  5  6  1  2  3;
          2  7  8  9 10 11 12;
          3  4  5  6  7  8  9];

>> K=zeros(12); f=zeros(12,1); f(4)=2e+3;

>> E=200e9;
>> A1=2e-3;  A2=6e-3;
>> I1=1.6e-5; I2=5.4e-5;
>> ep1=[E A1 I1]; ep3=[E A2 I2];
```

```

>> ex1=[0 0]; ex2=[6 6]; ex3=[0 6];
>> ey1=[0 4]; ey2=[0 4]; ey3=[4 4];
>> eq1=[0 0]; eq2=[0 0]; eq3=[0 -10e+3];

>> Ke1=beam2e(ex1,ey1,ep1);
>> Ke2=beam2e(ex2,ey2,ep1);
>> [Ke3,fe3]=beam2e(ex3,ey3,ep3,eq3);

>> K=assem(Edof(1,:),K,Ke1);
>> K=assem(Edof(2,:),K,Ke2);
>> [K,f]=assem(Edof(3,:),K,Ke3,f,fe3);

```

The system of equations are solved considering the boundary conditions in bc.

```

>> bc=[1 0; 2 0; 3 0; 10 0; 11 0];
>> [a,r]=solveq(K,f,bc)

```

```

a =                                r =

      0                        1.0e+004 *
      0
      0                        0.1927
    0.0075                      2.8741
   -0.0003                      0.0445
   -0.0054                      0
    0.0075                      0.0000
   -0.0003                     -0.0000
    0.0047                     -0.0000
      0                          0
      0                        0.0000
   -0.0052                     -0.3927
                                3.1259
                                0

```

The element displacements are obtained from the function **extract**, and the function **beam2s** computes the section forces and the displacements along the element.

```

>> Ed=extract_ed(Edof,a);
>> [es1,edi1]=beam2s(ex1,ey1,ep1,Ed(1,:),eq1,21)
es1 =                                edi1 =

    1.0e+004 *                      0.0003    0.0075
                                0.0003    0.0065
   -2.8741    0.1927    0.8152                      .
   -2.8741    0.1927    0.7767                      0.0000    0.0000
      .
   -2.8741    0.1927    0.0445

```

```
>> [es2,edi2]=beam2s(ex2,ey2,ep1,Ed(2,:),eq2,21)
```

```
es2 =                                edi2 =

    1.0e+004 *                      0.0003    0.0075
                                0.0003    0.0084
    -3.1259    -0.3927    -1.5707
    -3.1259    -0.3927    -1.4922
    .
    -3.1259    -0.3927    -0.0000
                                0.0000    0.0000
```

```
>> [es3,edi3]=beam2s(ex3,ey3,ep3,Ed(3,:),eq3,21)
```

```
es3 =                                edi3 =

    1.0e+004 *                      0.0075   -0.0003
                                0.0075   -0.0019
    -0.3927    -2.8741    -0.8152
    -0.3927    -2.5741     0.0020
    .
    -0.3927     3.1259    -1.5707
                                0.0075   -0.0003
```

A displacement diagram is displayed using the function `dispbeam2` and section force diagrams using the function `secforce2`.

```
>> figure(1)
>> plotpar=[2 1 0];
>> eldraw2(ex1,ey1,plotpar);
>> eldraw2(ex2,ey2,plotpar);
>> eldraw2(ex3,ey3,plotpar);
>> sfac=scalfact2(ex3,ey3,Ed(3,:),0.1);
>> plotpar=[1 2 1];
>> dispbeam2(ex1,ey1,edi1,plotpar,sfac);
>> dispbeam2(ex2,ey2,edi2,plotpar,sfac);
>> dispbeam2(ex3,ey3,edi3,plotpar,sfac);
>> axis([-1.5 7.5 -0.5 5.5]);
>> scalgraph2(sfac,[1e-2 0.5 0]);
>> title('Displacements')
```

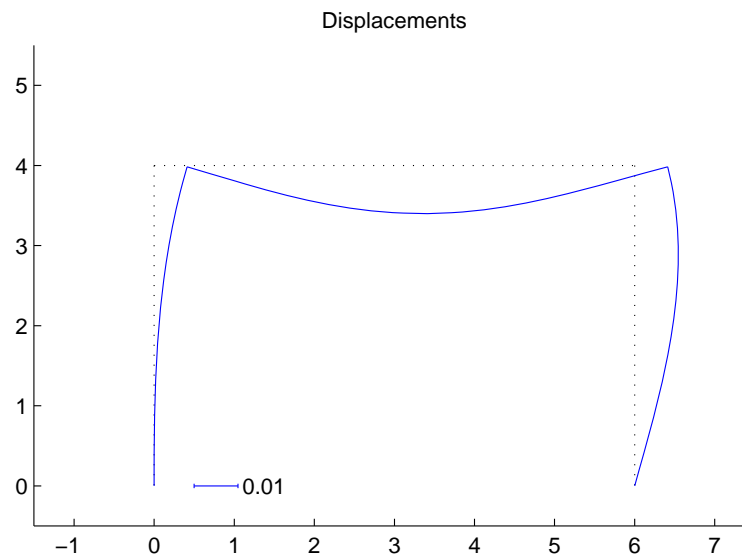
```

>> figure(2)
>> plotpar=[2 1];
>> sfac=scalfact2(ex1,ey1,es1(:,1),0.2);
>> secforce2(ex1,ey1,es1(:,1),plotpar,sfac);
>> secforce2(ex2,ey2,es2(:,1),plotpar,sfac);
>> secforce2(ex3,ey3,es3(:,1),plotpar,sfac);
>> axis([-1.5 7.5 -0.5 5.5]);
>> scalgraph2(sfac,[3e4 1.5 0]);
>> title('Normal force')

