FEMAK - User's manual

Tonni Franke Johansen, Ulf R. Kristiansen, Marc Dhainaut, B.Brouard

March 14, 1996

Preface

This is Femak User's guide ver.1.0 .
Until recently FEMAK has primarely been used by its developers. If we had any problems using it, we could just check the code.
I hope this document will help you in using FEMAK without being one of the developers. You may have suggestions on changes for the document, and you will find errors in the text. Please help make me aware of the possibilities to make this a usable document.
I would like to thank to the members of the finite element group at Group for acoustics, Norwegian University of Science and Technology (NUST) for their contributions to this manual. The members are/were: prof.U.R.Kristiansen, dr.B.Brouard, M.Dhainaut.
Further, I wish to thank prof.T.E.Vigran for his comments and suggestions on the manual.
I started the the work on this version in 1994-1995. In this period I was employed at NUST, financed by the European Unions program Human Capital and Mobility, under the program Sound and Vibration Network in Europe (SAVANTE).
Tonni Franke Johansen, editor of Femak ver.1.0 - User's guide.

Tonni F. Johansen
Group for acoustics
Norwegian University of Science and Technology, Trondheim, Norway

E-mail: johanse1@tele.unit.no

My new address is at Sør-Trøndelag College:

Tonni F. Johansen
Dep. Electrotechnical Engineering.

Sor-Trondelag College, Trondheim, Norway

E-mail: tonni@iet.hist.no

Contents

Chapter 1

Introduction

FEMAK is a finite element program developed for acoustic purposes. FEMAK is developed at the group for acoustics, Institute for telecommunication, Norwegian University of Science and technology, Trondheim. For several years there has been activity on numerical methods in acoustics in the group. The finite element method (FEM) and the boundary element method (BEM) have been applied. The interest of the staff is both in development of the methods, and in using programs as a research tool in acoustics. FEMAK is developed because of the need of a program to integrate the different standalone programs made earlier and as a framework for future research.
Many other FEM programs exists that may take acoustic waves into account. Some are dedicated for acoustic waves, while most of them are structural mechanics programs, where the treatment of acoustic waves is an add-on. These are normally not well suited for purely acoustic problems.
We seeked an established code that could help us with the administration of the data and the solutions of the equations. We found that the code presented by Dhatt and Touzot [1] was suitable for the purpose. That code has been extended with code suitable to solve acoustics problems. We have introduced complex matrices in the program, and FEMAK use files differently from [1].
This manual contains:

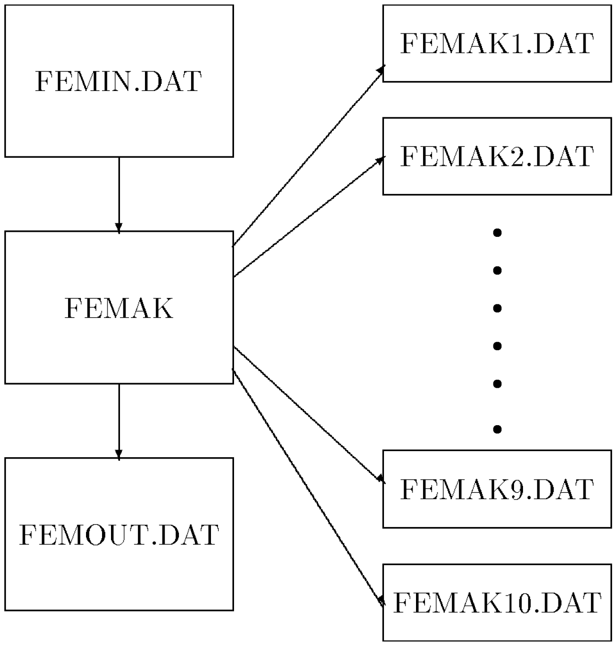
- A specification of the input data in ch.2 .
- The elements are described in ch.3. The description contains:
  - How to use the elements, geometry, material parameters, sources, impedances
  - theory of element
- Finally, some calculated examples are given in app. A.

Many of the ideas used are described in [1]. They will be repeated here.
FEMAK is only used to collect matrices and solve the equations. It has no graphical possibilities. FEMAK may be used with dedicated pre- and post- processors. Presently, we are using PATRAN[1] [2] for this purpose. Simple programs must be made in order to translate the output data from the PATRAN preprocessor to input data for FEMAK and also to make the output data from FEMAK readable for PATRAN.

[1]PATRAN is a trademark of PDA Engineering

1.1 Overview of program execution

As mentioned FEMAK is used to collect and solve equations. The structure of the program is simple.



Structure of FEMAK and its input and output files
The user must support the program with an input file. This file is always named FEMIN.DAT The execution of the program, and the solution of the equations are reported in an output file. This is always named FEMOUT.DAT. Both files are readable ASCII-files. There are more files created during the execution of FEMAK. These are named FEMAK??.DAT, where ?? is a number, 1-10. The files are unformatted. Some of them are files used by FEMAK during its execution, while some holds data from the computations. These may be used by post-processing programs.
The input file, FEMIN.DAT, is used to define the geometry of the model, the materials in the model, the boundary conditions, and the analysis type. The file is divided into blocks, each giving different types information to FEMAK or the user. This file is described in chapter 2.
The output file, FEMOUT.DAT, reports about the execution of the different blocks, and list the result of the analysis. The amount of details in the report are controlled in FEMIN.DAT.
The unformatted files are used as described here:

FEMAK1.DAT Data for elements with real coefficients.

FEMAK2.DAT Data for elements with complex coefficients.

FEMAK3.DAT Data for calculation of residual.

FEMAK4.DAT Result of solution of temporal problem.

FEMAK5.DAT Not used.

FEMAK6.DAT Data for side of an element with impedance.

FEMAK7.DAT Data for sources.

FEMAK8.DAT Not used.

FEMAK9.DAT Not used.

FEMAK10.DAT Result of calculation of dynamical, frequency domain problem.

Chapter 2

Description of command lines in FEMIN.DAT

In this chapter the FEMIN.DAT file is described in detail. All input lines are described shortly, and examples are provided. Each program block is described separately.
Many of the blocks are similarly described [1]. Note that the input formats are specified in this chapter, but it is also possible to use separating commas.

2.1 Block IMAG

This block is used to produce an echo of the input file (FEMIN.DAT) into the output file (FEMOUT.DAT).
Put this command in the start of the input-file in order to make a copy of the input file in the top of the output-file. Only one line.

2.1.1 Description of input lines

block heading
IMAG

2.2 Block COMT

This block is used to input comments.It will always be copied into the output-file. Comment lines may be up to 80 characters long. The block must be terminated by ablank line.

2.2.1 Description of input lines

block heading

COMT

repeated lines

comment lines, max.length 80 characters.

2.2.2 Example

```
COMT
  EigTest1.inp 1/10-1994 Tonni Franke Johansen
  This input file is used to calculate eigenvalues and eigenvectors
  (functions) of an rectangular domain.
  2D, lx=1.0 m, ly=2.0 m, h=0.1m.

COOR
  .
  .
```

2.3 Block COOR

This block is used to define the coordinates of the nodes of the finite element model. It should also be used to input the number of degrees of freedom for each node.
The block is terminated with a line with IN1 = 0.

2.3.1 Description of input lines

block heading

COORm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NNT,NDLN,NDIM,FAC(1),FAC(2),FAC(3)

Definition of variables.

Variables	Columns	Default	Format	Description
NNT	1-5	20	I5	Maximum number of nodes
NDLN	6-10	2	I5	Maximum number of DOF per node.
NDIM	11-15	2	I5	Number of dimensions (1,2 or 3)
				Note NDIM = 20 $\rightarrow$ axisymmetry
FAC(1)	16-25	1.0	F10.0	Scaling factor in direction x
FAC(2)	16-25	1.0	F10.0	Scaling factor in direction y
FAC(3)	16-25	1.0	F10.0	Scaling factor in direction z

repeated lines

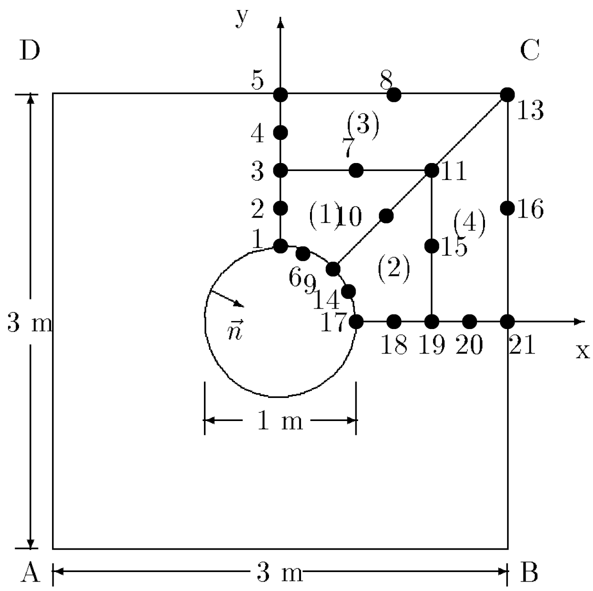
IN1,X1(1),X1(2),X1(3),IN2,X2(1),X2(2),X2(3),INCR,IDLN

Definition of variables.

Variables	Columns	Default	Format	Description
IN1	1-5	-	I5	Number of first node to be generated
X1(1)	6-15	-	F10.0	First node; x coordinate
X1(2)	16-25	-	F10.0	First node; y coordinate
X1(3)	26-35	-	F10.0	First node; z coordinate
IN2	36-40	IN1	I5	Number of last node to be generated
X2(1)	41-50	X1(1)	F10.0	Last node; x coordinate
X2(2)	51-60	X1(2)	F10.0	Last node; y coordinate
X2(3)	61-70	X1(3)	F10.0	Last node; z coordinate
INCR	71-75	1	I5	Pitch of the node numbers
IDLN	76-80	NDLN	I5	Number of DOF generated per node, if different from NDLN

2.3.2 Example

Here a simple example of a mesh is shown. The example is plate with a hole. The geometry is shown in the figure under. Because of symmetry only a quarter of of the plate is modeled. For simplicity the nodes are generated using scaling factors equal to 0.5.




```
COOR
21,1,2,0.5,0.5                         ; 21 node, 1 dof/node, 2D, scale 0.5
1,0.0,1.0,0.0,5,0.0,3.0,0.0,1          ; generate nodes 1 to 5
6,0.3827,0.9239,0.0,8,1.5,3.0,0.0,1    ; generate nodes 6 to 8
9,0.707,0.707,0.0,13,3.0,3.0,0.0,1     ; generate nodes 9 to 13
14,0.9239,0.3827,0.0,16,3.0,1.5,0.0,1  ; generate nodes 14 to 16
17,1.0,0.0,0.0,21,3.0,0.0,0.0,1        ; generate nodes 17 to 21
0                                      ; end of COOR-block
```

2.4 Block DLPN

This block is used to define the number of DOF's for each node. This must be done if the number of DOF's for a node is different from the default, NDLN, which is input in the COOR block. Note that it is possible to define the number of DOF's for a node in the COOR block using the IDLN parameter.
The block is terminated with a line with IDLN = 0.

2.4.1 Description of input lines

block heading

DLPNm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

repeated line

IDLN,K1

Variables	Columns	Default	Format	Description
IDLN	1-5	-	I5	Number of DOF's
K1	6-80	-	15I5	Node numbers (up to 15 on one line, more than 15 nodes with continuation lines (format 5X,15I5))

2.4.2 Example

Ex.1 Node no.1, 2, 3, and 10 has one DOF.

```
DLPN
1,1,2,3,10
0
```

Ex.2 Node 1-15 and 20-22 has two DOF's, while node 30 and 31 has one DOF. All other nodes has defined number of DOF's in the COOR block.

```
DLPN
2,1,2,3,4,5,6,7,8,9,10,11,12,13,14,15
␣␣␣␣␣20,21,22
1,30,31
0
```

2.5 Block COND

This block is used to prescribe values of the nodal unknowns. This is a compulsory block. This block has been modified from COND block of [1] to allow the use of loops when specifying the node numbers.
The block is terminated with a line with ICOD ≤ 0.
Note! Only real values are accepted. Do not use prescribed DOF's $\neq 0$ for frequency domain analysis (DYNM).

2.5.1 Description of input lines

block heading
CONDm m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.
repeated sets of lines ("outer lines")
ICOD,V

Variables	Columns	Default	Format	Description
ICOD	1-10	-	10I1	For each DOF. (Max.10 DOF's) 2 if prescribed by V 1 if prescribed to 0 0 if free
V	11-80	-	7F10.0	Values of prescribed DOF's

repeated sets of lines ("inner lines", belong to one ICOD,V line)
IHOW
KV
The sets of lines is terminated with IHOW = 0.

Variables	Columns	Default	Format	Description
IHOW	1	-	I1	Method for choosing node number 1:all nodes explicitly written 2:loop over the node numbers
KV	1-80	-	16I5	(dependent on IHOW) 1 : Nodenumbers (max. 16 per lines). (more than one line is possible) 2: starting, ending and step for the node numbers (default step size is 1)

2.5.2 Example

Ex.1 Node no.1, 2,...,57,62 and 70 has its DOF set to zero. Note that 16 numbers on a line, indicates a new line of numbers.

```
COND
1                            ; ICOD=1 (set DOF no.1 to zero)
1                            ; IHOW = 1
1,2,3,10,22,31,35,36,40,45,46,47,52,53,54,57       ; 16 first nodes
62,70                                              ;set to zero
0                            ; IHOW = 0, (end ICOD=1)
0                            ; end COND-block
```

Ex.2 Node 1-15 and 20, 22, 241 has DOF no.1 and 2 set to zero, while node 54-73 has DOF 1 set to zero and 4 set to 4.0.
Be careful: no commas for ICOD or between ICOD and their first value of V.

```
COND
11                      ; ICOD=11 (DOF 1 and 2 set to zero)
2                       ; IHOW=2 (looping)
```

```
1,15                    ; Nodes 1-15
2                       ; IHOW=2 (looping)
20,24,2                 ; Nodes 20,22,24
0                       ; IHOW = 0, (end ICOD=11)
10020000004.            ; ICOD=1002,V=4.0 (DOF 1 set to 0 and DOF no 4 to 4.0)
2                       ; IHOW=2 (looping)
54,73                   ; Nodes 54-73
0                       ; IHOW = 0, (end ICOD=1002)
0                       ; end COND-block
```

2.6 Block PRND

This block is used to input nodal properties. A nodal property is a real numeric value. It is only used for grouping nodes for incoming waves. It is possible to input up to 10 different nodal properties. The user must supply the program with the number of nodal properties. The nodal properties are generally given as real numbers. However, nodal property number one is always the node group (i.e. an integer).
This block has been changed from the PRND block in [1]. Its input lines are similar to those of the COND block.
The block is terminated with a line with NODGRP = 0 and all V1=0

2.6.1 Description of input lines

block heading

PRNDm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NPRN

Variables	Columns	Default	Format	Description
NPRN	1-5	0	I5	Number of groups of nodal properties

repeated sets of lines

NODGRP,V1

Variables	Columns	Default	Format	Description
NODGRP	1-5	0	I5	Node group number
V1	6-95	-	9F10.0	Values of nodal properties

repeated sets of lines

IHOW,KV

A set is terminated with a line with IHOW = 0

Variables	Columns	Default	Format	Description
IHOW	1	-	I1	Method for choosing node number 1:all nodes explicitly written 2:loop over the node numbers
KV	1-80	-	16I5	(dependent on IHOW) 1 : Nodenumbers. (more than one line is possible) 2: starting, ending and step for the node numbers (default step size is 1)

2.6.2 Example

One group of nodal properties. This group is to mark up the nodes that are in contact with an incoming wave (the nodes are 274, 276, 285, 287, 289). These nodes are members of node group no.1. This is the only node group.

```
PRND
1                      ; NPRN=1 (one group of nodal properties)
1                      ; NODGRP=1 (node group number)
1                      ; IHOW =1
274,276,285,287,289    ;  the nodes
0                      ; IHOW=0
0                      ; NODGRP=0 (finish PRND block)
```

2.7 Block PREL

This block is used to specify material properties of the different materials. All materials must be specified with the same number of properties. This means that the materials with fewest material properties must be "filled up" with zeros.
The block is terminated with a line with IGPE = 0.

2.7.1 Description of input lines

block heading

PRELm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NGPE,NPRE

Variables	Columns	Default	Format	Description
NGPE	1-5	0	I5	Number of groups of element properties
NPRE	6-10	0	I5	Maximum number of properties in any group

repeated lines

IGPE,V1

Variables	Columns	Default	Format	Description
IGPE	1-5	-	I5	Group number
V1	6-75	-	7F10.0	Values of properties (if NPRE > 7 this V1 must be given on additional lines with maximum 7 properties each)

Note For element type 4 the first parameter specify the model by a floating point number.

2.7.2 Example

Ex.1 Acoustic fluids, i.e. two properties, fluid density and sound velocity. Two different materials.

1 air at 20 °C, $\rho = 1.21\ \frac{kg}{m^3}$, c=340 $\frac{m}{s}$.

2 Water; $\rho = 1000.\ \frac{kg}{m^3}$, c=1500 $\frac{m}{s}$.

```
PREL
2,2                        ; NGPE=2, NPRE=2
1,1.21,340.                ; group 1 - air
2,1000.,1500.              ; group 2 - water
0                          ; end of PREL-block
```

Ex.2 Acoustic fluid and porous material using equivalent fluid model. Two materials. For the equivalent fluid the socalled Johnson-Allard model is used.

1 air at 20 °C, $\rho = 1.21\ \frac{kg}{m^3}$, c=340 $\frac{m}{s}$.