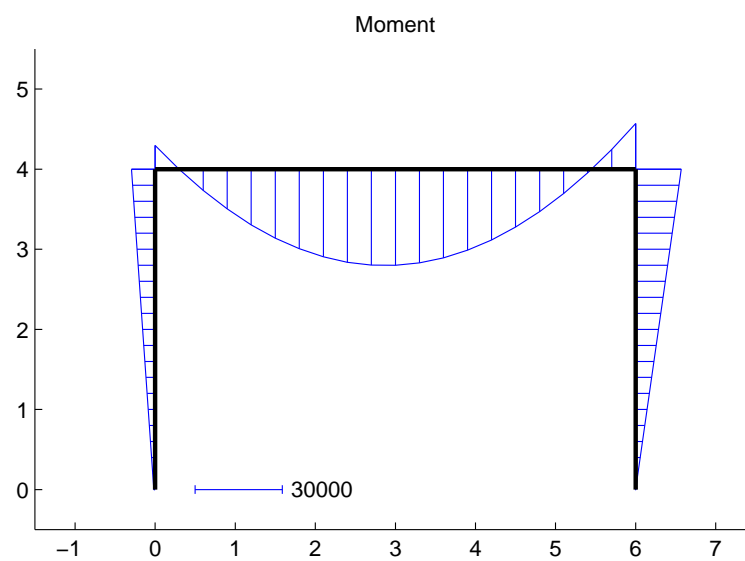
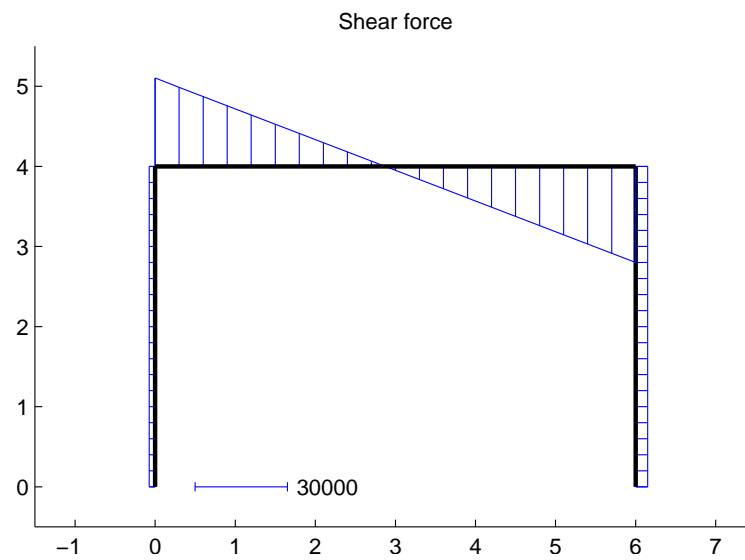
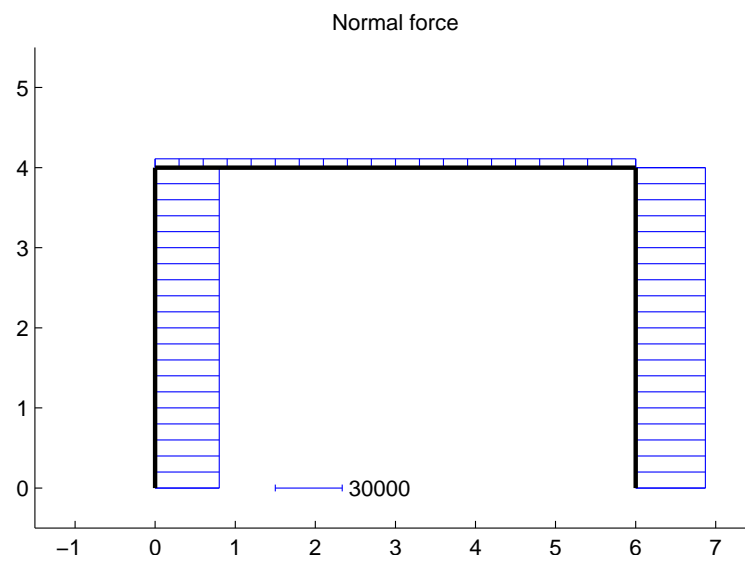
>> figure(3)
>> plotpar=[2 1];
>> sfac=scalfact2(ex3,ey3,es3(:,2),0.2);
>> secforce2(ex1,ey1,es1(:,2),plotpar,sfac);
>> secforce2(ex2,ey2,es2(:,2),plotpar,sfac);
>> secforce2(ex3,ey3,es3(:,2),plotpar,sfac);
>> axis([-1.5 7.5 -0.5 5.5]);
>> scalgraph2(sfac,[3e4 0.5 0]);
>> title('Shear force')

>> figure(4)
>> plotpar=[2 1];
>> sfac=scalfact2(ex3,ey3,es3(:,3),0.2);
>> secforce2(ex1,ey1,es1(:,3),plotpar,sfac);
>> secforce2(ex2,ey2,es2(:,3),plotpar,sfac);
>> secforce2(ex3,ey3,es3(:,3),plotpar,sfac);
>> axis([-1.5 7.5 -0.5 5.5]);
>> scalgraph2(sfac,[3e4 0.5 0]);
>> title('Moment')

```





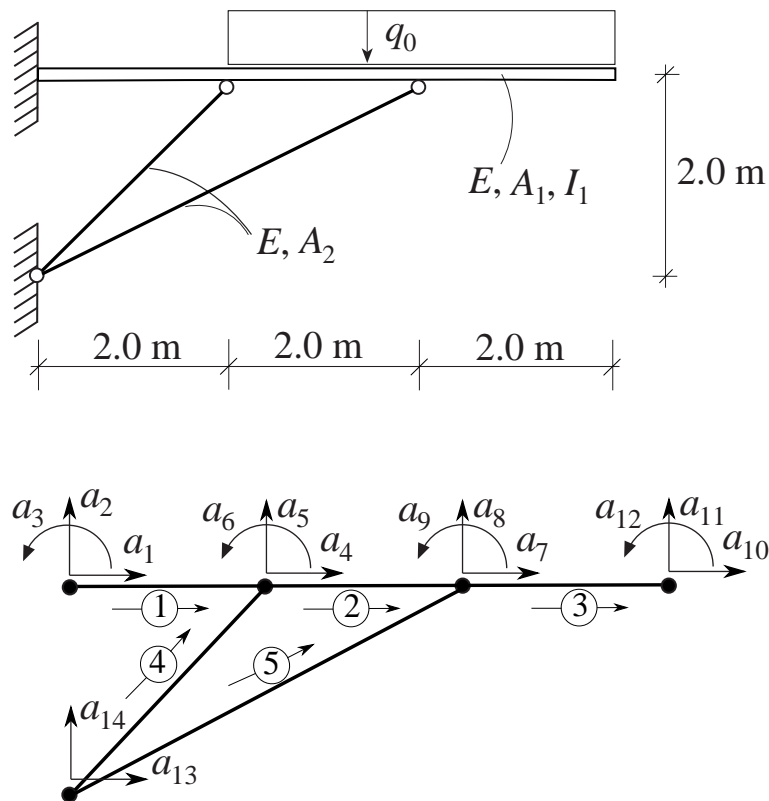


**Purpose:**

Analysis of a combined beam and bar structure.

**Description:**

Consider a structure consisting of a beam with  $A_1 = 4.0 \cdot 10^{-3} \text{ m}^2$  and  $I_1 = 5.4 \cdot 10^{-5} \text{ m}^4$  supported by two bars with  $A_2 = 1.0 \cdot 10^{-3} \text{ m}^2$ . The beam as well as the bars have  $E = 200 \text{ GPa}$ . The structure is loaded by a distributed load  $q = 10 \text{ kN/m}$ . The corresponding finite element model consists of three beam elements and two bar elements and has 14 degrees of freedom.



The computation is initialised by defining the topology matrix **Edof1** for the beam elements and **Edof2** for the bar elements. The matrix **K** ( $14 \times 14$ ), and vector **f** ( $14 \times 1$ ) are created and filled with zeros.

```
>> Edof1=[1  1  2  3  4  5  6;
>>         2  4  5  6  7  8  9;
>>         3  7  8  9 10 11 12];
>> Edof2=[4 13 14  4  5;
>>         5 13 14  7  8];
>>
>> K=zeros(14); f=zeros(14,1);
```

The element property vectors **ep1** and **ep2** and the element coordinate vectors **ex1**, **ex2**, **ex3**, **ex4**, **ex5**, **ey1**, **ey2**, **ey3**, **ey4** and **ey5** are defined.

```
>> E=200e9;    A1=4.0e-3;    A2=1.0e-3;    I1=5.4e-5;
>>
>> ep1=[E A1 I1];    ep4=[E A2];
>>
>> eq1=[0 0];        eq2=[0 -10e3];
>>
>> ex1=[0 2];        ey1=[2 2];
>> ex2=[2 4];        ey2=[2 2];
>> ex3=[4 6];        ey3=[2 2];
>> ex4=[0 2];        ey4=[0 2];
>> ex5=[0 4];        ey5=[0 2];
```

The element stiffness matrices **Ke1**, **Ke2** and **Ke3** are computed using **beam2e** and **Ke4** and **Ke5** are computed using **bar2e**. Element load vectors **fe2** and **fe3** are also given by **beam2e**.

```
>> Ke1=beam2e(ex1,ey1,ep1);
>> [Ke2,fe2]=beam2e(ex2,ey2,ep1,eq2);
>> [Ke3,fe3]=beam2e(ex3,ey3,ep1,eq2);
>> Ke4=bar2e(ex4,ey4,ep4);
>> Ke5=bar2e(ex5,ey5,ep4);
```

Based on the topology information, the global stiffness matrix **K** and load vector **f** are generated by assembling the element matrices using **assem**.

```
>> K=assem(Edof1(1,:),K,Ke1);
>> [K,f]=assem(Edof1(2,:),K,Ke2,f,fe2);
>> [K,f]=assem(Edof1(3,:),K,Ke3,f,fe3);
>> K=assem(Edof2(1,:),K,Ke4);
>> K=assem(Edof2(2,:),K,Ke5);
```

Considering the prescribed displacements in **bc**, the system of equations is solved using the function **solveq**, yielding displacements **a** and support forces **r**. According to the computation the vertical displacement at the end of the beam is 13.0 mm.

```
>> bc=[1 0; 2 0; 3 0; 13 0; 14 0];
>> [a,r]=solveq(K,f,bc)
```

```

a =
      0
      0
      0
    0.0002
   -0.0006
  -0.0010
   0.0004
  -0.0046
  -0.0033
   0.0004
  -0.0130
  -0.0045
      0
      0

r =
      1.0e+04 *
      -8.0702
     -0.6604
     -0.1403
           0
     -0.0000
     -0.0000
           0
     -0.0000
      0.0000
           0
           0
     -0.0000
      8.0702
      4.6604

```

The section forces `es1`, `es2`, `es3`, `es4` and `es5` are calculated using `bar2s` and `beam2s` from element displacements `ed1`, `ed2`, `ed3`, `ed4` and `ed5` obtained using `extract`. This yields the normal forces  $-35.4$  kN,  $-152.5$  kN in the bars and the maximum moment  $10.00$  kNm in the beam.