2 porous material. This model use 11 parameters which in this example are:

VPREE(1) - Model type (2.0)

VPREE(2) - ρ, pore fluid density (1.21 kg/m^3)

VPREE(3) - c, wave velocity in pore fluid (340 m/s)

VPREE(4) - B^2, Prandtl's number (0.71)

VPREE(5) - η, viscosity of pore fluid ($1.84 \cdot 10^{-5}$ poseuille

VPREE(6) - $\gamma = c_p/c_v$ of pore fluid (1.4)

VPREE(7) - ϕ porosity (0.90)

VPREE(8) - k_s structure factor (1.5)

VPREE(9) - Φ flow resistivity (10000 Ns/m^4)

VPREE(10) - Λ characteristic length of viscous losses (10^{-4} m)

VPREE(11) - Λ characteristic length of thermal losses (10^{-4} m)

```
PREL
2,11                                 ; NGPE=2, NPRE=11
1,1.21,340.,0.0,0.0,0.0,0.0,0.0      ; group 1 - 2 parameters+ zeros
0.0,0.0,0.0,0.0                      ;       more zeros
2,2.0,1.21,340.,0.71,1.84e-5,1.4,0.90  ; group 2 - porous material
1.50,10000.0,1.0e-4,1.0e-4           ;
0                                    ; end of PREL-block
```

2.8 Block ELEM

This block is used to define the elements. The elements are defined by element-type connectivity of nodes and material properties. It is possible to generate multiple elements on a line. Note that this block is somewhat changed from [1].
The block is terminated with a line with IEL = 0.

2.8.1 Description of input lines

block heading

ELEMm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NELT,NELTR,NELTC,NNEL,NTPE,NGRE,NSYM,NIDENT

Variables	Columns	Default	Format	Description
NELT	1-5	20	I5	Total number of elements
NELTR	6-10	-	I5	Number of elements with real parameters
NELTC	11-15	-	I5	Number of elements with complex parameters
NNEL	16-20	8	I5	Maximum number of nodes per elements
NTPE	21-25	1	I5	Default type of element
NGRE	26-30	1	I5	Default element group (not used)
NSYM	31-35	0	I5	Parameters for symmetry of equations NSYM = 0, symmetrical equations. NSYM = 1, nonsymmetrical equations.
NIDENT	36-40	0	I5	Used if all elements are identical NIDENT = 0, assume no elements are identical NIDENT = 1, all elements are identical Use default value with DYNM applications.

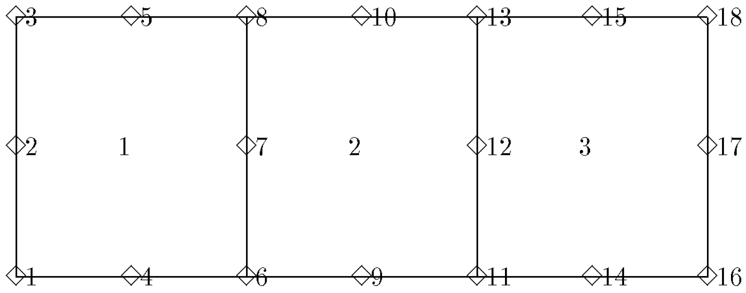
repeated line

IEL,IGEN,INCR,ITPE,IGPE,IGRE,KNE

Variables	Columns	Default	Format	Description
IEL	1-5	-	I5	Number of first element
IGEN	6-10	1	I5	Number of elements to generate
INCR	11-15	1	I5	Nodal number pitch for automatic element generation
ITPE	16-20	NTPE	I5	Element type number
IGPE	21-25	1	I5	Element properties group number
IGRE	26-30	1	I5	Element group number (not used)
KNE	31-89	-	10I5	Node numbers of the element, terminated by node number = 0. If the elements has more than 10 nodes, multiple lines with upto 16 nodes each must be used.

2.8.2 Examples

Three examples will be given, all with the geometry, node-numbering and element-numbering as shown below.



Ex.1 All elements are defined, one by one.
All elements are of type 3, which has real coefficients. Use only 1 material type (IGPE=1)

```
ELEM
3,3,0,8,3                              ; NELT=NELTR=3, NNEL=8, NTPE=3
1,1,1,3,1,1,1,4,6,7,8,5,3,2            ; Definition of element 1
2,1,1,3,1,1,6,9,11,12,13,10,8,7        ; Definition of element 2
3,1,1,3,1,1,11,14,16,17,18,15,13,12    ; Definition of element 3
0                                      ; End of block
```

Ex.2 All elements are generated in one statement.
All elements are of type 3, which has real coefficients. Use only 1 material type (IGPE=1)
The nodal pitch is 5 (INCR=5).

```
ELEM
3,3,0,8,3                              ; NELT=NELTR=3, NNEL=8, NTPE=3
1,3,5,3,1,1,1,4,6,7,8,5,3,2            ; Definition of elements 1-3
0                                      ; End of block
```

Ex.3 Two different materials
The elements 1 and 2 are of material no.1 (IGPE=1), these are generated in one statement.
The nodal pitch is 5 (INCR=5). Element no.3 is of material no.2 (IGPE=2).

```
ELEM
3,3,0,8,3                              ; NELT=NELTR=3, NNEL=8, NTPE=3
1,2,5,3,1,1,1,4,6,7,8,5,3,2            ; Definition of elements 1-2 (IGPE=1).
3,1,1,3,2,1,11,14,16,17,18,15,13,12    ; Definition of elements 3 (IGPE=2).
0                                      ; End of block
```

2.9 Block SRCE

This block is used for definition of acoustic sources, but also fluctuating distributed forces. In many cases this is similar to boundary conditions, where COND or SOLC may be used. Several different sources may be used in combination.

- point-sources with a volume velocity in one or more nodes.
- velocity on a boundary
- incoming waves.

The different sources are grouped in ”source groups”. Different types of groups exists for ease of inputting data. The input is separated in two sections:

1. Defining the different source groups. Some of the groups needs extra data (i.e. group type 0, point source, where all sources have the same volume velocity).
2. Defining each of the sources.

2.9.1 Description of input lines

block heading

SRCEm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NGSRC,NFREQ1,NSRC

Variables	Columns	Default	Format	Description
NGSRC	1-5	-	I5	Number of groups of sources
NFREQ1	6-10	-	I5	Number of frequencies (will be read in other blocks too)
NSRC	11-15	-	I5	total number of sources (sum of point sources, velocity on side of element or other type of source)

1.section
repeated lines

IGSRC,SRCPAR

Variables	Columns	Default	Format	Description
IGSRC	1-5	-	I5	Group number
SRCPAR	6-10	-	I5	Group type number (Note that some groups needs specifications on the following line (i.e.source strengths))

All groups of sources must be defined in this section.

2.section
repeated lines

IEL0,IFL,IGSRC

Variables	Columns	Default	Format	Description
IEL0	1-5	-	I5	Element number
IFL	6-10	-	I5	Element side number, may also be node number for point sources
IGSRC	11-15	-	I5	Source groups. (Note that some sources needs specifications on several lines (i.e. velocities or parameters for incoming wave))

All sources must be defined in this section.
Note please read the individual source type descriptions.

2.9.2 Source type 0, volume source

This source offers the possibility to put a volume source in a node. The node number and the source strength must be supported. It is also possible to use this source to specify a nodal force, note however its frequency dependence. [1]
The value of a source term is calculated as $-j\rho\omega Q$ where Q is the volume velocity (or nodal force).
Syntax:

1. Define the source type using SRCPAR=0
2. Immediately afterwards the source-strengths must be defined. One source strength must be given for each frequency. Only four frequencies at a line.
3. After all source groups are defined, the nodes having a source must be defined by giving its element number(one of the element it belongs to is enough), its node number and its source group number.

```
IGSRC,0                                     ; IGSRC, SRCPAR=0
Re(Q1),Im(Q1),Re(Q2),Im(Q2),Re(Q3),Im(Q3),Re(Q4),Im(Q4)
Re(Q5),Im(Q5),Re(Q6),Im(Q6),Re(Q7),Im(Q7),Re(Q8),Im(Q8)
.               ;define source strength for all frequencies here
.
.
.               ; after definition of source-types, define the sources.
.
IELO,NODNR,IGSRC
```

Example

Ex. Seven frequencies, one source in node 101, element 10. Source strength Q=1 for all frequencies.

```
SRCE
1,7,1                              ; NGSRC=1,NFREQ1=7,NSRC=1
1,0                                ; Use source group 1
1.0,0.0,1.0,0.0,1.0,0.0,1.0,0.0    ; Define the Source Strengths
1.0,0.0,1.0,0.0,1.0,0.0            ; forthe 7 frequencies
10,101,1                           ; Element 10, node 101, grp.1
```

2.9.3 Source type 20 and 21, Defined velocity.

These source-types are used to specify a given normal velocity at the boundary of a calculation domain. They are primarily useful for acoustic waves in fluids (ELEM03). Their only difference is that type 20 needs data for each frequency, while 21 has frequency independent velocity. The velocities must be given in each node. In this manner a velocity distribution may be defined.
The values of the source vector are $-j\omega \mathbf{s_a}\overline{\mathbf{v}}$ where

$$\mathbf{s_a} = \int_{\Gamma_e} \mathbf{N^T} \rho_f \mathbf{N} d\Gamma \tag{2.9-1}$$

[1] Note that this source type may give problems. For a point source, the theoretical value of the pressure should tend to infinity. The variation of the pressure close to the source should vary very rapidly and therefore a very fine mesh is needed. We have found that when using elements of size $\approx \lambda/4$ the FEMAK results and theoretical results are not close, close to the source, while some elements away from the source they agree well. Be careful when using this source-type.

and Γ_e is the side of an element, $\overline{\mathbf{v}}$ is velocity, N is elemental shape functions and ρ_f is the fluid density.
Syntax:

1. Define the source type using SRCPAR=20 or SRCPAR=21

2. After all source groups are defined, the elementnumber, the elementside (see definition in the chapter 3) and the source group must be given. On the following lines the velocities in the nodes of this side must be defined.

```
IGSRC,20         ; IGSRC, SRCPAR=20
.
.                ; after definition of source-types, define the sources.
.
IELO,IFL,IGSRC     ; A source of type 20
Re(Vn1fr1),Im(Vn1fr1),Re(Vn2fr1),Im(Vn2fr1),Re(Vn3fr1),Im(Vn3fr1)
.
.
Re(Vn1frn),Im(Vn1frn),Re(Vn2frn),Im(Vn2frn),Re(Vn3frn),Im(Vn3frn)
```

`Vnpfrq` is velocity in node p at frequency q.

```
IGSRC,21         ; IGSRC, SRCPAR=21
.
.                ; after definition of source-groups, define the sources.
.
IELO,IFL,IGSRC         ; A source of type 21
Re(Vn1),Im(Vn1),Re(Vn2),Im(Vn2),Re(Vn3),Im(Vn3)
```

`Vnp` is the velocity in node p (frequency independent).

Example

Ex. Two frequencies, two sources,

1 Velocity at side 3 on element 16,v = 1.0 m/s for both frequencies. This type of source can use source type 21. Note that the normal is pointing out of the volume, therefore negative values are used for the nodal velocities.

2 Velocity at side 5 on element 120, v = 1.0 m/s for for the first frequency v = 1.0 + j 2.0 m/s at the other frequency. This type of source can use source type 20

```
SRCE
2,2,2                         ; NGSRC=2,NFREQ1=2,NSRC=2
1,21                          ; grp.no.1
2,20                          ; grp.no.2
16,3,1                        ; the first source
-1.0,0.0,-1.0,0.0,-1.0,0.0    ; and its velocities
120,5,2                       ; the second source and its velocities
-1.0,0.0,-1.0,0.0,-1.0,0.0    ; first frequency
-1.0,-2.0,-1.0,-2.0,-1.0-2.0  ; second frequency
```

2.9.4 Source type 50, incoming planewave

This source type is used to model an incoming plane wave. It can be used for transmission loss analysis in ducts or scattering type problems. The theory of this type of source is outlined by Craggs, [3]. We recommend this source type to be used only for duct systems. It should be noted that this source is a very simple implementation of the problem of incoming waves. The incoming wave is only considered at the boundary "seeing" the incoming

wave. This is a different procedure than those which only calculates the reflected/scattered wave, [4]. These other procedures must update the boundary conditions.
The user must specify the nodes on the calculation domain being the "receivers" of the incoming wave. These nodes must be marked as a group of nodes using the PRND block. In the SRCE block the source is specified by its amplitude and direction, and by reference to the group of nodes defined in the PRND block [2]. The boundary that "sees" (that is marked with a group of nodes) the incoming wave will also have a reflected wave. To treat this wave properly an impedance must be specified. It is important to remember that this impedance is to be used for the reflected wave.
The definition of the incoming wave is:

$$p_i(x) = Ae^{-jk(l_x x + l_y y + l_z z)} \tag{2.9-2}$$

where $k = \frac{\omega}{c}$ is the wavenumber, l_x, l_y, and l_z are the directional cosine of the wavenumber in the x-, y- and z-direction. It is necessary that $l_x^2 + l_y^2 + l_z^2 = 1$, note that FEMAK will check this and recalculate the directional cosines of k if necessary. A is the complex amplitude of the wave. Note that the pressure in the origin is A. For this type of source the normal of the boundary must point outwards. The user must supply an outwards pointing normal for the element side by appropriate numbering of nodes. See definition of elements in chapter 3.
Syntax:

1. Mark the nodes receiving the incoming wave in the PRND block.
2. Define the source type using SRCPAR=50
3. After all source groups are defined, the material used must be specified. This is done by referencing one of the elements having nodes that "sees" the incoming wave. Using this method the input will be similar to the other sources too. The following two lines must specify:
   (a) the nodegroup
   (b) A, l_x, l_y and l_z

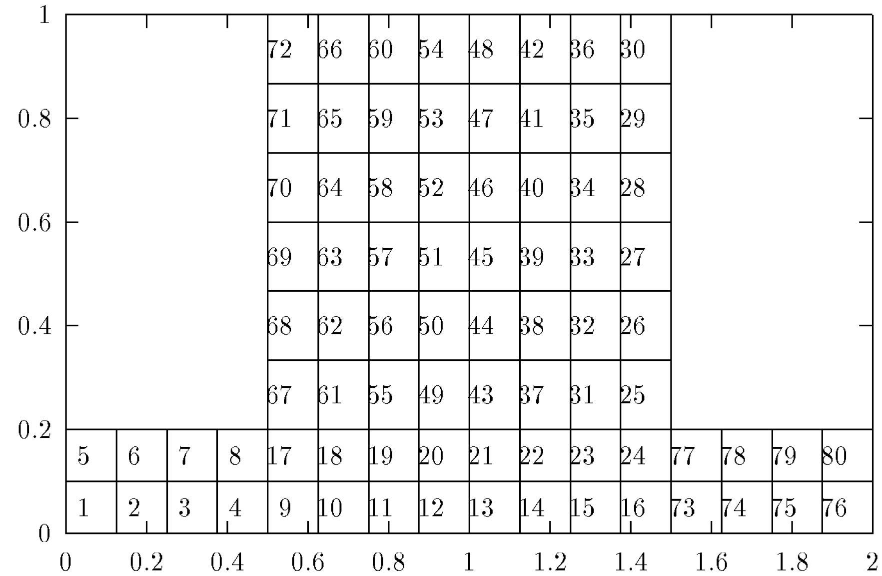
Only real values of l_x, l_y, l_z may be used

```
IGSRC,50                                ; Source type 50 used
.
.
IELO,IFL,IGSRC                          ; give an element to specify material
NODGRP                                  ; The nodegroup
Re(A),Im(A),Re(lx),Im(lx),Re(ly),Im(ly) ; parameters specifying the incoming wave
```

Example Transmission loss in a duct system

Two-dimensional example. The duct under consideration is shown below. A plane wave with amplitude 1.0 Pa is coming in from the left ($l_x = 1.0$, $l_y = 0.0$). Only the nodes at the far left side will be "receiving nodes"(node number 274, 276, 285, 287, 289 (see the input file acouex3.inp)). Note that the elementsides; 2 for element 1 and 5, and elementside 3 for elements 76 and 80 must be defined to have impedance ρc (assuming ducts of infinite extent both to the left and to the right).

[2]In the "receiving nodes" FEMAK will only calculate the reflected pressure, while the total pressure will be calculated in all other nodes. In the FEMOUT.DAT file, the total pressure is printed for all nodes.