```

>> Ed1=extract_ed(Edof1,a);
>> Ed2=extract_ed(Edof2,a);
>>
>> es1=beam2s(ex1,ey1,ep1,Ed1(1,:),eq1,11)
>> es2=beam2s(ex2,ey2,ep1,Ed1(2,:),eq2,11)
>> es3=beam2s(ex3,ey3,ep1,Ed1(3,:),eq2,11)
>> es4=bar2s(ex4,ey4,ep2,Ed2(1,:))
>> es5=bar2s(ex5,ey5,ep2,Ed2(2,:))

```

```

es1 =

      1.0e+04 *

      8.0702      0.6604      0.1403
      8.0702      0.6604      0.0082
           .           .           .
      8.0702      0.6604     -1.1806

```

es2 =

1.0e+04 \*

|        |         |         |
|--------|---------|---------|
| 6.8194 | -0.5903 | -1.1806 |
| 6.8194 | -0.3903 | -1.0825 |
| .      | .       | .       |
| .      | .       | .       |
| 6.8194 | 1.4097  | -2.0000 |

es3 =

1.0e+04 \*

|   |         |         |
|---|---------|---------|
| 0 | -2.0000 | -2.0000 |
| 0 | -1.8000 | -1.6200 |
| . | .       | .       |
| 0 | 0.0000  | -0.0000 |

es4 =

1.0e+04 \*

-3.5376  
-3.5376

es5 =

1.0e+05 \*

-1.5249  
-1.5249



---

### 8.3 Nonlinear analysis

This section illustrates some nonlinear finite element calculations.

| Nonlinear analysis |  |
|--------------------|--|
| exn_bar2g          | Analysis of a plane truss considering geometric nonlinearity |
| exn_beam2g         | Analysis of a plane frame considering geometric nonlinearity |
| exn_beam2g_b       | Buckling analysis of a frame                                 |
| exn_bar2m          | Analysis of a plane truss considering material nonlinearity  |

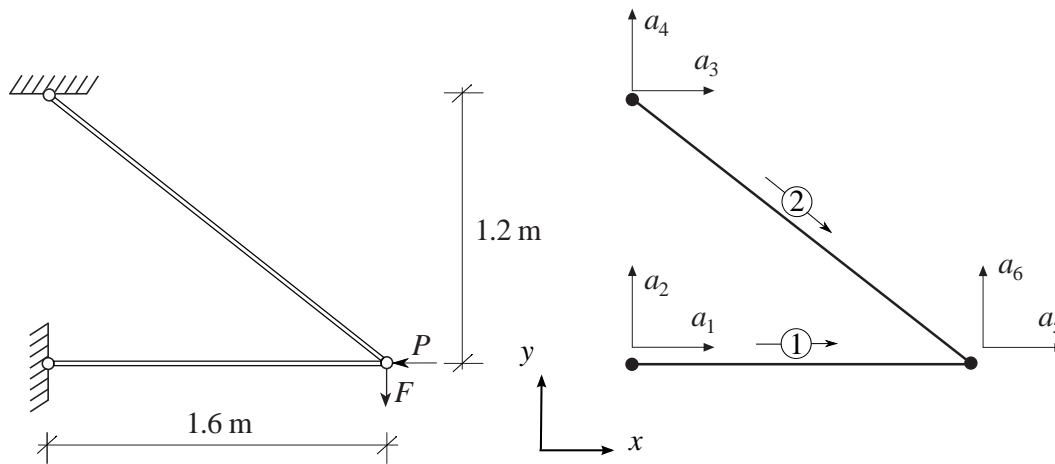
**Note:** The examples listed above are supplied as .m-files under the directory `examples`. The example files are named according to the table.

**Purpose:**

Plane truss considering geometric nonlinearity.

**Description:**

Consider a plane truss consisting of two bars with the properties  $E = 200$  GPa,  $A_1 = 6.0 \cdot 10^{-4}$  m<sup>2</sup> and  $A_2 = 3.0 \cdot 10^{-4}$  m<sup>2</sup>. The truss is loaded by a force  $P = 10$  MN to the left and a force  $F = 0.2$  MN downwards. The corresponding finite element model consists of two elements and six degrees of freedom.



The element property vectors **ep1** and **ep2** and the element coordinate vectors **ex1**, **ex2**, **ey1**, and **ey2** are defined. Initial values are given to the variables axial forces **QX1** and **QX2**. The element stiffness matrices **Ke1** and **Ke2** are computed using **bar2ge**.

The computation is initialised by defining the topology matrix **Edof**, containing element numbers and global element degrees of freedom. The element property vectors **ep1** and **ep2** and the element coordinate vectors **ex1**, **ex2**, **ey1**, and **ey2** are also defined.

```
>> Edof=[1  1  2  5  6;
>>        2  3  4  5  6];
>> E=10e9;
>> A1=4e-2;    A2=1e-2;
>> ep1=[E A1]; ep2=[E A2];
>> ex1=[0 1.6]; ey1=[0 0];
>> ex2=[0 1.6]; ey2=[1.2 0];
```

The bar element function considering geometric nonlinearity **bar2ge** requires the value axial force  $Q_{\bar{x}}$ . Since the axial forces are a result of the computation procedure is iterative. Initially, the axial forces are set to zero, i.e.  $Q_{\bar{x}}^{(1)} = 0$  and  $Q_{\bar{x}}^{(2)} = 0$  which are stored in **QX1** and **QX2**. This means that the first iteration is equivalent to a linear analysis using **bar2e**. To make sure that the first iteration is performed the scalar used for storing the previous axial force in element 1 **QX01** is set to 1. To avoid dividing by 0 in the second convergence check, a nonzero but small value is assumed for the initial axial force in Element 1, i.e.  $Q_{\bar{x},0}^{(1)} = 0.0001$ . In each



iteration the axial forces QX1 and QX2 are updated according to the computational result. The iterations continue until the difference in axial force QX1 of the two latest iterations is less than an accepted error `eps` chosen as  $1.0 \cdot 10^{-6} (QX1 - QX01)/QX01 < \text{eps}$ .