1. Mark nodes in PRND block. Choose nodegroup no 1
2. Choose source type 50.
3. Define the incoming wave.
4. Define impedances.

```
.
PRND
1                          ; 1 group of nodal parameter
1                          ; NODGRP=1
1                          ;->| IHOW=1
289,287,274,276,285        ;  |The 5 nodes
0                          ;<-| finished IHOW=1
0                          ; Finish PRND
.
.
SRCE
1,1,1                      ; NGSRC=1, NFREQ1=1, NSRC=1
1,50                       ; incoming wave, type 50
1,2,1                      ; element 1 - specifying material
1                          ; Nodes in NODGRP=1
1.0,0.0,1.0,0.0,0.0,0.0    ; A=1.0, lx=1.0, ly=0.0
.
.
CIMP
1,1,4                      ; NGIMP=1, NFREQ1=1, NIMP=4
1,0                        ; impedance type 0
1.0,0.0                    ; the impedance value
1,2,1                      ; element 1 side 2
5,2,1                      ; element 5 side 2
76,3,1                     ; element 76 side 3
80,3,1                     ; element 80 side 3
```

2.9.5 Source type 55, incoming planewave, used with wave envelope element

This source type is used to model an incoming plane wave. It should be used for scattering, reflection and transmission type problems where an infinite domain is modeled using wave envelope elements, WEE (ELEM11). Wave envelope elements can only consider outgoing waves. Therefore the influence term of the incoming wave is accounted in the interface

nodes between the FEM - mesh (inner domain) and WEEM-mesh (outer domain). As for source type 50 the user must mark these nodes using the PRND-block. FEMAK will make a new group for the outer nodes in those ELEM11's influenced (usually all elements).
It should be noted that this source is a very simple implementation of the problem of incoming waves. The incoming wave is only considered at the boundary "seeing" the incoming wave. This is a different procedure than those which only calculates the reflected/scattered wave, [4].
As already mentioned, the user must specify the nodes on the calculation domain being the "receivers" of the incoming wave (interface between FEM and WEEM domains). These nodes must be marked as a group of nodes using the PRND block. In the SRCE block the source is specified by its amplitude and direction, and by reference to the group of nodes defined in the PRND block [3]. The outer edge of the wave envelope elements must have an appropriate impedance for handling of the reflected/scattered wave.
The definition of the incoming wave is:

$$p_i(x) = Ae^{-jk(l_x x + l_y y + l_z z)} \tag{2.9-3}$$

where $k = \frac{\omega}{c}$ is the wavenumber, l_x, l_y, and l_z are the directional cosine of the wavenumber in the x-, y- and z-direction. It is necessary that $l_x^2 + l_y^2 + l_z^2 = 1$, note that FEMAK will check this and recalculate the directional cosines of k if necessary. A is the complex amplitude of the wave. Note that the pressure in origin is A. The user must supply an outwards pointing normal for the element side by appropriate numbering of nodes. See definition of elements in chapter 3.
This source can only be used for 2D and axisymmetrical cases.
Syntax:

1. Mark the "interface" nodes the incoming wave in the PRND block.
2. Define the source type using SRCPAR=55
3. After all source groups are defined, the material used must be specified. This is done by referencing one of the elements having nodes that "sees" the incoming wave. Using this method the input will be similar to the other sources too. The following two lines must specify:
   (a) the nodegroup
   (b) A, l_x, l_y and l_z

Only real values of l_x, l_y, l_z may be used

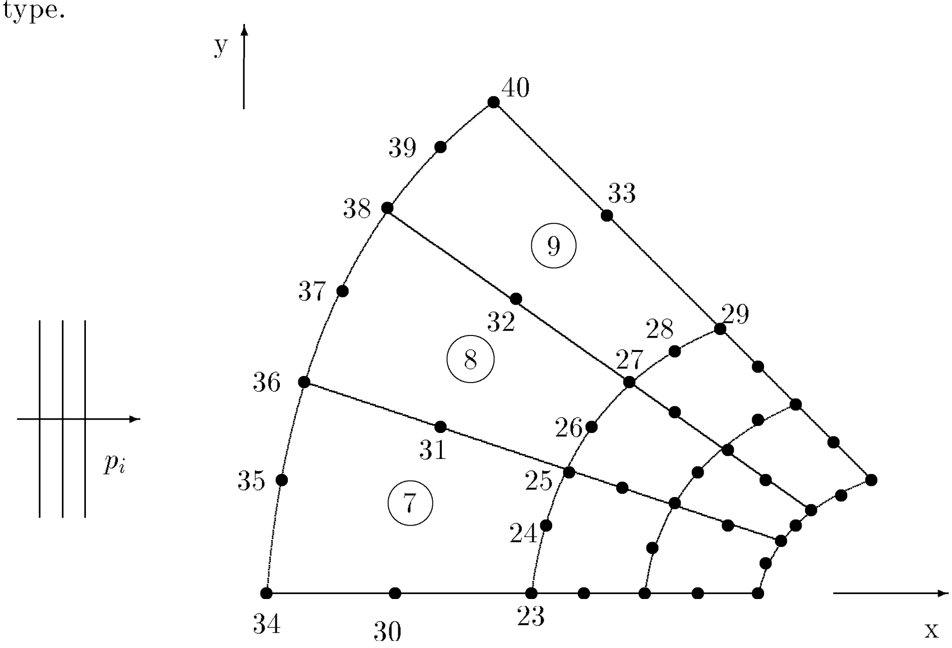
```
IGSRC,55
.
.
IELO,IFL,IGSRC
NODGRP
Re(A),Im(A),Re(lx),Im(lx),Re(ly),Im(ly)
```

Example

Idealized case. Incoming wave to a very simple mesh. FEM-domain consists of 6 elements while WEEM-domain consists of 3 elements (elements 7-9). The incoming wave is horizontal direction and along the positive x-axis.

[3] In the "receiving nodes" and the other nodes in the WEE having incoming waves, FEMAK will only calculate the reflected pressure, while the total pressure will be calculated in all other nodes. In the FEMOUT.DAT file, the total pressure is printed for all nodes.

The incoming wave is "seen" by the nodes interfacing the two domains, that is nodes 23-29. These nodes are marked using the PRND block. Use element 7 to reference the material type.



1. Mark nodes in PRND block. Choose nodegroup no 1
2. Choose source type 55.
3. Define the incoming wave.
4. Define impedances.

```
 .
 .
PRND
1                      ; 1 group of nodal parameters
1                      ; NODGRP=1
2                      ; ->| IHOW=2
23,29,1                ;   | nodes 23 to 29
0                      ; <-| finish IHOW=2
0                      ; finish PRND
 .
 .
SRCE
1,1,1                   ; NGSRC=1, NFREQ1=1, NSRC=1
1,55                    ; Incoming wave, type 55
7,1,1                   ; element 7 - specifying material
1                       ; Nodes in NODGRP=1
1.0,0.0,1.0,0.0,0.0,0.0 ; A=1.0, lx=1.0, ly=0.0
CIMP
1,1,3                   ; NGIMP=1, NFREQ1=1, NIMP=3
1,0                     ; IMPTYP=0
1.0,0.0                 ; impedance = RHO*C
7,3,1                   ; element 7, side 3
8,3,1                   ; element 8, side 3
9,3,1                   ; element 9, side 3
```

Note that here the outer nodes (30-40) will be assigned to a new groups of nodes, group no.2 (because no.1 is the highest group number defined in the PRND block).

2.9.6 Source type 25 and 26, force on a side of an element

These source-types are used to specify a force on a side of an element. They may only be used with element type 5 (porous material using Biot theory) and 20 (elastic solid with loss). It can only be used for 2D and axisymmetric case. The user must supply the force in each node and the direction of the force (constant direction over an element-side).
For source type 25 the nodal forces must be specified for each frequency. For source type 26 the nodal forces are frequency independent.
Syntax:

1. Define the source type using SRCPAR=25 or SRCPAR=26
2. After all source groups are defined, the elementnumber, the elementside (see definition in the chapter 3) and the source group must be given. Immediately afterwards the direction of the forces and the forces in the nodes of this side must be given.

```
IGSRC,25          ; define a source group with source type 25
.
.                 ; after definition of all source-groups:
.
IELO,IFL,IGSRC         ; the element, side and source group
lx,ly                  ; directional cosine of force
Re(F1fr1),Im(F1fr1),Re(F2fr1),Im(F2fr1),Re(F3fr1),Im(F3fr1)
.                      ; complex force in each node
.                      ; for each frequency
Re(F1frn),Im(F1frn),Re(F2frn),Im(F2frn),Re(F3frn),Im(F3frn)
```

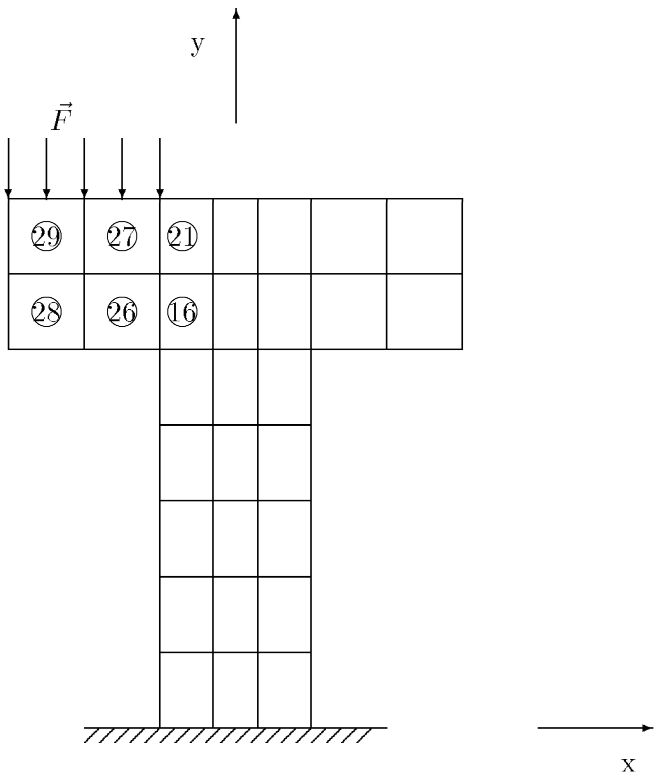
`Fpfrq` is the force in node p at frequency q, while lx and ly is the directional cosine of the force in the x- and y-direction.

```
IGSRC,26               ; define a source group with source type 26
.
.                      ; after definition of all source-groups:
.
IELO,IFL,IGSRC                ; the element, side and source group
lx,ly                         ; directional cosine of force
Re(F1),Im(F1),Re(F2),Im(F2),Re(F3),Im(F3)
     ; complex force in each node
```

`Fp` is the velocity in node p (frequency independent), while lx and ly is the directional cosine of the force in the x- and y-direction.

Example

A simple 2-dimensional model of a T-shaped steel device. The device is clamped at $y = 0.0$. The forces attack at the left side of the "horizontal part of the T", that is at side 5 of element 27 and 29. The force acts downwards (negative y-direction). Source-type 26 is used because the forces are independent of frequency.




```
.
.
SRCE
1,1,2                            ; NGSRC=1,NFREQ1=1,NSRC=2
1,26                             ; Group 1, type 26
27,5,1                           ; IEL=27,IFL=5,IGSRC=1
0.0,1.0                          ; lx=0.0, ly=1.0
-1.0,0.0,-1.0,0.0,-1.0,0.0       ; force in each node
29,5,1                           ; IEL=29,IFL=5,IGSRC=1
0.0,1.0                          ; lx=0.0, ly=1.0
-1.0,0.0,-1.0,0.0,-1.0,0.0       ; force in each node
.
.
```

2.10 Block CIMP

This block is used for definition of impedances. The CIMP block is specially designed for use in acoustics. The impedances must be specified for each frequency. This block is also used to couple FEM-domains to infinite fields. The different impedances are grouped in "impedance groups". Different types of groups exists for ease of inputting data and also for general type of data. Note that the data are specific acoustic impedance normalized with respect to the characteristic impedance of the media used (ρc).
The input is separated in two sections:

1. Defining the different impedance groups. Some of the groups needs extra data (i.e. group type 0, where all impedances have the same value).

2. Defining each of the impedances.

2.10.1 Description of input lines

block heading

CIMPm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NGIMP,NFREQ1,NIMP

Variables	Columns	Default	Format	Description
NGIMP	1-5	-	I5	Number of groups of impedances
NFREQ1	6-10	-	I5	Number of frequencies (will be read in other blocks too)
NIMP	11-15	-	I5	total number of impedances (element-sides with a specified impedance, or coupling procedure)

1.section
repeated lines

IGIMP,IMPPAR

Variables	Columns	Default	Format	Description
IGIMP	1-5	-	I5	Group number
IMPPAR	6-10	-	I5	Group type number (Note that some groups needs specifications on the following line (i.e.impedances))

All groups of impedances must be defined here.

2.section
repeated lines

IEL0,IFL,IGIMP

Variables	Columns	Default	Format	Description
IEL0	1-5	-	I5	Element number
IFL	6-10	-	I5	Element side number
IGIMP	11-15	-	I5	Impedance group. ((Note that some groups needs specifications on several lines (i.e.velocities or parameters for incoming wave))

All impedances must be defined here.

2.10.2 Impedance type 0, constant impedance over a surface

This impedance type offers the possibility to define a constant impedance over a surface (ensemble of sides of element). It can only be used with element type 3 and 11. The impedance must be specified for each frequency.
The contribution to the system-matrices:

$$A_z = j\omega \mathbf{c_a} \tag{2.10-1}$$

where

$$\mathbf{c_a} = \int_{\Gamma_e} \mathbf{N^T} \rho_f A \mathbf{N} d\Gamma \tag{2.10-2}$$

where A is the admittance; A=1/Z, and ρ_f is the fluid density.
Syntax:

1. Define the impedance type using IMPPAR=0
2. Immediately afterwards the normalized specific impedances must be defined. One impedance must be given for each frequency. Only four frequencies at a line.
3. After all impedance are defined, the elementnumber, the elementside (see definition in the chapter 3) and the impedance group must be given.

```
IGIMP,0
Re(z1),Im(z1),Re(z2),Im(z2),Re(z3),Im(z3),Re(z4),Im(z4)
Re(z5),Im(z5),Re(z6),Im(z6),Re(z7),Im(z7),Re(z8),Im(z8)
.               ;define impedance for all frequencies here
.
.
.               ; after definition of impedance-groups, define the sources.
.
IELO,IFL,IGIMP
```

Where `zn` is normalized impedance at frequency no. n.

Example

Ex. Impedances similar to that of a spherical wave of radius a=2 m at the frequencies 50, 100, 150, 200 and 250 Hz $z = \frac{jka}{1+jka}$. Side no.4 of elements 1,2,5,6,7,8,9 and 10 define the circle segment. The values of impedances are:

Frequency	normalized specific impedance
25	0.4606 + j 0.4984
50	0.7735 + j 0.4186
75	0.8848 + j 0.3192
100	0.9318 + j 0.2521
125	0.9552 + j 0.2068

```
CIMP
1,2,8                              ; NGIMP=1,NFREQ1=2,NIMP=8
1,0                                ; IGIMP=0, IMPPAR=0
0.4606,0.4984,0.7735,0.4186,0.8848,0.3192,0.9318,0.2521
0.9552,0.2068                      ; The spec.impedances for the 5 freq.
1,4,1                              ; IEL0=1,IFL=4,IGIMP=1
2,4,1                              ; IEL0=2,IFL=4,IGIMP=1
5,4,1                              ; IEL0=5,IFL=4,IGIMP=1
6,4,1                              ; IEL0=6,IFL=4,IGIMP=1
7,4,1                              ; IEL0=7,IFL=4,IGIMP=1
8,4,1                              ; IEL0=8,IFL=4,IGIMP=1
9,4,1                              ; IEL0=9,IFL=4,IGIMP=1
10,4,1                             ; IEL0=10,IFL=4,IGIMP=1
```

2.10.3 Impedance type 100, impedance of a circular hole in an infinite baffle

This impedance type offers the possibility to end a duct with a circular cross section in an infinite baffle. It can be used for study of ducts or horns (musical or loudspeakers). The theory of this impedance is described in chapter 3 (element type 100). The elements connected to this impedance must be of type 3 (acoustic waves in fluid).
Note that this impedance also needs a special element, type 100. This element has no contribution to the stiffness or mass matrix. [4] [5]
The element is implemented after the theory described in [5]. It uses a modal decomposition of the field in the duct, and matches this to the field in the outer domain. The user must input the number of modes used in the modal decomposition.
It is possible to calculate the pressure in the external domain using VELPAR = 100 and NPEXT > 0 in the DYNM block.
Syntax:

1. Define an element of type 100 in the ELEM-block.
2. Define the impedance type using IMPPAR=100
3. The next line the number of modes in the model, the number of nodes in the hole and the radius of the hole must be given.
4. After all impedance are defined, the elementnumber, the elementside (see definition in the chapter 3) and the impedance group must be given.

```
IGIMP,100
nomodes,nonodes,rad
.
.               ; after definition of impedance-groups, define the sources.
.
IELO,IFL,IGIMP
```

Where nomodes is the number of the modes in the model, nonodes is the number of nodes in the hole (identical to the number of nodes in element 100) and rad is the radius of the hole. Note: one should choose nomodes < nonodes/2.

Example

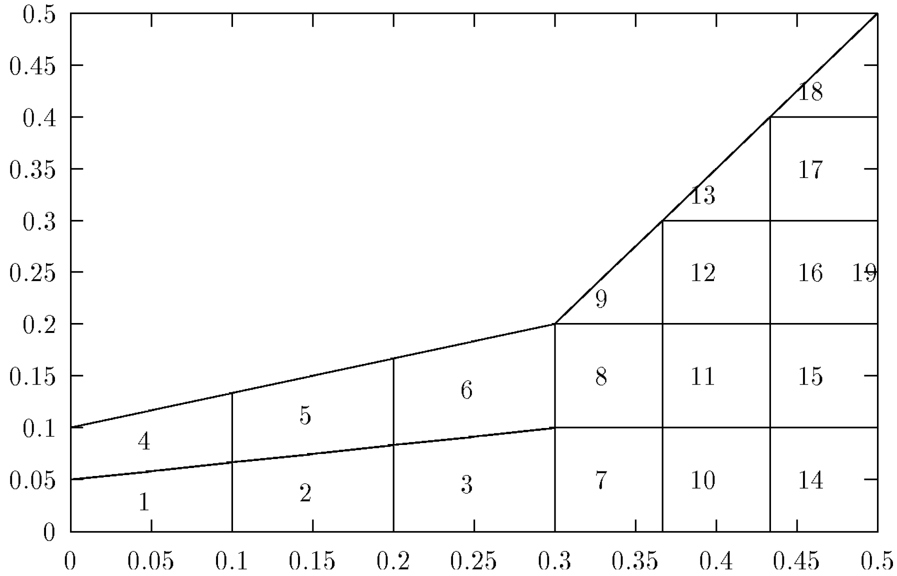
A simple horn with two conical sections in a baffle will be investigated. The throat of the horn has radius .1 m, the inner section is narrow, 36.9°, and 0.3 m long. The outer section is wider, 90°, and 0.3 m long. The mesh is shown below. The full input file is found in horn1.inp. here some details will be shown. Some results can be found in the appendix.

```
ELEM
19,19,0,11,3                    ; NELT=19,NELTR=19,NELTC=0,NNEL=11,NTPE=3
.
.
19,1,1,100,1,1,61,62,63,64,65,66,67,68,69,70    ; special coupling/impedance
71                                              ; element no.100
0
CIMP
```

[4] It is only used for coupling the duct to an infinite half space. In the same manner, it is intended to implement other types of coupling procedures with special elements (i.e. higher order modes in ducts, or the DtN method).