```
>> QX1=0.0001; QX2=0;
>> QX01=1;
>> eps=1e-6;
>> n=0;

>> while(abs((QX1-QX01)/QX01)>eps)
```

In each iteration the global stiffness matrix  $K$  ( $6 \times 6$ ) and the load vector  $f$  ( $6 \times 1$ ) is initially filled with zeros. The nodal loads of 10.0 MN and 0.2 MN acting at lower right corner of the frame are placed in position 5 and 6 of the load vector, respectively. Element stiffness matrices are computed by `bar2ge` and assembled using `assem`, after which the system of equations is solved using `solveq`. Based on the computed displacements  $a$ , new values of section forces and axial forces are computed by `beam2gs`. If QX1 does not converge in 20 iterations the analysis is interrupted.

```
>> n=n+1
>> K=zeros(6,6);
>> f=zeros(6,1);
>> f(5)=-10e6;
>> f(6)=-0.2e6;
>>
>> Ke1=bar2ge(ex1,ey1,ep1,QX1);
>> Ke2=bar2ge(ex2,ey2,ep2,QX2);
>> K=assem(Edof(1,:),K,Ke1);
>> K=assem(Edof(2,:),K,Ke2);
>> bc=[1 0;2 0;3 0;4 0];
>> [a,r]=solveq(K,f,bc)
>>
>> Ed=extract_ed(Edof,a);
>>
>> QX01=QX1;
>> [es1,QX1]=bar2gs(ex1,ey1,ep1,Ed(1,:))
>> [es2,QX2]=bar2gs(ex2,ey2,ep2,Ed(2,:))
>>
>> if(n>20)
>>     disp('The solution does not converge')
>>     break
>> end
>> end
```

After 7 iterations the computation has converged and the axial forces are

QX1 =

-1.1136e+07

QX2 =

1.4833e+06

The displacements according to the linear analysis and the analysis considering geometric nonlinearity are respectively:

a =

0  
0  
0  
0  
-0.0411  
-0.0659

a =

0  
0  
0  
0  
-0.0445  
-0.1088

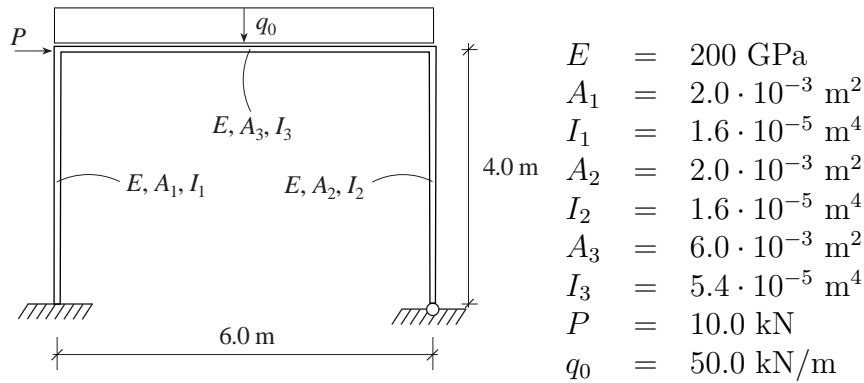
the vertical displacement at the node to the right is 108.8 mm, which is 1.6 times larger than the result from a linear computation according to the first iteration. The axial force in Element 2 is 1.483 kN, which is 4.5 times larger than the value obtained in the linear computation.

**Purpose:**

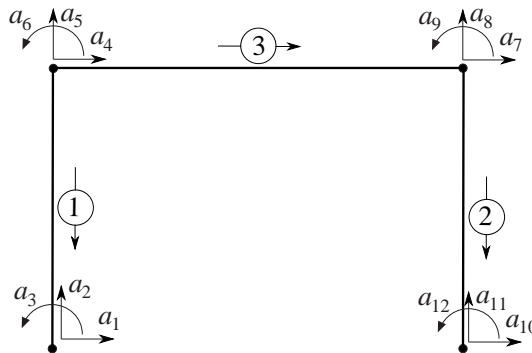
Analysis of a plane frame considering geometric nonlinearity.

**Description:**

The frame of `exs_beam2` is analysed again, but it is now subjected to a load five times larger than in `exs_beam2`. Geometric nonlinearity is considered.



The corresponding computational model consists of three beam elements and twelve degrees of freedom.



The computation is initialised by defining the topology matrix `Edof`, containing element numbers and global element degrees of freedom. The element property vectors `ep1`, `ep2` and `ep3`, the element load vectors `eq1`, `eq2` and `eq3`, and the element coordinate vectors `ex1`, `ex2`, `ex3`, `ey1`, `ey2`, and `ey3` are also defined.