[5] This coupling procedure has given excellent results in studies of horns. However, when used with some complicated meshes generated with PATRAN it has given erroneous results. It is believed that the elements coupled to element of type 100 should have an outwards pointing normal (see definition of elements in chapter 3).

```
1,29,1                         ; NGIMP=1,NFREQ1=29,NIMP=1
1,100                          ; IGIMP=1,IMPPAR=100
3,11,0.5                       ; NOMODES=3, NOMODES=11,RAD=0.5 m
19,1,1                         ; IEL0=19,IFL=1,IGIMP=1
```



2.11 Block SOLC

This block is used to input concentrated loads. It is only used with static analysis. The block is terminated with a line with IG = 0.

2.11.1 Description of input lines

block heading

SOLCm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

Groups of lines, the first line

IG,V

Variables	Columns	Default	Format	Description
IG	1-5	-	I5	Number of group
V	6-75	-	7F10.0	Load values for each d.o.f. V may be longer by additional lines with the format (5X,7F10.0)

the second line

KV

Variables	Columns	Default	Format	Description
KV	1-80	-	16I5	Node numbers with loads must be terminated with zero additional nodes can be given on extra line with format (16I5)

2.11.2 Example

The forces acting on a elastic body is modeled using two sets of consentrated forces in nodes.

1. 10 N downwards, acting on nodes 1, 5 and 6
2. 15 N 45° with respect to the positive x-axis on nodes 25, 28 and 31.

```
SOLC
1,0.0,-10.0          ; 1.group (IG=1) , Fx=0.0 N, Fy=-10.0 N
1,5,6                ; Nodes 1,5,6
2,10.61,10.61        ; 2.group (IG=2) , Fx=10.61 N, Fy=10.61 N
25,28,31             ; Nodes 25,28,31
0                    ; end of block (IG=0)
```

2.12 Block SOLR

This block is to compute and assemble distribute loads (gravitional force) It is only used with static analysis.

2.12.1 Description of input lines

block heading

SOLRm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

2.13 Block DYNM

This block is used for assemblage and solution of dynamical problem. It is used for wave propagation with harmonic time-variation. Complex variables are used in this block.
The system-matrices are collected for maximum execution speed of FEMAK. The frequency-independent matrices (real) are collected first and saved (in core). The frequency- dependent matrices (complex) are collected for each frequency.
For elements with coupling to external fields the field values in external points can be calculated.

2.13.1 Description of input lines

block heading

DYNMm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

NFREQ,PRPAR,VELPAR,INTPAR,NPEXT

Variables	Columns	Default	Format	Description
NFREQ	1-5	-	I5	Number of frequencies
PRPAR	6-10	-	I5	Parameters for level of printing (not implemented)
VELPAR	11-15	-	I5	Parameters for postcalculation of velocity (derivative of d.o.f.) only used when set to the value 100, in order to calculate the velocity in nodes at element type 100, and the pressure in the external points
INTPAR	16-20	-	I5	Parameters for postcalculation of intensity (energy-flux) (not implemented)
NPEXT	21-25	-	I5	Number of external calculation points. (for elements with coupling to external domains)

If NPEXT $\neq$ 0 ,input coordinates as in COOR-block

repeated lines

IN1,X1(1),X1(2),X1(3),IN2,X2(1),X2(2),X2(3),INCR

This set of lines is terminated with $IN1 = 0$.

Variables	Columns	Default	Format	Description
IN1	1-5	-	I5	Number of first node to be generated
X1(1)	6-15	-	F10.0	First node; x coordinate
X1(2)	16-25	-	F10.0	First node; y coordinate
X1(3)	26-35	-	F10.0	First node; z coordinate
IN2	36-40	IN1	I5	Number of last node to be generated
X2(1)	41-50	X1(1)	F10.0	Last node; x coordinate
X2(2)	51-60	X1(2)	F10.0	Last node; y coordinate
X2(3)	61-70	X1(3)	F10.0	Last node; z coordinate
INCR	71-75	1	I5	Pitch of the node numbers

Input frequencies, 1'st line, specification of input-mode

FREQPAR

Variables	Columns	Default	Format	Description
FREQPAR	1-5	-	I5	Describe how frequencies are input FREQPAR = 0, input all frequencies explicitely

Input frequencies

VFREQ

Variables	Columns	Default	Format	Description
VFREQ	1-120	-	10E12.5	Frequencies, 10 on each line

2.13.2 Example

In this case computations are to be done at thirteen frequencies, 100 to 800 Hz in step of 50 Hz. Furthermore an coupling with impedance type 100 is used and the external field is to be computed on a semicircle of radius 10 m, every 5° (19 points).

```
DYNM
13,0,100,0,19                ;NFREQ=13, PRPAR=0, VELPAR=100, INPTPAR=0, NPEXT=19
1,10.5,0.0                   ; point no.1
2,10.46,0.87                 ; point no.2
3,10.35,1.74                 ;    .
4,10.16,2.59                 ;    .
5,9.90,3.42                  ;    .
6,9.56,4.23
7,9.16,5.00
8,8.69,5.74
9,8.16,6.43
10,7.57,7.07
11,6.93,7.66
12,6.24,8.19
13,5.50,8.66
14,4.73,9.06
15,3.92,9.40
16,3.09,9.66
17,2.24,9.85
18,1.37,9.96                 ; point no.18
19,0.50,10.0                 ; point no.19
0                            ; end of input of external points (IN1=0)
0                            ; FREQPAR=0
100.0,150.0,200.0,250.0,300.0,400.0,500.0,550.0,600.0,650.0
700.0,750.0,800.0            ; The frequencies
```

2.14 Block LINM

This block is used for assemblage and solution of linear, static problems. Real variables are used in this block.

2.14.1 Description of input lines

block heading

$\boxed{\text{LINMm}}$

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

1'st line

$\boxed{\text{NRES}}$

Variables	Columns	Default	Format	Description
NRES	1-5	0	I5	Parameter for computation of residues $[K]u - F$ if NRES = 1

2.14.2 Example

Assemblage and solution of problem. Test solution using print of residues.

```
LINM
1                             ; NRES=1 for printing of residue
```

2.15 Block TEMP

This block is used for assemblage and solution of time-dependent problem and it is also used to input the initial condition for the non-prescribed DOF. It is possible to set the value of one or more DOF's. A warning is written in FEMOUT.DAT when there is an attempt on setting already prescribed node in the COND block. This block has been modified from the TEMP block of [1].
The block is terminated with a line with DPAS = 0.0

2.15.1 Description of input lines

block heading

TEMPm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

repeated sets of lines ("outer lines") used for input of initial data

ICOD,V

Variables	Columns	Default	Format	Description
ICOD	1-10	-	10I1	For each DOF. Max.10 2 if prescribed by V 1 if prescribed to zero 0 if non prescribed
V	11-80	-	7F10.0	Values of prescribed DOF's

repeated sets of lines ("inner lines", belong to one ICOD,V line)

IHOW

KV

The sets of lines is terminated with a line with IHOW = 0 The input of initial conditions is terminated with a line with ICOD = 0

execution part (from [1])

DPAS,NPAS,NITER,IMETH,EPSDL,OMEGA

Variables	Columns	Default	Format	Description
IHOW	1	-	I1	Method for choosing the node number 1 : all nodes explicitly written 2 : loop over the node number
KV	1-80	-	16I5	(depending on IHOW) 1 : Nodenumbers (more than one line is possible) 2 : starting, ending and step for the node number
DPAS	1-10	0.2	F10.0	Time step
NPAS	11-15	1	I5	Number of identical time step
NITER	16-20	5	I5	Maximum number of iteration per step
IMETH	21-25	1	I5	Method 1 : Computation of K at each iteration 2 : K constant 3 : computation of K at start of each step
EPSDL	26-35	0.01	F10.0	Admissible error of the norm
OMEGA	36-45	1.0	F10.0	Coefficient of Euler method (α) ($\alpha = 0$:explicit, $\alpha = 1$:implicit)

2.15.2 Example

Ex.1 Node no. 1, 2, 3, ..., 155, 157, 159 and 19 to 152 has it's DOF set to 1.0; Node no.18, 153 and 156, 158, 160 has it's DOF set to 3.2.
First 10 steps with time step equal to 0.01 and `EPSDL = 1E-5` then 10 steps with time step equal to 0.05 and `EPSDL = 1E-3`.
Be careful : no commas for ICOD and also between ICOD and the first value of V.

```
TEMP
20000000001.                ;  ICOD=2000000000, V(1)=1.0 (DOF no.1 is prescribed)
2                           ;  IHOW=2
19,152                      ;  Nodes 19 to 152 are prescribed
1                           ;  IHOW=1
1,2,3,4,5,6,7,8,9,10,11,12,13,14,15,16
17,154,155,157,159          ;  Prescribed nodes
0                           ;  IHOW=0
20000000003.2               ;  ICOD=2000000000, V(1)=3.2 (DOF no.1 is prescribed)
2                           ;  IHOW=2
156,160,2                   ;  Nodes 156,158,160 are prescribed
1                           ;  IHOW=1
18,153                      ;  Nodes 18 and 153 are prescribed
0                           ;  IHOW=0
0                           ;  ICOD=0 (end of input of initial conditions)
1E-2,10,,,1E-5              ;  DPAS=1E-2, NPAS=10, EPSDL=1E-5 (first 10 steps)
5E-2,,,,1E-3                ;  DPAS=5E-2, EPSDL=1E-3          (last 10 steps)
0.0                         ;  DPAS=0.0 (end of block)
```

Ex.2 Node no. 1 to 8 has it's 2^{nd} and 4^{th} DOF set to 1.0 and 2.0 respectively.

```
TEMP
02020000001.,2.             ;  ICOD=0202000000, V(1)=1.0,V(2) (DOF no.2 and4 are prescribed)
2                           ;  IHOW=2
1,8                         ;  Nodes 1 to 8 are prescribed
0                           ;  IHOW=0
0                           ;  ICOD=0 (end of input of initial conditions)
1E-1,5                      ;  DPAS=1E-1, NPAS=5
0.0                         ;  DPAS=0.0 (end of block)
```

2.16 Block VALP

This block is used for assemblage and solution of eigenvalue-problem. Eigenvalues and eigenvectors are calculated using the generalized Jacobi method as described in [1] p.340-347.
May only be used for real, symmetric equations. This means that cases where materials with loss, boundaries with loss or cases with elasto-acoustic coupling may not be solved.

2.16.1 Description of input lines

block heading

VALPm

m is an integer in the range 0-4. m specify the level of output to FEMOUT.DAT. Default value of m is 0.

One line

NVALP,NITER,EPSLB,SHIFT,NSS,NMDIAG,NSWM,TOLJAC

Variables	Columns	Default	Format	Description
NVALP	1-5	1	I5	Number of eigenvalues desired
NITER	6-10	10	I5	Maximum number of iterations
EPSLB	11-20	0.001	F10.0	Admissable error on the eigenvalues
SHIFT	21-30	0.0	F10.0	(Not used)
NSS	31-35	*	I5	Subspace dimension
NMDIAG	36-40	0	I5	(Not used)
NSWM	41-45	12	I5	Maximum number of cycles for the Jacobi method
TOLJAC	46-55	10^{-12}	F10.0	Tolerance of the Jacobi method

* NSS=Min(NVALP+8,2*NVALP)

2.16.2 Example

Find the first 5 eigenvalues. Use default values for the other parameter,

- Max.number of iterations is 10 (NITER=10)
- Admissible error on the eigenvalues is 0.001 (EPSLB=0.001)
- The subspace dimension is calculated by FEMAK (NSS=min(NVALP+8,2·NVALP) = 10)
- Maximum number of cycles for the Jacobi method is 12 (NSWM=12)
- Tolerance of the Jacobi method is 10^{-12} (TOLJAC=10^{-12})

```
VALP
5                   ; NVALP=5
```

2.17 Block STOP

Stop the execution of FEMAK. All records in the input file after this block are neglected by the program.

2.17.1 Description of input lines

block heading

$\boxed{\text{STOP}}$

Chapter 3

Description of elements

In this chapter the different elements and their properties are described. The elements are

ELEM01 Quasiharmonic element. From [1]

- Three versions:
  - 1D (3nodes)
  - 2D (8-nodes)
  - 3D (20 nodes)
- 1 DOF pr node, temperature, velocity potential, etc.
- Real coefficients.

ELEM02 Elastic solid, 2D plane stress and plane strain element. From [1].

- One version:
  - 2D or axisymmetrical (8-nodes)
- 2 DOF pr node; global displacement u_x, u_y.
- Real coefficients.

ELEM03 Acoustic fluid, harmonic waves. This is normally the most used element in the program.

- Two versions:
  - 2D or axisymmetrical (8-nodes)
  - 3D (20 nodes)
- 1 DOF pr node; acoustic pressure, p.
- Real coefficients.

ELEM04 Porous material, harmonic waves, Rayleigh model(equivalent fluid).

- Two versions:
  - 2D or axisymmetrical (8-nodes)
  - 3D (20 nodes)
- 1 DOF pr node; acoustic pressure, p.
- Complex coefficients.

ELEM05 Porous material, harmonic waves, Biot model (elastic frame, viscous and thermal losses).

- Two versions:

  - 2D or axisymmetrical (8-nodes)
  - 3D (20 nodes)
- - 4 DOF pr. node (2D), axisymmetrical; global displacement and weighted relative displacement, u_x, u_y, w_x, w_y
  - 6 DOF pr. node (3D); global displacement and weighted relative displacement, u_x, u_y, u_z, w_x, w_y, w_z
- Complex coefficients.

ELEM08 Plate element (flat plate) - beam .

- Two versions:
  - 2D (3-nodes) (beam)
  - 3D (9 nodes) (plate)
- - 2 DOF pr. node (2D); normal displacement, bending angle.
  - 3 DOF pr. node (3D); normal displacement, bending angle about the two axis perpendicular to the normal.
- Real coefficients.

ELEM11 "Semi-infinite element" (Wave-envelope element).

- Two versions:
  - 2D or axisymmetrical (8-nodes)
  - 3D (20 nodes)
- 1 DOF pr. node; acoustic pressure, p
- Complex coefficients.

ELEM20 Elastic solid, 2D plane strain element with loss. (Rayleigh model)

- One versions:
  - 2D or axisymmetrical (8-nodes)
- 2 DOF pr. node; global displacement u_x, u_y.
- Complex coefficients.

ELEM91 Coupling element, plate - acoustic element (ELEM08-ELEM03),

- Two version:
  - 2D (6 nodes)
  - 3D (17 nodes)
- 2D (6-nodes).
  - 2 DOF pr node for beam ; normal displacement, bending angle.
  - 1 DOF pr node for acoustic fluid; acoustic pressure, p.
- 3D (17-nodes).
  - 3 DOF pr node for plate (9 nodes); normal displacement, bending angle about the two axis perpendicular to the normal.
  - 1 DOF pr node for acoustic fluid (8 nodes); acoustic pressure, p.
- Real coefficients.

ELEM92 Coupling elements, porous material, Biot model - acoustic element (ELEM05-ELEM03)

- One version:
  - 2D or axisymmetrical (6-nodes)
- - 4 DOF pr node for porous material; global displacement and weighted relative displacement, u_x, u_y, w_x, w_y
  - 1 DOF pr node for acoustic fluid; acoustic pressure, p.
- Real coefficients.

ELEM93 Coupling elements, elastic solid - acoustic element (ELEM02,ELEM20-ELEM03)

- One version:
  - 2D or axisymmetrical (6-nodes)
- - 2 DOF pr node for elastic solid ; global displacement, u_x, u_y
  - 1 DOF pr node for acoustic nodes; acoustic pressure.
- Real coefficients.

ELEM94 Coupling elements, porous material, Biot model - elastic solid (ELEM05-ELEM20)

- One version:
  - 2D or axisymmetrical (6-nodes)
- - 4 DOF pr node for porous material; global displacement and weighted relative displacement, u_x, u_y, w_x, w_y
  - 2 DOF pr node for elastic solid; global displacement, u_x, u_y
- Real coefficients.

ELEM95 Coupling elements, porous material, Biot model - plate/beam (ELEM05-ELEM08)

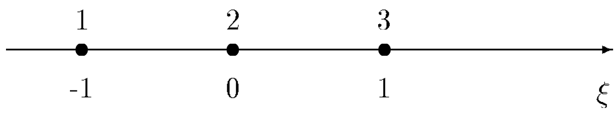
- Two version:
  - 2D or axisymmetrical (6-nodes)
  - 3D (17 nodes)
- 2D (6-nodes).
  - 4 DOF pr node for porous material; global displacement and weighted relative displacement, u_x, u_y, w_x, w_y
  - 2 DOF pr node for beam; normal displacement, w_n, and bending angle, θ
- 3D (17-nodes).
  - 6 DOF pr node for porous material (8 nodes); global displacement and weighted relative displacement, u_x, u_y, u_z, w_x, w_y, w_z
  - 3 DOF pr node for plate (9 nodes); normal displacement, bending angle about the two axis perpendicular to the normal.
- Real coefficients.

IMP100 Coupling procedure for axisymmetrical domain, ending in a circular hole in an infinite baffle.

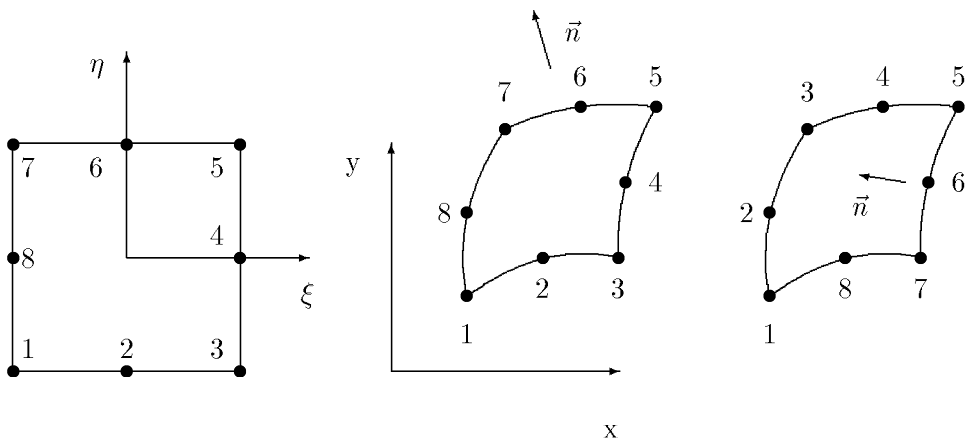
- One version:
  - axisymmetrical (The user must define the number of nodes)
- 1 DOF pr node; acoustic pressure, p.
- Complex coefficients.