```

>> Edof=[1  4  5  6  1  2  3 ;
>>        2  7  8  9 10 11 12;
>>        3  4  5  6  7  8  9];
>>
>> E=200e9;
>> A1=2e-3;      A2=2e-3;      A3=6e-3;
>> I1=1.6e-5;    I2=1.6e-5;    I3=5.4e-5;

```

```
>> ep1=[E A1 I1]; ep2=[E A2 I2]; ep3=[E A3 I3];
>> eq1=[0];          eq2=[0];          eq3=[-50e3];
>> ex1=[0 0];        ex2=[6 6];        ex3=[0 6];
>> ey1=[4 0];        ey2=[4 0];        ey3=[4 4];
```

The beam element function considering geometric nonlinearity **beam2ge** requires the value axial force  $Q_{\bar{x}}$ . Since the axial forces are a result of the computation the computation procedure is iterative. Initially, the axial forces are set to zero, i.e.  $Q_{\bar{x}}^{(1)} = 0$ ,  $Q_{\bar{x}}^{(2)} = 0$  and  $Q_{\bar{x}}^{(3)} = 0$  which are stored in QX1, QX2 and QX3. This means that the first iteration is equivalent to a linear analysis using **beam2e**. To make sure that the first iteration is performed the scalar used for storing the previous axial force in element 1 QX01 is set to 1. To avoid dividing by 0 in the second convergence check, a nonzero but small value is assumed for the initial axial force in Element 1, i.e.  $Q_{\bar{x},0}^{(1)} = 0.0001$ . In each iteration the axial forces QX1, QX2 and QX3 are updated according to the computational result. The iterations continue until the difference in axial force QX1 of the two latest iterations is less than an accepted error **eps** chosen as  $1.0 \cdot 10^{-6}$   $(QX1 - QX01)/QX01 < \text{eps}$ .

```
>> QX1=0.0001;  QX2=0;  QX3=0;
>> QX01=1;
>> eps=1e-6;
>> n=0;

>> while(abs((QX1-QX01)/QX01)>eps)
```

In each iteration the global stiffness matrix **K** ( $12 \times 12$ ) and the load vector **f** ( $12 \times 1$ ) are initially filled with zeros. The nodal load of 10.0 kN acting at upper left corner of the frame is placed in position 4 of the load vector. Element matrices are computed by **beam2ge** and assembled using **assem**, after which the system of equations is solved using **solveq**. Based on the computed displacements **a**, new values of section forces and axial forces are computed by **beam2gs**. If QX1 does not converge in 20 iterations the analysis is interrupted.

```
>>  n=n+1
>>  K=zeros(12,12);
>>  f=zeros(12,1);
>>  f(4)=10e3;
>>
>>  [Ke1]=beam2ge(ex1,ey1,ep1,QX1);
>>  [Ke2]=beam2ge(ex2,ey2,ep2,QX2);
>>  [Ke3,fe3]=beam2ge(ex3,ey3,ep3,QX3,eq3);
>>
>>  K=assem(Edof(1,:),K,Ke1);
>>  K=assem(Edof(2,:),K,Ke2);
>>  [K,f]=assem(Edof(3,:),K,Ke3,f,fe3);
>>
```

```

>> bc=[1 0;2 0;3 0;10 0;11 0];
>> [a,r]=solveq(K,f,bc)
>>
>> Ed=extract_ed(Edof,a);
>>
>> QX01=QX1;
>> [es1,QX1]=beam2gs(ex1,ey1,ep1,Ed(1,:),QX1,eq1,11)
>> [es2,QX2]=beam2gs(ex2,ey2,ep2,Ed(2,:),QX2,eq2,11)
>> [es3,QX3]=beam2gs(ex3,ey3,ep3,Ed(3,:),QX3,eq3,11)
>>
>> if(n>20)
>>     disp('The solution does not converge')
>>     break
>> end
>> end

```

After 4 iterations the computation has converged and the axial forces are

QX1 =

-1.4242e+05

QX2 =

-1.5758e+05

QX3 =

-1.8163e+04

The displacements according to the linear analysis and the analysis considering geometric nonlinearity are respectively:

a =

```

0
0
0
0.0377
-0.0014
-0.0269
0.0376
-0.0016
0.0233
0
0
-0.0258

```

a =

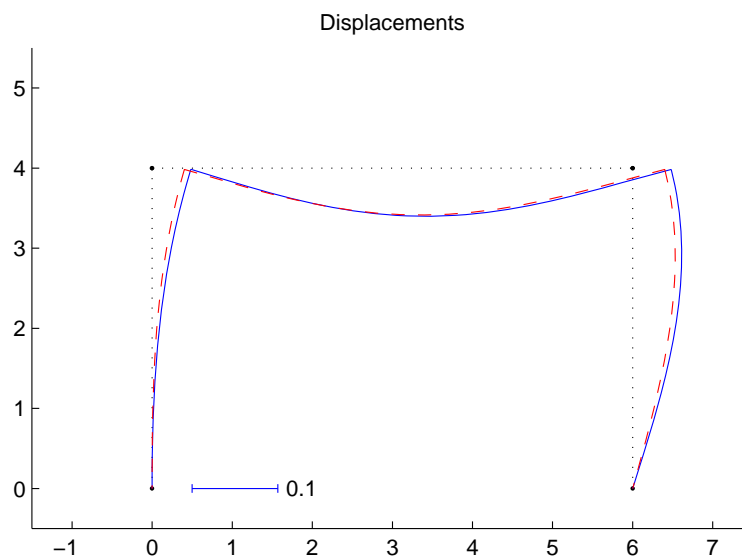
```

0
0
0
0.0451
-0.0014
-0.0281
0.0450
-0.0016
0.0238
0
0
-0.0295

```

Considering geometrical nonlinearity, the horizontal displacement of the upper left corner of the frame increases from 37.7 to 45.1 mm. A displacement diagram is displayed using the function `dispbeam2`. The displacement according to the linear calculation is illustrated using a dashed line and the displacement considering geometrical nonlinearity is illustrated using a solid line.

```
>> figure(1)
>> plotpar=[3 1 0];
>> eldraw2(ex1,ey1,plotpar);
>> eldraw2(ex2,ey2,plotpar);
>> eldraw2(ex3,ey3,plotpar);
>> sfac=scalfact2(ex3,ey3,edi3,0.1);
>> plotpar=[1 2 0];
>> dispbeam2(ex1,ey1,edi1,plotpar,sfac);
>> dispbeam2(ex2,ey2,edi2,plotpar,sfac);
>> dispbeam2(ex3,ey3,edi3,plotpar,sfac);
>> plotpar=[2 4 0];
>> dispbeam2(ex1,ey1,edi10,plotpar,sfac);
>> dispbeam2(ex2,ey2,edi20,plotpar,sfac);
>> dispbeam2(ex3,ey3,edi30,plotpar,sfac);
>> axis([-1.5 7.5 -0.5 5.5]);
>> scalgraph2(sfac,[0.1 0.5 0]);
>> title('Displacements')
```



**Purpose:**

Buckling analysis of a plane frame.

**Description:**

Buckling safety of the frame analysed in `exn_beam2g` is performed. The same computational model as in `exn_beam2g` is used. First, the same computation as in `exn_beam2g` is performed. In this computation the linear stiffness matrix is obtained by saving the stiffness matrix established using the function `assem` in the first iteration, i.e.

```
>> if n==1;
>>     K0=K;
>> end;
```

On the basis of the linear stiffness matrix  $\mathbf{K}_0$  and geometric nonlinear stiffness matrix  $\mathbf{K}_a$  obtained in that computation and stored in `K0` and `K`, the generalised eigen value problem  $(\mathbf{K}_a - \lambda \mathbf{K}_0)\phi = 0$  is established. Considering prescribed displacements specified in `b`, the generalised eigen value problem is solved using `eigen`. Thereafter the loading factors corresponding the buckling modes is computed as  $\alpha_i = \frac{1}{1-\lambda_i}$ . The loading factor corresponding the first buckling mode obtained is  $\alpha_1 = 6.89$ .

```
>> b=bc(:,1);
>> [lambda,phi]=eigen(K,K0,b);
>> nmods=size(lambda);
>> one=ones(nmods);
>> alpha=one./(one-lambda);
>> alpha(1)
>> phi(:,1)
```

```
alpha(1)
```

```
ans =
```

```
6.8904e+00
```

The shape of the frame at buckling is given by the first eigen vector, i.e.

```
phi(:,1)
```

```
ans =
```

```
0
0
0
-1.2708e-03
-2.4706e-06
1.4668e-04
-1.2719e-03
2.4706e-06
-6.8722e-06
0
0
5.3425e-04
```



**Purpose:**

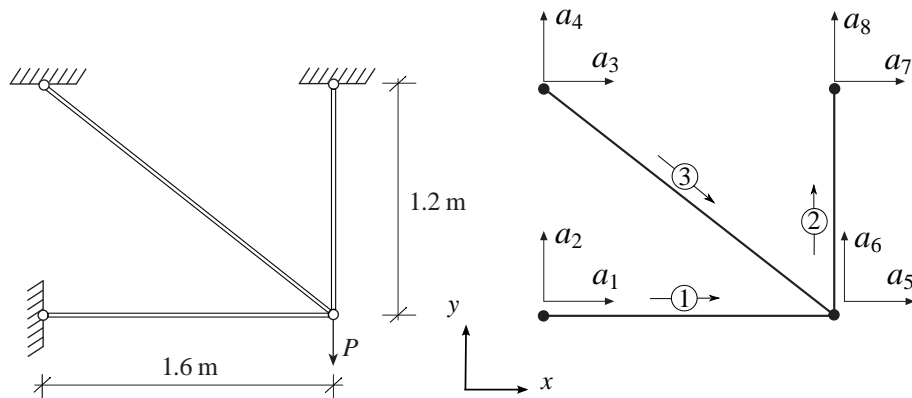
Analysis of a plane truss considering material nonlinearity.

**Description:**

The truss of `exs_bar2` is analysed again, but now the material behaviour is assumed to be nonlinear.

The truss consists of three bars with the properties  $E = 200 \text{ GPa}$ ,  $\sigma_Y = 400.0 \text{ MPa}$ ,  $A_1 = 6.0 \cdot 10^{-4} \text{ m}^2$ ,  $A_2 = 3.0 \cdot 10^{-4} \text{ m}^2$  and  $A_3 = 10.0 \cdot 10^{-4} \text{ m}^2$ . The corresponding finite element model consists of three elements and eight degrees of freedom. The truss is loaded by a single force  $P$ . The load is applied in increments  $\Delta P = 4.0 \text{ kN}$ .

The corresponding computational model consists of three bar elements and eight degrees of freedom.



The computation is initialised by defining the topology, boundary condition, geometry and element property matrices.

```
>> edof=[1 1 2 5 6;
>>       2 5 6 7 8;
>>       3 3 4 5 6]
>>
>> bc=[1 0; 2 0; 3 0; 4 0; 7 0; 8 0];
>>
>> ex=[0 1.6; 1.6 1.6; 0 1.6]
>> ey=[0 0; 0 1.2; 1.2 0]
>>
>> Em=200.0e9
>> E=ones(3,1)*Em;
>> A1=6.0e-4; A2=3.0e-4; A3=10.0e-4
>> A=[A1 A2 A3] '
>> SY=400.0e6
>> Ns=SY*A
```

The computation is performed incrementally. The nodal load is applied in increments, chosen to be  $\Delta P_i = 4$  kN. The limit of the number of increments is chosen to be 100. Matrices for storage of the total values of displacements, support forces and normal forces are defined. Matrices for storage of the number of plastic elements and the force-displacement history are defined.