3.1 ELEM01 - Quasiharmonic elements

This element is used for anisotropic quasiharmonic problems. It is described in [1]. The element is isoparametric. It can be used for different types of problems. See theory section for description of governing equation.



Reference element. Nodenumbering 1 dimensional version



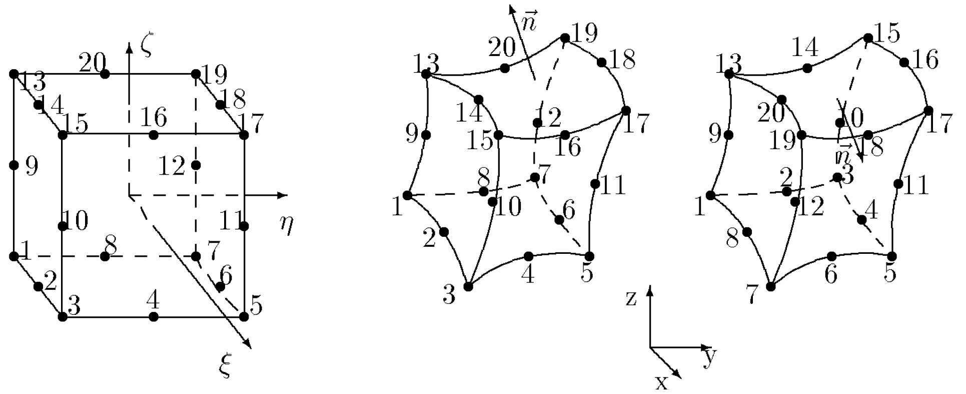
Reference element. Nodenumbering 2 dimensional version. Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)



Reference element. Nodenumbering 3 dimensional version Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 1, 9, 13, 20, 19, 12, 7, 8)

Side 3 ξ = 1, (node 3, 4, 5, 11, 17, 16, 15, 10)

Side 4 η = -1, (node 1, 2, 3, 10, 15, 14, 13, 9)

Side 5 η = 1, (node 5, 6, 7, 12, 19, 18, 17, 11)

Side 6 ζ = -1, (node 1, 8, 7, 6, 5, 4, 3, 2)

Side 7 ζ = 1, (node 13, 14, 15, 16, 17, 18, 19, 20)

The element has one DOF for each node.

Dimension	DOF	Comment
1D, 2D or 3D	ϕ	variable of quasiharmonic equation i.e. temperature, velocity potential

The element matrices are symmetric and have real coeffients.

3.1.1 ELEM01 - Material parameters

The material is described by four coefficients. The anisotropy is described using the first three coefficients. For time varying problems a fourth coefficient is added.
For laminar fluid flow (steady state) (unknown - fluid potential):

1 d_x - permeability in x-direction

2 d_x - permeability in y-direction

3 d_x - permeability in z-direction

4 not used

For heat transfer (unknown - temperature):

1 d_x - thermal conductivity in x-direction

2 d_x - thermal conductivity in y-direction

3 d_x - thermal conductivity in z-direction

4 ρc - thermal capacity

3.1.2 ELEM01 - Available source options

Nodal sources can be introduced using the SOLC-block.

3.1.3 ELEM01 - Available impedance options

None

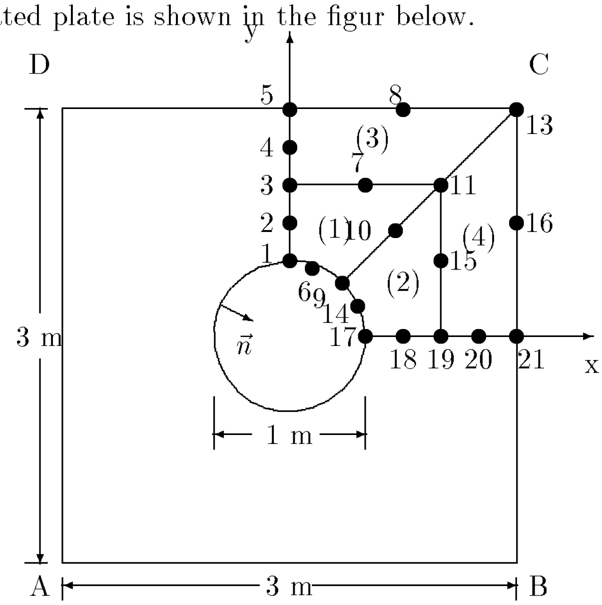
3.1.4 ELEM01 - Use of the element

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties. (PREL)
3. Define the prescribed DOF's. (COND)
4. Define nodal connectivity of element. (ELEM)
5. Define sources/loads. (SOLC)
6. Define initial conditions (TEMP).
7. Assemble and solve equations. (LINM or TEMP)

The points 1, 2, 3, 4, and 7 are mandatory. Usually all points are included in the definition of a modell.

Example: Heat conduction in perforated plate

This example on heat conduction in a perforated plate can be found in [1], p.447-480. There are minor differences both because FEMAK is sligthly modified from MEF in [1], and because I think there are some errors in that text considering the loads (sources). The perforated plate is shown in the figur below.



Geometry and mesh of heat transfer problem

This example is 2-dimensional. The material is concrete, it is isotropic with the thermal conductivity $d = d_x = d_y = 1.4$ w/(m°C) . The boundary conditions are:

- u=0, on AD and BC (specified temperature). (COND - block)
- $\frac{\partial u}{\partial n} = \frac{\partial u}{\partial y} = 0$ on AB and CD (insulated edges) (Natural boundary condition).
- $d\frac{\partial u}{\partial n} = 1$, Specified heat flux on the inside circle (SOLC block).

The model use a quarter of the plate because of the symmetry in the problem. The heat flux is applied by concentrated nodal loads:

- $\pi/48$ at nodes 1 and 17

- $\pi/24$ at nodes 9
- $\pi/12$ at nodes 6 and 14

```
COMT
  Bigex3.inp 3/9-1995, Tonni F. Johansen
  Example from Dhatt & Touzot p.449 ...
  To check elem01, Steady state case.

COOR
21,1,2,0.5,0.5
1,0.0,1.0,0.0,5,0.0,3.0,0.0,1
6,0.3827,0.9239,0.0,8,1.5,3.0,0.0,1
9,0.707,0.707,0.0,13,3.0,3.0,0.0,1
14,0.9239,0.3827,0.0,16,3.0,1.5,0.0,1
17,1.0,0.0,0.0,21,3.0,0.0,0.0,1
0
COND
1
1
13,16,21
0
0
PREL
1,4
1,1.4,1.4,1.4,2.03e6
0
ELEM
4,4,0,8,1,0
1,2,8,1,1,1,1,6,9,10,11,7,3,2
3,2,8,1,1,1,3,7,11,12,13,8,5,4
0
SOLC
1,0.06545
1,17
2,0.1309
9
3,0.2618
6,14
0
LINM
1
STOP
```

The results of this example, together with a transient analysis, is shown in the appendix.

3.1.5 ELEM01 - Theory of element

The governing equation of this element is:

$$\frac{\partial}{\partial x}(d_x\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(d_y\frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(d_z\frac{\partial u}{\partial z}) + f_V = \rho c\frac{\partial u}{\partial t} \tag{3.1-1}$$

With boundary conditions:

$$u = u_0,\ \forall\ x \in \Gamma_d \tag{3.1-2}$$

$$\frac{\partial u}{\partial n} = h_0,\ \forall\ x \in \Gamma_n \tag{3.1-3}$$

Where usually $f_V = 0$ or $\rho c\frac{\partial u}{\partial t} = 0$.
Using galerkin's method one obtain:

$$[\mathbf{k}]\mathbf{u} - [\mathbf{m}]\dot{\mathbf{u}} = \mathbf{f} + \mathbf{s}\ \overline{\mathbf{h_0}} \tag{3.1-4}$$

where

$$\mathbf{k} = \int_{\Omega_e} \mathbf{B^T D B} d\Omega \tag{3.1-5}$$

$$\mathbf{m} = \rho c \int_{\Omega_e} \mathbf{N^T N} d\Omega \tag{3.1-6}$$

$$\mathbf{s} = \int_{\Gamma_e} \mathbf{N^T} d_x n_x \mathbf{N} d\Omega \tag{3.1-7}$$

$$\mathbf{f} = \int_{\Gamma_e} \mathbf{N^T} \mathbf{f_V} d\Omega \tag{3.1-8}$$

Ω_e and Γ_e indicates integrations domains in one element.

$$\mathbf{B} = \left\{ \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{array} \right\} [N_1, N_2, ...] \tag{3.1-9}$$

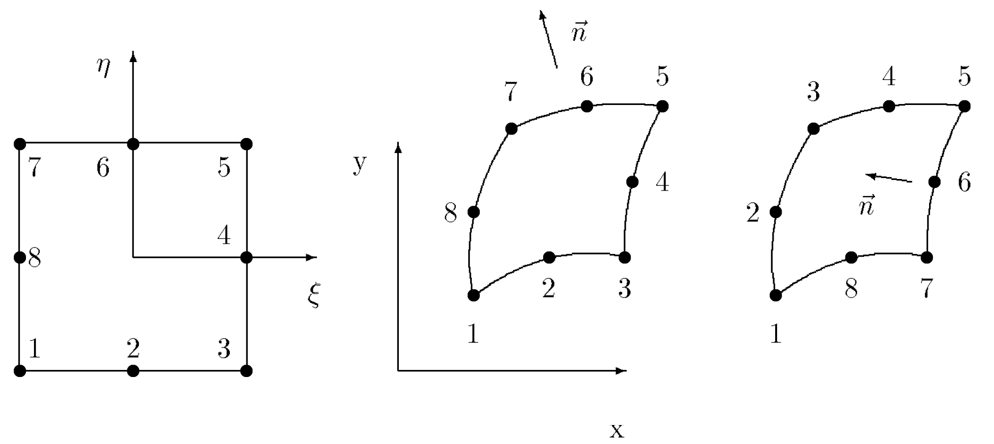
$$\mathbf{D} = \left\{ \begin{array}{ccc} d_x & 0 & 0 \\ 0 & d_y & 0 \\ 0 & 0 & d_z \end{array} \right\} \tag{3.1-10}$$

List of examples in appendix.

1. genex3. Heat conduction in plate, steady state.
2. genex4. Heat conduction in plate, transient state.

3.2 ELEM02 - Elastic solid

This element is used for isotropic elastic solids. It is described in [1]. It is 2 dimensional. It can be used for wave propagation in elastic solids.
Second order shape functions are used. 8 nodes for the 2-dimensional case.
The material is described using Young's modulus, Poisson's coefficient and the specific mass. It can handle both plane stress and plane strain.



Reference element. Nodenumbering 2 dimensional version. Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)

The element has two DOF's for each node, the global displacements in two dimensions.

Dimension	DOF	Comment
2D	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
2D axisym.	u_z	Global displacement z-axis
	u_r	Global displacement r-axis

The element matrices are symmetric and have real coeffients. It may be coupled to other media.

- Fluid (ELEM03). Must use coupling element (ELEM93).

3.2.1 ELEM02 - Material parameters

The fluid is described by only four parameters.

1. Young's modulus
2. Poisson's coefficient
3. A parameter selecting plane stress or plain strain.
4. Specific mass

3.2.2 ELEM02 - Available source options

- Nodal forces/loads can be introduced using the SOLC-block.
- Distributed forces can be introduced using the SOLR-block.

3.2.3 ELEM02 - Available impedance options

None

3.2.4 ELEM02 - Use of the element

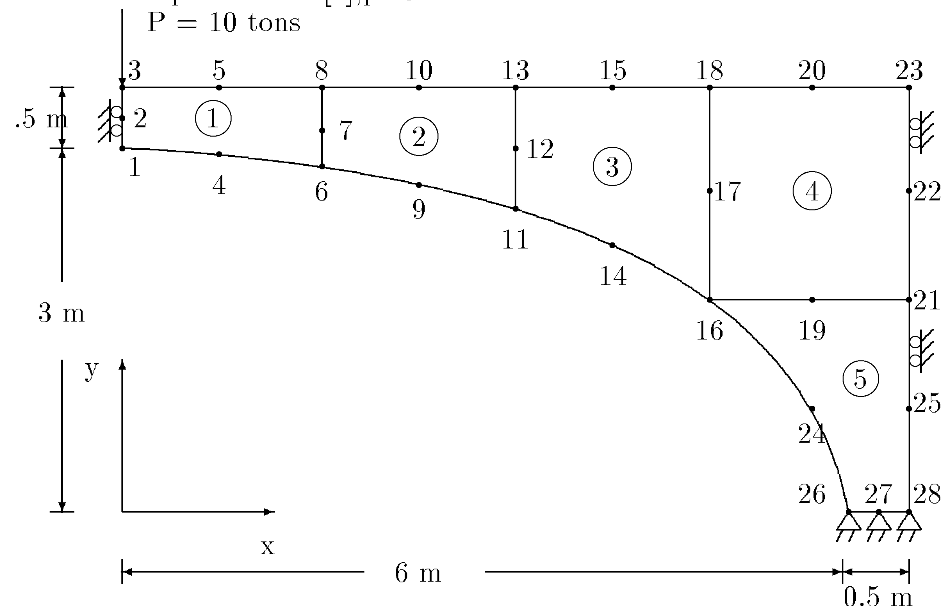
In this description no coupling to other media is involved. For coupled problems look in the sections for coupling elements (ELEM93).
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of material. (PREL)
3. Define the prescribe DOF's. *i.e.* the known displacements.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Define forces/loads (SOLR,SOLC)
6. Assemble and solve
   - Eigenvalueproblem. (VALP)
   - Dynamical problem (DYNM)
   - Static problem (LINM)

The points 1, 2, 3, 4, and 6 are mandatory. Usually all points are included in the definition of a modell.

Example: Elliptical concrete arch

For the example see also [1],p.481.



Material constants:

E - $0.2 \cdot 10^7$ tons/m^2

ν - 0.3

ρ - 2.3 tons/m^3

thickness - 1 m

The model use two types of loads, a distributed dead weight (using the SOLR-block), and a consentrated force at a node (using the SOLC-block). Here the input-file for static analysis is shown. In the appendix this file and its eigenvalue problem is discussed briefly.

```
COMT
  Bigex2.inp 16/9-1995, Tonni F. Johansen
  Example from Dhatt & Touzot p.481 ...
  example on use of elem02 and elem20

COOR
28,2,20
3,0.0,3.5,0.0,23,6.5,3.5,0.0,5
5,0.75,3.5,0.0,20,5.75,3.5,0.0,5
2,0.0,3.25
7,1.625,3.194
12,3.25,3.0109
17,4.8750,2.6245
19,5.6875,1.75
1,0.0,3.0
4,0.8125,2.9724
6,1.625,2.8879
9,2.4375,2.7413
11,3.25,2.5218
14,4.0625,2.2077
16,4.8750,1.7489
24,5.75,0.86
26,6.0,0.0
27,6.25,0.0
28,6.5,0.0
25,6.5,0.87
21,6.5,1.75
22,6.5,2.62
0
COND
11
1
26,27,28
0
10
1
1,2,3,25,21,22,23
0
0
PREL
1,4
1,2.0e6,0.3,0.0,2.3
0
ELEM
5,5,0,8,2
1,4,5,0,1,0,1,4,6,7,8,5,3,2
5,1,0,0,1,0,26,27,28,25,21,19,16,24
0
SOLC
1,0.0,-10.0
3
0
```

```
SOLR
LINM
1
STOP
```

3.2.5 ELEM02 - Theory of element

The stress-strain relation for the elastic material is:

$$\sigma = \mathbf{D}\epsilon = \mathbf{DLu} \tag{3.2-1}$$
$$\epsilon = \mathbf{Lu} \tag{3.2-2}$$

Where σ is the stress tensor, $\mathbf{D}$ is the elasticity matrix, $\mathbf{u}$ is the displacement of the solid, and ϵ is the strain tensor.

$$\mathbf{L}^T\sigma + \omega^2\rho\mathbf{u} = \mathbf{0} \tag{3.2-3}$$
$$\mathbf{L}^T\mathbf{DLu} + \omega^2\rho\mathbf{u} = \mathbf{0} \tag{3.2-4}$$

When introducing FEM discretization, $\mathbf{u} = \mathbf{N}\ \bar{u}$, using Galerkin's method and Green's theorem:

$$\begin{aligned}&\int_\Omega \mathbf{L^T N^T D L N}\, d\Omega\, \overline{\mathbf{u}} - \omega^2 \int_\Omega \rho \mathbf{N^T N}\, d\Omega\, \overline{\mathbf{u}} \\ &+ \int_\Omega \mathbf{N^T} \underline{\mathbf{f_V}}\, d\Omega + \int_{\Gamma_N^s} \mathbf{N^T}(\sigma - \underline{\sigma})\, d\Gamma = 0\end{aligned} \tag{3.2-5}$$

$$\left[\mathbf{k} - \omega^2\mathbf{m}\right]\bar{\mathbf{u}} = \mathbf{f}$$

$\underline{\sigma}$ is prescribed stress and $\underline{\mathbf{f_V}}$ distributed forces (gravity).

Stiffness, mass and damping matrices

$$\mathbf{k} = \int_{\Omega_b} \mathbf{B^T D B}\, d\Omega \tag{3.2-6}$$

$$\mathbf{m} = \int_{\Omega_b} \mathbf{N^T}\rho\mathbf{N}\, d\Omega \tag{3.2-7}$$

$$\mathbf{B} = \mathbf{LN} \tag{3.2-8}$$

Source terms:

$$\mathbf{f} = -\int_\Omega \mathbf{N^T}\underline{\mathbf{f_V}}\, d\Omega + \int_{\Gamma_N^s} \mathbf{N^T}\underline{\sigma}\, d\Gamma \tag{3.2-9}$$

The stress-strain relationships used in this element are:
2-dimensional, plane strain:

$$\mathbf{D} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

2-dimensional, plane stress:

$$\mathbf{D} = \frac{E}{(1+\nu)(1-\nu)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x_1} & 0 \\ 0 & \frac{\partial}{\partial x_2} \\ \frac{\partial}{\partial x_2} & \frac{\partial}{\partial x_1} \end{bmatrix}$$

$\mathbf{u}^T = [u_1 u_2]$, $\sigma^T = [\sigma_{11}, \sigma_{22}, \sigma_{12}]$,
See e.g. [6], ch.4-6

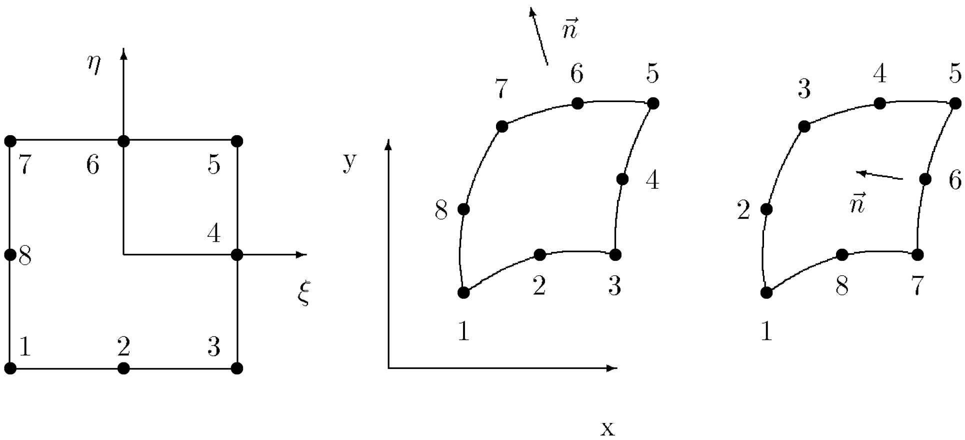
List of examples in appendix.

1. elaex11 - Eigenvalue elastic material.

2. bigex2 - Plane stress, static case.

3.3 ELEM03 - acoustic fluid

This element is used for acoustic waves in a fluid. It is used for solution of the wave equation, typically with harmonic time variation assumed, in other words, the Helmholtz equation is the governing equation. 2-dimensional, 3-dimensional and axisymmetrical problems may be solved. Second order shape functions are used. 8 nodes for the 2-dimensional and axisymmetrical case and 20 nodes for the 3-dimensional case. See figures below.
The fluid is described by its density and its speed of sound.



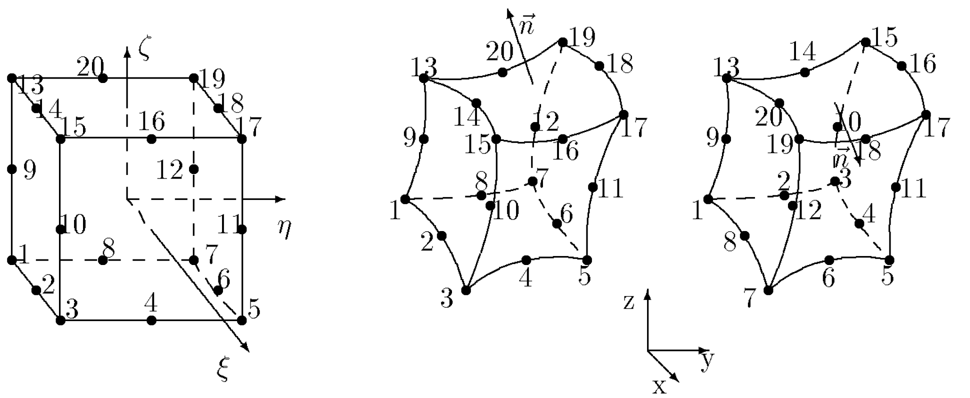
Reference element. Nodenumbering 2 dimensional version. Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)



Reference element. Nodenumbering 3 dimensional version Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 1, 9, 13, 20, 19, 12, 7, 8)

Side 3 ξ = 1, (node 3, 4, 5, 11, 17, 16, 15, 10)

Side 4 η = -1, (node 1, 2, 3, 10, 15, 14, 13, 9)

Side 5 η = 1, (node 5, 6, 7, 12, 19, 18, 17, 11)

Side 6 ζ = -1, (node 1, 8, 7, 6, 5, 4, 3, 2)

Side 7 ζ = 1, (node 13, 14, 15, 16, 17, 18, 19, 20)

The element has only one DOF for each node, the acoustic pressure.

Dimension	DOF	Comment
2D, 2D axisym., 3D	p	acoustic pressure

It can be given various boundary conditions: specified pressure, specified velocity or impedance. The normal of the element will point out of the element if it is numbered globally as shown in the figure.
The element matrices are symmetric and have real coefficients. It may be coupled to other type of elements

- Infinite domain, described using wave-envelope element (ELEM11) directly without any coupling element.
- Porous media, described as an equivalent fluid (ELEM04) directly without any coupling element.
- Porous media, described using Biot theory, (ELEM05). Must use coupling element (ELEM92).
- Plate (ELEM08). Must use coupling element (ELEM91).
- Elastic solid, without loss (ELEM02). Must use coupling element (ELEM93).
- Elastic solid, with loss (ELEM20). Must use coupling element (ELEM93).

3.3.1 ELEM03 - Material parameters

The fluid is described by only to parameters.

1. Density of the fluid.
2. Speed of sound in the fluid.

3.3.2 ELEM03 - Available source options

Typical sources are incoming waves, point sources or vibrating surfaces.

source type 1 - Volume source in a node

source type 20,21 - Specified velocities.

source type 50 - Incoming plane wave.

souce type 55 - Incoming plane wave (used with ELEM11)

Note that the normal always is assumed to point out of the acoustic domain. The user must support the correct definition for the normal where this definition is critical (source type 50 and 55).

3.3.3 ELEM03 - Available impedance options

Only one impedance type is available, type 0. Constant impedance for a surface; specified explicitely for each frequency.

3.3.4 ELEM03 - Use of the element

Here the use of the element for modelling pure acoustic problems is described (Helmholtz equation). No coupling to other media is involved. For coupled problems look in the sections for coupling elements (ELEM91, ELEM92, ELEM93) or the sections for the elements that do not need coupling elements (ELEM04, ELEM11).
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the acoustic fluid (fluids). (PREL)
3. Define the prescribe DOF's. *i.e.* known pressures.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Define velocities at boundaries or other types of sources. (SRCE)
6. Define impedances (CIMP)
7. Assemble and solve
   - Eigenvalueproblem. (VALP)
   - Dynamical problem (DYNM)

The points 1, 2, 3, 4, and 7 are mandatory. Usually all points are included in the definition of a modell.

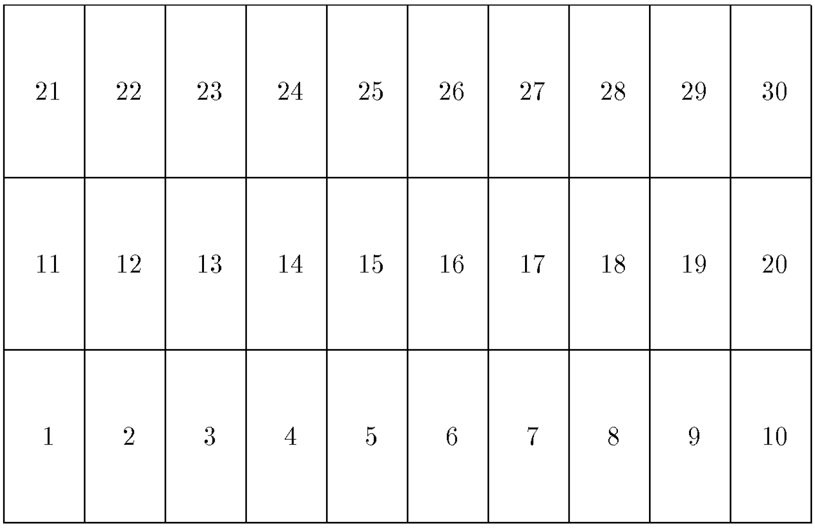
Example: Straight tube with vibrating surface and impedance

This is a simple example on waves in a straight tube with circular cross-section. The geometry is therefore axisymmetric, and the acoustic field is also assumed to be axisymmetric. The axisymmetrical option in the FEMIN (NDIM = 20 in the COOR-block) is used.
The geometry and the mesh are shown below.
The source is a vibrating boundary at z=0.0. The tube is terminated by an impedance in z=1.0 m. The impedance is normalized to ρc and is set: Z=1.0+j
The input file will be:

```
COMT
  acouex1.inp      March 9. 1995 T.F.Johansen
  Example no.1 of use of FEMAK for pure acoustic problem.
  Straight tube, 1.0 m long and radius 0.3 m.
  Velocity 1.0 m/s in z=0.0m and impedance z=1+j in z=1.0m

COOR
117,1,20
1,0.00,0.0,0.0,7,0.00,0.3,0.0,1
8,0.05,0.0,0.0,11,0.05,0.3,0.0,1
12,0.10,0.0,0.0,18,0.10,0.3,0.0,1
19,0.15,0.0,0.0,22,0.15,0.3,0.0,1
23,0.20,0.0,0.0,29,0.20,0.3,0.0,1
30,0.25,0.0,0.0,33,0.25,0.3,0.0,1
34,0.30,0.0,0.0,40,0.30,0.3,0.0,1
41,0.35,0.0,0.0,44,0.35,0.3,0.0,1
45,0.40,0.0,0.0,51,0.40,0.3,0.0,1
52,0.45,0.0,0.0,55,0.45,0.3,0.0,1
```

```
56,0.50,0.0,0.0,62,0.50,0.3,0.0,1
63,0.55,0.0,0.0,66,0.55,0.3,0.0,1
67,0.60,0.0,0.0,73,0.60,0.3,0.0,1
74,0.65,0.0,0.0,77,0.65,0.3,0.0,1
78,0.70,0.0,0.0,84,0.70,0.3,0.0,1
85,0.75,0.0,0.0,88,0.75,0.3,0.0,1
89,0.80,0.0,0.0,95,0.80,0.3,0.0,1
96,0.85,0.0,0.0,99,0.85,0.3,0.0,1
100,0.90,0.0,0.0,106,0.90,0.3,0.0,1
107,0.95,0.0,0.0,110,0.95,0.3,0.0,1
111,1.00,0.0,0.0,117,1.00,0.3,0.0,1
0
PREL
1,2
1,1.205,343.4
0
COND
0
ELEM
30,30,0,8,3
1,10,11,3,1,1,1,8,12,13,14,9,3,2
11,10,11,3,1,1,3,9,14,15,16,10,5,4
21,10,11,3,1,1,5,10,16,17,18,11,7,6
0
SRCE
1,10,3
1,21
1,2,1
-1.0,0.0,-1.0,0.0,-1.0,0.0
11,2,1
-1.0,0.0,-1.0,0.0,-1.0,0.0
21,2,1
-1.0,0.0,-1.0,0.0,-1.0,0.0
CIMP
1,10,3
1,0
1.0,1.0,1.0,1.0,1.0,1.0,1.0,1.0
1.0,1.0,1.0,1.0,1.0,1.0,1.0,1.0
1.0,1.0,1.0,1.0
10,3,1
20,3,1
30,3,1
DYNM
10
0
50.,100.,150.,200.,250.,300.,350.,400.,450.,500.,550.,600.,650.,700.,750.,800.,850.,900.,950.,1000.
STOP
```



3.3.5 ELEM03 - Theory of element

For the fluid we have Newtons 2^{nd} law and the equation of continuity of volume flow:

$$-\nabla p = j\rho\omega\mathbf{v} \tag{3.3-1}$$

$$j\omega p = -K\nabla\cdot\mathbf{v} \tag{3.3-2}$$

This element is used to solve the wave equation for time-harmonic waves (Helmholtz equation).

$$\nabla^2 p - k^2 p = 0,\ \forall\ x \in \Omega \tag{3.3-3}$$

With boundary conditions:

$$p = p_0,\ \forall\ x \in \Gamma_d \tag{3.3-4}$$

$$\frac{\partial p}{\partial n} = h_0,\ \forall\ x \in \Gamma_n \tag{3.3-5}$$

$$\frac{\partial p}{\partial n} - \alpha p = 0,\ \forall\ x \in \Gamma_m \tag{3.3-6}$$

Here p is the acoustic pressure, $k = \frac{\omega}{c}$ is the wavenumber, ω is the angular frequency. $\frac{\partial}{\partial n}$ is the normal gradient
This equations may be handled as shown i.e. in [7], and the appropriate FEM equations are:

$$\int_\Omega \nabla\mathbf{N}^\mathbf{T}\nabla^\mathbf{T}\mathbf{N}d\Omega\,\overline{\mathbf{p}} - \omega^2\int_\Omega \mathbf{N}^\mathbf{T}\frac{1}{c^2}\mathbf{N}d\Omega\,\overline{\mathbf{p}} + j\omega\int_\Gamma \mathbf{N}^\mathbf{T}\rho A\mathbf{N}d\Gamma\,\overline{\mathbf{p}} = -j\omega\int_\Gamma \mathbf{N}^\mathbf{T}\rho\mathbf{N}d\Gamma\,\overline{\mathbf{v}} \tag{3.3-7}$$

$$\left[\mathbf{k_a} - \omega^2\mathbf{m_a} + j\omega\mathbf{c_a}\right][\overline{\mathbf{p}}] = -j\omega\mathbf{s_a}\overline{\mathbf{v}} \tag{3.3-8}$$

where

$$\mathbf{k_a} = \int_{\Omega_e} \mathbf{B_a^T}\mathbf{B_a}d\Omega \tag{3.3-9}$$

$$\mathbf{m_a} = \frac{1}{c^2}\int_{\Omega_e} \mathbf{N}^\mathbf{T}\mathbf{N}d\Omega \tag{3.3-10}$$

$$\mathbf{c_a} = \int_{\Gamma_e} \mathbf{N^T} \rho A \mathbf{N} d\Gamma \tag{3.3-11}$$

$$\mathbf{s_a} = \int_{\Gamma_e} \mathbf{N^T} \rho \mathbf{N} d\Omega \tag{3.3-12}$$

Ω_e and Γ_e indicates integrations domains in one element.

$$\mathbf{B_a} = \left\{ \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{array} \right\} [N_1, N_2, ...] \tag{3.3-13}$$

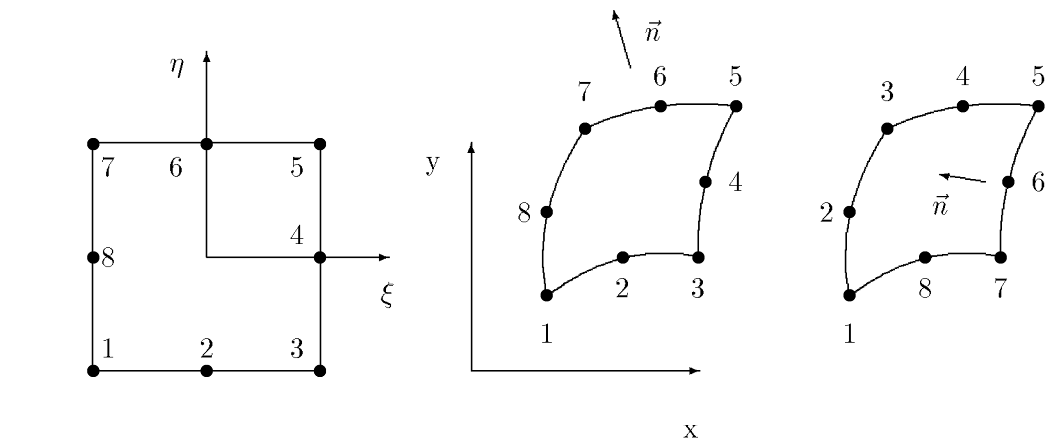
Where ρ is the fluid density, v is particle velocity, $\mathbf{A} = v/p$ is the admittance.
See e.g. [7] or [3] for details.

List of examples in appendix.

1. acouex1 - Simple infile
2. acouex3 - Transmission loss
3. acouex21 - Radiation impedance
4. stmpwee1, stmpwee5, stmpwee7 - Radiation impedance using wave envelope elements
5. horn1 - Horn loudspeaker
6. porex1 - Absorbent material in Kundt's tube.
7. layer1 - Layered material, aluminum, elastic porous material.

3.4 ELEM04 - Porous material, equivalent fluid

This element is used for acoustic waves in porous material. It is assumed that the frame is rigid. Then it is possible to model the propagation using Helmholtz equation with an equivalent fluid model. The viscous and thermal losses are included in the equivalent density and bulk modulus.
Different models are used for the equivalent fluids.