```
>> dp=4.0e3
>>
>> incr=100;
>>
>> a=zeros(8,1);
>> r=zeros(8,1);
>> es=zeros(3,1);
>>
>> plbar=0;
>> pl(1,:)=[0 0];
```

For each computational step the global stiffness matrix  $\mathbf{K}$  ( $8 \times 8$ ) and the incremental load vector  $\mathbf{df}$  ( $8 \times 1$ ) are initially filled with zeros. The load increment is placed in position 6 of the incremental load vector. Element matrices are computed by `bar2ge` and assembled using `assem`. The determinant of the stiffness matrix is computed to determine whether if the structure has turned to a mechanism and the computation should be interrupted. The system of equations is solved using `solveq`. The increments of displacements and support forces are added to the previously computed total values. Based on the displacement increments, the increments of normal forces are computed and added to the previously obtained normal forces. The computed normal forces are then compared to the yield forces of the elements. If the yield force is exceeded, the modulus of elasticity is set to zero.

```
>> for i=1:incr
>>   K=zeros(8);
>>   df=zeros(8,1);
>>   df(6)=-dp;
>>   for j=1:3
>>       ep=[E(j),A(j)];
>>       Ke=bar2e(ex(j,:),ey(j,:),ep);
>>       K=assem(edof(j,:),K,Ke);
>>   end;
>>   Kr=red(K,bc(:,1));
>>   if det(Kr)<=0
>>       disp(['Determinant zero after increment ',num2str(i-1)])
>>       break;
>>   end;
>>
>>   [da,dr]=solveq(K,df,bc);
>>   a=a+da;
```

```

>> r=r+dr;
>>
>> ded=extract_ed(edof,da);
>> for j=1:3
>>     ep=[E(j),A(j)];
>>     desj=bar2s(ex(j,:),ey(j,:),ep,ded(j,:));
>>     des(j,1)=desj(1);
>> end;
>> es=es+des;
>> for j=1:3
>>     E(j)=Em; if abs(es(j))>=Ns(j); E(j)=0; end
>> end;
>> newplbar=sum(abs(es)>Ns);
>> if newplbar > plbar
>>     plbar=newplbar;
>>     disp([num2str(plbar),' plastic elements for increment ',num2str(i), ...
>>         ' at load = ', num2str(i*dp)])
>>     es
>> end;
>>
>> pl(i+1,:)=[-a(6),i*dp];
>> end;

```

After 42 increments the yield stress has been reached in Element 2 and after 76 increments in Element 1. After 76 increments, for  $P = 304.0\text{kN}$ , the truss becomes a mechanism.

1 plastic elements for increment 42 at load = 168000

es =

```

1.0e+05 *

-0.6267
1.2099
0.7834

```

2 plastic elements for increment 76 at load = 304000

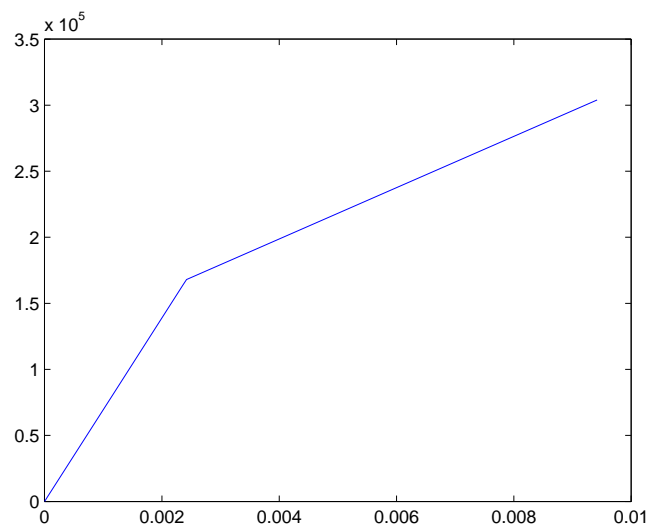
es =

```
1.0e+05 *  
-2.4401  
1.2099  
3.0501
```

Determinant zero after increment 76

A force-displacement diagram is displayed using the function `plot`.

```
>> figure(1)  
>> plot(pl(:,1),pl(:,2),'-');
```



# Index

abs, 24  
assem, 100  
axis, 124  
  
bar1e, 46  
bar1s, 47  
bar1we, 49  
bar1ws, 50  
bar2e, 52  
bar2ge, 56  
bar2gs, 58  
bar2s, 54  
bar3e, 60  
bar3s, 62  
beam1e, 65, 69, 71  
beam1s, 67  
beam2e, 73  
beam2ge, 85  
beam2gs, 87  
beam2s, 76  
beam2we, 79  
beam2ws, 82  
beam3e, 90  
beam3s, 93  
  
clear, 4  
clf, 125  
coordxtr, 101  
  
det, 25  
diag, 26  
diary, 5  
disp, 6  
dispbeam2, 118  
  
echo, 7  
eigen, 103  
eldisp2, 120  
eldraw2, 119  
extract\_ed, 104  
  
figure, 126  
fill, 127  
for, 113  
format, 8  
  
full, 27  
function, 115  
  
grid, 128  
  
help, 9  
hold, 129  
  
if, 112  
insert, 106  
inv, 28  
  
length, 29  
load, 10  
  
max, 30  
min, 31  
  
ones, 32  
  
plot, 130  
print, 131  
  
quit, 11  
  
red, 107  
  
save, 12  
scalfact2, 121  
scalgraph2, 122  
script, 116  
secforce2, 123  
size, 33  
solveq, 108  
sparse, 34  
spring1e, 43  
spring1s, 44  
spy, 35  
sqrt, 36  
statcon, 109  
sum, 37  
  
text, 132  
title, 133  
type, 13  
  
what, 14

---

while, 114

who, 15

whos, 15

xlabel, 134

ylabel, 134

zeros, 38

zlabel, 134