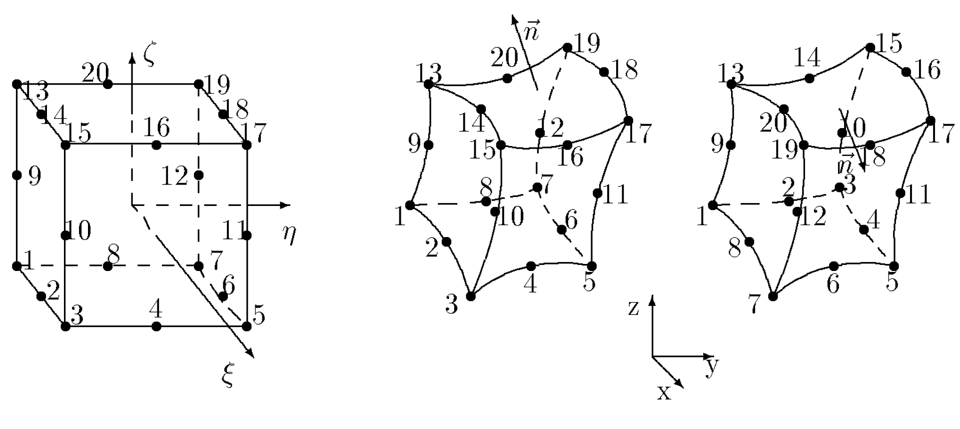
Reference element. Nodenumbering 2 dimensional version Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)



Reference element. Nodenumbering 3 dimensional version Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 1, 9, 13, 20, 19, 12, 7, 8)

Side 3 ξ = 1, (node 3, 4, 5, 11, 17, 16, 15, 10)

Side 4 η = -1, (node 1, 2, 3, 10, 15, 14, 13, 9)

Side 5 η = 1, (node 5, 6, 7, 12, 19, 18, 17, 11)

Side 6 ζ = -1, (node 1, 8, 7, 6, 5, 4, 3, 2)

Side 7 ζ = 1, (node 13, 14, 15, 16, 17, 18, 19, 20)

The element has only one DOF for each node, the pore pressure.

Dimension	DOF	Comment
2D, 2D axisym., 3D	p	acoustic (pore) pressure

The element matrices are symmetric and have complex coefficients.
It may be coupled to other type of elements:

- Acoustic fluid ELEM03

3.4.1 ELEM04 - Material parameters

The equivalent fluid can be described using different models. Only three are implemented yet. The first material parameter defines the material description type.

1 Identify model of equivalent fluid.

- **1** "Craggs' (Zwicker,Kosten)" model. Assuming no thermal losses (adiabatic compression) and frequency independent viscous losses.
- **2** Johnson-Allard. Using characteristic lengths to describe viscous and thermal losses in the porous material.
- **10** Old implementation of "Craggs(Zwicker,Kosten)" model. This implementation is directly after Craggs' articles, but they have some errors. Only for historical reasons.

other parameters If "Craggs'(Zwicker,Kosten) model" (parameter 1 = 1.0) is used:

- **2** ρ - Density of pore fluid.
- **3** c - Wave velocity of the fluid in the pores.
- **4** Φ - Flow resistivity.
- **5** ϕ - porosity
- **6** k_s - Structure factor.

other parameters If "Johnson-Allard's model" (parameter 1 = 2.0) is used:

- **2** ρ - Density of pore fluid.
- **3** c - Wave velocity of the fluid in the pores.
- **4** B^2 - Prandtl's number of pore fluid.
- **5** η -Viscosity of pore fluid
- **6** $\gamma = c_p/c_v$ of pore fluid
- **7** ϕ - porosity
- **8** k_s - Structure factor.
- **9** Φ Flow resistivity.

10 Λ Characteristic length for viscous losses

11 Λ' Characteristic length for thermal losses

other parameters If old implementations of “Craggs' (Zwicker,Kosten) model” (parameter 1 = 10.0) is used:

2 ρ - Density of pore fluid.

3 c - Wave velocity of the fluid in the pores.

4 Φ - Flow resistivity.

5 ϕ - porosity

6 k_s - Structure factor.

3.4.2 ELEM04 - Available source options

None

3.4.3 ELEM04 - Available impedance options

None

3.4.4 ELEM04 - Use of the element

This element is most often used with other elements, primarely ELEM03. The use of this element is similar to that of ELEM03.
A description of typical use of the element with ELEM04 is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the fluids. (PREL)
3. Define the prescribe DOF's. *i.e.* known pressure.(COND)
4. Define nodal connectivity of element.(ELEM)
5. Assemble and solve equations. (DYNM)

All the points are mandatory.

Example: Kundt's tube

Here some details of an example is shown. The example is carried out in more detail the appendix. The example is a calculation on a Kundt's tube. The reason for the choice is the possibility to compare with analyttical models (plane wave assumptions).
The geometry is shown below. The tube is axisymmetrical, radius is 0.1 m, the air- column is 0.30 m, while the thickness of the porous material is 0.1 m. Material parameter of the porous material is, Craggs'(Zwicker,Kosten) model (VPREE(1)=1.0), flow resisitivy: $\Phi = 10000 Ns/m^4$ (VPREE(4)=10000), porosity: $\phi = 0.9$ (VPREE(5)=0.9), structure factor: $k_s = 1.5$ (VPREE(6)=1.5).
Here only the material definition block of the input file is shown:

```
.
.
PREL                                    ; Definition of material properties
2,6
1,1.205,343.4                           ; material 1 - air
2,1.0,1.205,343.4,10000.0,0.90,1.5      ; material 2 - porous material
0
.
.
```

19	20	21	22	23	24	37	38	39	40
13	14	15	16	17	18	33	34	35	36
7	8	9	10	11	12	29	30	31	32
1	2	3	4	5	6	25	26	27	28

Elements number 1 to 24 are of type 3 (fluid), while elements 25 to 40 are of type 4 (porous material).

3.4.5 ELEM04 - Theory of element

As mentioned before, the wave propagation in a material with rigid frame may be described with an equivalent fluid. This means that the viscous and thermal losses are modelled using a fluid with complex density and complex bulk modulus. When this is done the wave propagation may be described using Helmholtz equation. In this section the theory of such waves will not be described. See e.g. Allard [8].

$$\nabla p - k_{eq}^2 p = 0,\ \forall\ x \in \Omega \tag{3.4-1}$$

With boundary conditions:

$$p = p_0,\ \forall\ x \in \Gamma_d \tag{3.4-2}$$

$$\frac{\partial p}{\partial n} = h_0,\ \forall\ x \in \Gamma_n \tag{3.4-3}$$

$$\frac{\partial p}{\partial n} - \alpha p = 0,\ \forall\ x \in \Gamma_m \tag{3.4-4}$$

Note that the wavenumber k_{eq} is complex.
There exists different models for the equivalent fluids. Two of the more common are:

Rayleigh model The pores are assumed to be straight, parallell tubes. The tubes may be tilted compared to the waves' direction of propagtion.

Johnson model The pores are assumed to have a more random organisation. The geometry of the pores are described using "characteristic lengths".

Here the two models implemented will be described. Note that the notation may have been changed from the references.

Craggs' model

This model is based on Craggs' papers [9], [10]. These articles describe well how to implement an finite element model of an equivalent fluid model, ensuring continuity of volume flow of fluid between fluid and porous material. It uses Rayleigh model for the porous material.

Note We believe that it is an error in the description of the physical model in these articles, this has been changed in FEMAK. (It concerns the use of the effective density in the model, it will be described later on). It is also an error in the scaling of the matrices which is done to ensure continuity of volume velocity between a porous material and its surrounding fluid.
Newtons law and continuity of volume flow:

$$-\nabla p = (j\frac{\omega\rho k_s}{\phi} + \Phi)\mathbf{u} \tag{3.4-5}$$

$$-\frac{j\omega\phi}{\rho c^2}p = \nabla\mathbf{u} \tag{3.4-6}$$

where ρ is density of the fluid in the pores, k_s is the structure factor, Φ is the flow resistivity, ϕ is the porosity, c is the adiabatic wavevelocity of the fluid in the pores and $\mathbf{u}$ is the mean velocity of the fluid passing through the pores.
Note that we have changed the first of these equations compared to [9].
This may be rewritten as

$$\nabla^2 p - k_{eq}^2 p = 0,\ \forall\ x \in \Omega \tag{3.4-7}$$

where the equivalent bulk modulus, density, wavevelocity, and wavenumber are:

$$K_{eq} = \frac{K}{\phi} = \frac{\rho c^2}{\phi} \tag{3.4-8}$$

$$\rho_{eq} = \frac{\rho k_s}{\phi} + \frac{\Phi}{j\omega} \tag{3.4-9}$$

$$c_{eq} = \sqrt{\frac{K_{eq}}{\rho_{eq}}} \tag{3.4-10}$$

$$k_{eq} = \frac{\omega}{c_{eq}} \tag{3.4-11}$$

Here the use of structure factor is in accordance with [8] and is different from Craggs' notation.
These equations will, when using the Galerkin's method or as shown in [9], give:

$$\int_\Omega \nabla\mathbf{N^T}\nabla^\mathbf{T}\mathbf{N}d\Omega\,\overline{\mathbf{p}} - w^2\int_\Omega \mathbf{N^T}\frac{1}{c_{eq}^2}\mathbf{N}d\Omega\,\overline{\mathbf{p}} = -j\omega\int_\Gamma \mathbf{N^T}\rho_{eq}\mathbf{N}d\Gamma\,\overline{\mathbf{v}} \tag{3.4-12}$$

This is similar to the equations for ELEM03. Note that the impedance matrix is omitted here.

$$\left[\mathbf{k_p} - \omega^2\mathbf{m_p}\right][\overline{\mathbf{p}}] = -j\omega\mathbf{s_p}\overline{\mathbf{v}} \tag{3.4-13}$$

where

$$\mathbf{k_p} = \int_{\Omega_e}\mathbf{B_p^T}\mathbf{B_p}d\Omega \tag{3.4-14}$$

$$\mathbf{m_p} = \frac{1}{c_{eq}^2}\int_{\Omega_e}\mathbf{N^T}\mathbf{N}d\Omega \tag{3.4-15}$$

$$\mathbf{s_p} = \rho_{eq}\int_{\Gamma_e}\mathbf{N^T}\mathbf{N}d\Omega \tag{3.4-16}$$

$$\mathbf{B_p} = \left\{\begin{array}{c}\frac{\partial}{\partial x}\\ \frac{\partial}{\partial y}\\ \frac{\partial}{\partial z}\end{array}\right\}[N_1, N_2, ...] \tag{3.4-17}$$

Ω_e and Γ_e belong to one element. The material parameters are constant for each element. When coupling to ELEM03 a special procedure is introduced in [10].[1] When considering continuity of volume flow between the porous material and the fluid, introduce discrete volumic sources in the nodes at the boundaries. Here the total volumic source will be $a(U_a + U_p) = Q_b$. U_a is the volume velocity in the fluid, U_p is the volume velocity in the porous material, and Q_b is the volumic source at the boundary (which should be 0). The source contributions found be by $\mathbf{s_a}$ for ELEM03 and $\mathbf{s_{eq}}$ for ELEM04.
This indicate that the total source term will be

$$j\omega\rho U_a + j\omega\rho_{eq}U_p \neq 0 \tag{3.4-18}$$

In order to force this source term to zero the total set of equations for the equivalent fluid can be multiplied with ρ/ρ_{eq}. (which ensures continuity of volume flow over the boundary between the fluid and the porous material) This scaling with respect to the density of the fluid in the pore may be used to ensure continuity of volume flow between different porous materials too.
The new matrices are:

$$\left[\mathbf{k}^* - \omega^2\mathbf{m}^*\right][\overline{\mathbf{p}}] = -j\omega\mathbf{s}^*_{\mathbf{p}}\overline{\mathbf{v}} \tag{3.4-19}$$

where

$$\mathbf{k}^*_{\mathbf{p}} = \frac{\rho}{\rho_{eq}}\int_{\Omega_e}\mathbf{B}^{\mathbf{T}}_{\mathbf{p}}\mathbf{B}_{\mathbf{p}}d\Omega \tag{3.4-20}$$

$$\mathbf{m}^*_{\mathbf{p}} = \frac{\rho}{\rho_{eq}}\frac{1}{c_{eq}}\int_{\Omega_e}\mathbf{N}^{\mathbf{T}}\mathbf{N}d\Omega \tag{3.4-21}$$

$$\mathbf{s}^*_{\mathbf{p}} = \frac{\rho}{\rho_{eq}}\rho_{eq}\int_{\Gamma_e}\mathbf{N}^{\mathbf{T}}\mathbf{N}d\Omega \tag{3.4-22}$$

In the next paragraph the other models for equivalent fluids will be described. However all the equations will be equal to those of Craggs' (Zwicker, Kosten) model. Therefore only the expressions for the equivalent bulk modulus and density will be given.

Johnson-Allard's model

As mentioned before, in this model the pores may have a geometry of more random character than in the Rayleigh model. It can be shown that when using appropriate material parameters this model will approximate the Rayleigh model closely. Furthermore, in this model the frequency dependent characteristics of the thermal and viscous losses in the pores are taken into account. The theory will not be given in any detail here. Only the expressions used will be listed. See [8] for the details.
Newtons law and continuity volume flow:

$$-\nabla p = j\omega\rho_{eq}\mathbf{u} \tag{3.4-23}$$
$$j\omega p = -K_{eq}\nabla\mathbf{u} \tag{3.4-24}$$

where the equivalent bulk modulus, density, wavevelocity, and wavenumber are:

$$K_{eq} = \frac{\gamma P_0}{\phi(\gamma - (\gamma-1)[1 + \frac{\Phi'\phi G'_j(B^2\omega)}{jB^2\rho\omega k_s}]^{-1})} \tag{3.4-25}$$

$$\rho_{eq} = \frac{k_s\rho}{\phi}(1 + \frac{\Phi\phi}{j\rho\omega k_s}G_j(\omega)) \tag{3.4-26}$$

[1] **Note** The idea is used here but not as described in [10] because there the equivalent density is wrong.

$$c_{eq} = \sqrt{\frac{K_{eq}}{\rho_{eq}}} \tag{3.4-27}$$

$$k_{eq} = \frac{\omega}{c_{eq}} \tag{3.4-28}$$

where

$$G_j(\omega) = \sqrt{1 + \frac{4jk_s^2\eta\rho\omega}{\Phi^2\Lambda^2\phi^2}} \tag{3.4-29}$$

$$G'_j(\omega) = \sqrt{1 + \frac{4jk_s^2\eta\rho\omega}{\Phi'^2\Lambda'^2\phi^2}} \tag{3.4-30}$$

$$\Lambda = \frac{1}{c}\sqrt{\frac{8k_s\eta}{\phi\Phi}} \tag{3.4-31}$$

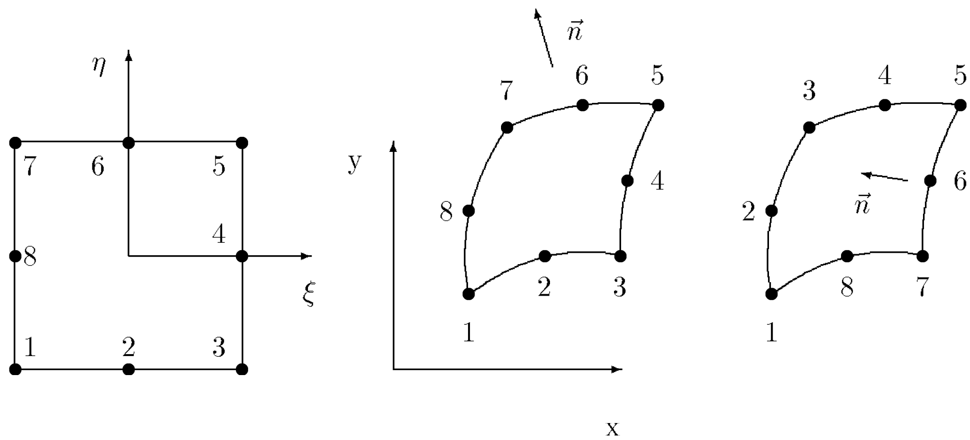
$$\Lambda' = \frac{1}{c'}\sqrt{\frac{8k_s\eta}{\phi\Phi}} = \sqrt{\frac{8k_s\eta}{\phi\Phi'}} \tag{3.4-32}$$

List of examples in appendix.

1. porex1 - Kundt's tube. Different models of the porous material compared.

3.5 ELEM05 - Porous material, Biot model

This element is used for acoustic waves in porous material. It is assumed that the frame is elastic. It is necessary to model the waves in the elastic frame, the waves in the pores and also the coupling between the two phases. The waves are described by the displacements of the fluid and the frame (the displacement of the frame and the relative displacement between the frame and the pore fluid). Therefore this model is considerably more storage-demanding than the equivalent fluid model (ELEM04).



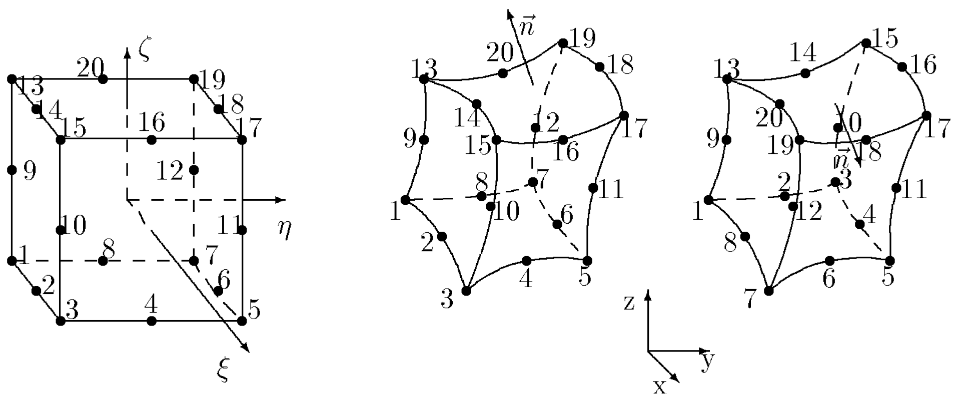
Reference element. Nodenumbering 2 dimensional version
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)



Reference element. Nodenumbering 3 dimensional version
Each side of the element has a number:

Side 2 ξ = -1, (node 1, 9, 13, 20, 19, 12, 7, 8)

Side 3 ξ = 1, (node 3, 4, 5, 11, 17, 16, 15, 10)

Side 4 η = -1, (node 1, 2, 3, 10, 15, 14, 13, 9)

Side 5 η = 1, (node 5, 6, 7, 12, 19, 18, 17, 11)

Side 6 ζ = -1, (node 1, 8, 7, 6, 5, 4, 3, 2)

Side 7 ζ = 1, (node 13, 14, 15, 16, 17, 18, 19, 20)

The element has 4 DOF for each node in the 2D and axisymmetric cases, and 6 DOF for each node in 3D case. The degrees of freedom are the global displacement components in the different directions.

Dimension	DOF	Comment
2D	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
	w_x	Global weighted relative displacement x-axis
	w_y	Global weighted relative displacement y-axis
2D axisym.	u_z	Global displacement z-axis
	u_r	Global displacement r-axis
	w_z	Global weighted relative displacement z-axis
	w_r	Global weighted relative displacement r-axis
3D	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
	u_z	Global displacement z-axis
	w_x	Global weighted relative displacement x-axis
	w_y	Global weighted relative displacement y-axis
	w_z	Global weighted relative displacement z-axis

The element matrices are symmetric and have complex coefficients.
It may be coupled to other type of elements:

- Acoustic fluid ELEM03. Must use coupling element (ELEM92).
- Elastic solid, with loss (ELEM20). Must use coupling element (ELEM94).

3.5.1 ELEM05 - Material parameters

The material parameters of this element is similar to those of ELEM04 (equivalent fluid) using Johnson-Allard model. In fact the same model is used to describe the thermal and viscous losses.

1. ρ_0 - Density of pore fluid.
2. c - Wave velocity of the fluid in the pores.
3. B^2 - Prandtl's number of pore fluid.
4. η -Viscosity of pore fluid
5. $\gamma = c_p/c_v$ of pore fluid
6. ρ_1 - Density of the porous material.
7. $Re[G]$ - Real part of the shear modulus
8. $Im[G]$ - Imaginary part of the shear modulus
9. $Re[\nu]$ - Real part of the Poisson's ratio
10. $Im[\nu]$ - Imaginary part of the Poisson's ratio

11. ϕ - porosity
12. k_s - Structure factor (tortuosity).
13. Φ - Flow resistivity.
14. Λ - Characteristic length for viscous losses
15. Λ' - Characteristic length for thermal losses

The model used is described in [8]

3.5.2 ELEM05 - Available source options

Typically a wave in this type of material is generated by external waves, either acoustic waves in a fluid or elastic waves in a solid. Therefore this element usually will be used with other elements, and the sources are described in the other type of elements. However, a type of source is included; source type 25 and 26. This give the opportunity to define stress at a boundary of an element.

3.5.3 ELEM05 - Available impedance options

None

3.5.4 ELEM05 - Use of the element

This element is most often used with other elements, primarely ELEM03 and ELEM20. The use of this element is similar to that of ELEM03.
A description of typical use of the element with ELEM03 is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the acoustic fluid (fluids). (PREL)
3. Define the prescribe DOF's. *i.e.* known displacements.(COND)
4. Define nodal connectivity of element.
5. Define velocities at boundaries or other types of sources. (SRCE)
6. Assemble and solve equations. (DYNM)

The points 1, 2, 3, 4, and 6 are mandatory. Usually all points are included in the definition of a modell.
Note on use for axisymmetric problems For the nodes on axis (r=0) the radial displacement of the frame and the radial relative displacement must be set to zero.

Example: Kundt's tube

Here some details of an example is given. The example is detailed more in the appendix. The example is a calculation on a Kundt's tube. The reason for the choice is the possibility to compare with analytical models (plane wave assumptions).
The geometry is shown in the appendix. The tube is axisymmetrical, radius is 0.1 m, the air- column is 0.30 m, while the thickness of the porous material is 0.1 m. Material parameter of the porous material is,

- density of the porous material, $\rho_1 = 35.0 kg/m^3$, VPREE(6)=35.0,
- shear modulus $G = 10^5 + j10^3$ Pa (VPREE(7)=1.0e5, VPREE(8)=1.0e3),

- Poissons rations 0.3 (VPREE(9)=0.3, VPREE(10)=0.0),
- porosity: $\phi = 0.9$ (VPREE(11)=0.9),
- structure factor: $k_s = 1.5$, (VPREE(12)=1.5),
- flow resisitivy: $\Phi = 10000 Ns/m^4$ (VPREE(13)=10000),
- characteristic length vicous loss $\Lambda = 2.0 \cdot 10^{-4}$ m,VPREE(14)=2.0e-4,
- characteristic length thermal loss $\Lambda' = 3.0 \cdot 10^{-4}$ m VPREE(15)=3.0e-4 .

Here only the material definition block of the input file :

```
.
.
PREL                                    ; Definition of material properties
2,15
1,1.205,343.4,0.0,0.0,0.0,0.0,0.0       ; material 1 - air
0.0,0.0,0.0,0.0,0.0,0.0,0.0
0.0
2,1.205,343.4,0.71,1.84e-5,1.4,43.0,1.0e5  ; material 2 - porous material
1.0e3,0.30,0.0,0.90,1.5,10000.,2.0e-4
3.0e-4
0
.
.
```

3.5.5 ELEM05 - Theory of element

Stress-strain relations

The Biot theory takes into account the elasticity of the frame of a porous material. The waves in the frame (the solid) and in the fluid must be described. In a series of articles Biot describes the elastic properties of the porous material. [11], [12], [13], [14], [15] .
The stress-strain relations are given:

$$\begin{aligned} \sigma_{ij}^s &= [(P-2N)\theta^s + Q\theta^f]\delta_{ij} + 2Ne_{ij}^s && (3.5\text{-}1) \\ \sigma_{ii}^f &= -\phi p = Q\theta^s + R\theta^f && (3.5\text{-}2) \end{aligned}$$

where σ is stress component in the direction indicated by the subscribed index, the superscibed index indicates fluid, f, or solid, s.

$$\begin{aligned} \theta &= \sum \frac{\partial u_i}{\partial x_i} \\ e_{ij} &= \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \end{aligned}$$

ϕ is the porosity of the material, p is the pressure of the fluid. N, P, Q, and R are the elasticitiy coefficients.
This presentation is given in [11] and also used by Allard in [8].
Another representation is also possible:

$$\begin{aligned} \sigma_{ij} &= 2\mu e_{ij} + (\lambda + \alpha^2 M)\theta^s - \alpha M\zeta && (3.5\text{-}3) \\ \pi &= \alpha M\theta^s + M\zeta && (3.5\text{-}4) \end{aligned}$$

where σ is the total stress ($\sigma_{ij} = \sigma_{ij}^s + \sigma_{ij}^f$) and $\pi = -p$, the pore pressure, μ and λ are the Lamé coeffisients of the elastic frame in vacuum. $\zeta = \phi(\theta^f - \theta^s)$, a measure for relative displacement between the frame and the fluid. This presentation is used in the litterature

I have found on Biot-theory and the finite element method. The connection between the two representations is given in [15].
The last version is used in FEMAK, but here the formulas to determine the different coefficients are reproduced for both versions.
N= μ is the shear modulus. The three bulk-moduli are introduced; K_f bulk modulus of the fluid, K_s bulk modulus of solid frame material, and K_b bulk modulus of the frame in vacuum. If the frame material is very stiff ($K_s \gg K_f$ and $K_s \gg K_b$), the following formulas can be used:

$$Q \approx K_f(1-\phi) \tag{3.5-5}$$

$$P \approx \frac{4}{3}N + K_b + \frac{(1-\phi)^2}{\phi}K_f \tag{3.5-6}$$

$$R = \frac{\phi^2 K_s}{1-\phi-\frac{K_b}{K_s}+\phi\frac{K_s}{K_f}} \approx \phi K_f \tag{3.5-7}$$

Furthermore, the following relations as given in [15]:

$$\alpha = \phi\frac{Q+R}{R} \approx \frac{K_f(1-\phi)+\phi K_f}{\phi K_f}\phi = 1 \tag{3.5-8}$$

$$M = \frac{R}{\phi^2} \approx \frac{K_f}{\phi} \tag{3.5-9}$$

$$\lambda = P - 2N - \frac{Q^2}{R} = K_b - \frac{2}{3}\mu \tag{3.5-10}$$

Wave equations

The wave equations using the first representation of the elasticity may be expressed as given by Allard [8], p.129:

$$(\rho_1+\rho_a)\frac{\partial^2\mathbf{u}^s}{\partial t^2} - \rho_a\frac{\partial^2\mathbf{u}^f}{\partial t^2} = (P-N)\nabla\nabla\cdot\mathbf{u}^s + Q\nabla\nabla\cdot\mathbf{u}^f + N\nabla^2\mathbf{u}^s - \Phi\phi^2 G(\omega)\frac{\partial}{\partial t}(\mathbf{u}^s-\mathbf{u}^f) \tag{3.5-11}$$

$$(\phi\rho_0+\rho_a)\frac{\partial^2\mathbf{u}^f}{\partial t^2} - \rho_a\frac{\partial^2\mathbf{u}^s}{\partial t^2} = R\nabla\nabla\cdot\mathbf{u}^f + Q\nabla\nabla\cdot\mathbf{u}^s + \Phi\phi^2 G(\omega)\frac{\partial}{\partial t}(\mathbf{u}^s-\mathbf{u}^f) \tag{3.5-12}$$

Where Φ is flow resistivity. ρ_0 is the density of the fluid, ρ_1 is the density of the frame, and $\rho_a = \rho_0\phi(k_s-1)$. k_s is the tortuosity (structure factor).
Inertial coupling is ensured by the appearance of $\mathbf{u}^s$ and $\mathbf{u}^f$ in both equations. Viscous losses are accounted with $\Phi G(\omega)$. The function $G(\omega)$ is dependent on the shape of the pores. The thermal losses is not explicitly shown in the equations. However they are accounted by a complex bulk modulus for the fluid.

$$K_f = (\gamma P_0)/\phi/[\gamma-(\gamma-1)[1+\frac{\Phi'\phi}{jB^2\omega k_s}G'(B^2\omega)]^{-1}]$$

Where details may be found in [8] p.92.
In the alternative representation the wave equation will be [13]:

$$\frac{\partial^2}{\partial t^2}(\rho\mathbf{u}+\rho_0\mathbf{w}) = \mu\nabla^2\mathbf{u} + (\mu+\lambda)\nabla\nabla\mathbf{u} - \alpha M\nabla\nabla\mathbf{w} \tag{3.5-13}$$

$$\frac{\partial^2}{\partial t^2}(\rho_0\mathbf{u}+\frac{\rho_0}{\phi}\mathbf{w}) = \alpha M\nabla\nabla\mathbf{u} - M\nabla\nabla\mathbf{w} + \frac{\rho_a}{\phi^2}\frac{\partial^2}{\partial t^2}\mathbf{w} + \Phi G(\omega)\frac{\partial}{\partial t}\mathbf{w} \tag{3.5-14}$$

where $\mathbf{w} = \phi(\mathbf{u}^f - \mathbf{u}^s)$ and $\rho = \phi\rho_0 + (1-\phi)\rho_1$, the density of the porous material. The two last terms in the last equation takes into account the viscous losses. Using the definition of ρ_a the last equation can be rewritten:

$$\frac{\partial^2}{\partial t^2}(\rho_f \mathbf{u} + \frac{\rho_1}{\phi} k_s \mathbf{w}) = \alpha M \nabla\nabla \mathbf{u} - M\nabla\nabla\mathbf{w} + \Phi G(\omega)\frac{\partial}{\partial t}\mathbf{w} \tag{3.5-15}$$

When assuming harmonic waves the equations may be rewritten:

$$\mu\nabla^2\mathbf{u} + (\mu+\lambda)\nabla\nabla\mathbf{u} - \alpha M\nabla\nabla\mathbf{w} + \omega^2\rho\mathbf{u} + \omega^2\rho_f\mathbf{w} = \mathbf{0} \tag{3.5-16}$$

$$\alpha M\nabla\nabla\mathbf{u} - M\nabla\nabla\mathbf{w} - j\omega\Phi G(\omega)\rho_f\mathbf{w} + \omega^2\rho_f\mathbf{u} + \omega^2\frac{\rho_f}{\phi}k_s\mathbf{w} = \mathbf{0} \tag{3.5-17}$$

I will use the two last equations in the rest of this section.
Note that the model used to describe the thermal and viscous losses for this element is the "Johnson-Allard" model which is described in [8] and in the theory section of ELEM04.

Biot theory and the finite element method

In the finite element method litterature it is common to restate the wave equations in a matrix formulation. I will use the formulation as given in Simon et.al. [16], and in Degrande [17]. [2]

$$\mathbf{L}^T\sigma + \omega^2\rho\mathbf{u} + \omega^2\rho_f\mathbf{w} = \mathbf{0} \tag{3.5-20}$$

$$\nabla\pi - j\omega\Phi G(\omega)\rho_f\mathbf{w} + \omega^2\rho_f\mathbf{u} + \omega^2\frac{\rho_f}{\phi}k_s\mathbf{w} = \mathbf{0} \tag{3.5-21}$$

The boundary conditions must be taken into account. The Dirichlet type boundary conditions:

$$\mathbf{u} - \underline{\mathbf{u}} = \mathbf{0}\ , x \in \Gamma_D^u \tag{3.5-22}$$

$$\mathbf{w} - \underline{\mathbf{w}} = \mathbf{0}\ , x \in \Gamma_D^w \tag{3.5-23}$$

The Neumann type boundary conditions:

$$\sigma_{\mathbf{n}} - \underline{\sigma_{\mathbf{n}}} = \mathbf{0}, x \in \Gamma_N^u \tag{3.5-24}$$

$$\pi - \underline{\pi} = 0, x \in \Gamma_N^w \tag{3.5-25}$$

In the above equations we have (3-dimensional):

$$\mathbf{L}^T = \begin{bmatrix} \frac{\partial}{\partial x_1} & 0 & 0 & \frac{\partial}{\partial x_2} & 0 & \frac{\partial}{\partial x_3} \\ 0 & \frac{\partial}{\partial x_2} & 0 & \frac{\partial}{\partial x_1} & \frac{\partial}{\partial x_3} & 0 \\ 0 & 0 & \frac{\partial}{\partial x_3} & 0 & \frac{\partial}{\partial x_2} & \frac{\partial}{\partial x_1} \end{bmatrix}$$

$\mathbf{u}^T = [u_1u_2u_3]$, $\mathbf{w}^T = [w_1w_2w_3]$, $\sigma^T = [\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{23}, \sigma_{13}]$, $\nabla^T = [\frac{\partial}{\partial x_1}, \frac{\partial}{\partial x_2}, \frac{\partial}{\partial x_3}]$ and $\mathbf{m}^T = [111000]$.

[2]It is important to note that both Simon et.al. and Degrande uses a different notation for the viscous losses. Simon et.al. use:

$$\nabla\pi - j\frac{\omega}{k}\mathbf{w} + \omega^2\rho_f\mathbf{w} + \omega^2\frac{\rho_f}{\phi}\mathbf{w} - = 0 \tag{3.5-18}$$

Here the parameter k accounts for the viscous losses. The two equations are equal if

$$\frac{1}{k} = \Phi G(\omega) + \frac{\rho_a}{\phi^2}\frac{\partial}{\partial t} \tag{3.5-19}$$

The strain-displacement relationship is: $\mathbf{e} = \mathbf{L}\ \mathbf{u}$ and $\zeta = \nabla^T \mathbf{w}$ and the stress-strain relationship:

$$\sigma = (\mathbf{D} + \alpha^2 M \mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{e} + \alpha M \mathbf{m}\zeta \tag{3.5-26}$$

$$\pi = \alpha M \mathbf{m}^\mathbf{T}\mathbf{e} + M\zeta \tag{3.5-27}$$

The elasticity matrix of the frame in vacuum is:

$$\mathbf{D} = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

Introducing finite element approximations: $\mathbf{u} = \mathbf{N}\overline{\mathbf{u}}$, $\mathbf{w} = \mathbf{N}\overline{\mathbf{w}}$. $\overline{\mathbf{u}}$, and $\overline{\mathbf{w}}$ are the nodal values of the variables.

$$\mathbf{L}^\mathbf{T}[(\mathbf{D} + \alpha^2 M \mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{L}\mathbf{N}\overline{\mathbf{u}} + \alpha M \mathbf{m}\nabla^T \mathbf{N}\overline{\mathbf{w}}] + \omega^2\rho\mathbf{N}\overline{\mathbf{u}} + \omega^2\rho_f\mathbf{N}\overline{\mathbf{w}} = \mathbf{0} \tag{3.5-28}$$

$$\nabla(\alpha M \mathbf{m}^\mathbf{T}\mathbf{L}\mathbf{N}\overline{\mathbf{u}} + \alpha Q\nabla^T\mathbf{N}\overline{\mathbf{w}}) - j\omega\sigma G(\omega)\rho_f\mathbf{N}\overline{\mathbf{w}} + \omega^2\rho_f\mathbf{N}\overline{\mathbf{u}} + \omega^2\frac{\rho_f}{\phi}\alpha_\infty\mathbf{N}\overline{\mathbf{w}} = \mathbf{0} \tag{3.5-29}$$

Using Galerkin's method, the following equations are found:

$$\begin{aligned} &\textstyle\int_\Omega \mathbf{N}^\mathbf{T}\{\mathbf{L}^\mathbf{T}[(\mathbf{D} + \alpha^2 M\mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{L}\mathbf{N}\overline{\mathbf{u}} + \alpha M\mathbf{m}\nabla^T\mathbf{N}\overline{\mathbf{w}}] + \omega^2\rho\mathbf{N}\overline{\mathbf{u}} + \omega^2\rho_f\mathbf{N}\overline{\mathbf{w}}\}d\Omega - \\ &\qquad \textstyle\int_{\Gamma_N^s} \mathbf{N}^\mathbf{T}(\sigma_\mathbf{n} - \underline{\sigma_\mathbf{n}})d\Gamma = 0 \end{aligned} \tag{3.5-30}$$

$$\begin{aligned} &\textstyle\int_\Omega \mathbf{N}^\mathbf{T}[\nabla(\alpha M\mathbf{m}^\mathbf{T}\mathbf{L}\mathbf{N}\overline{\mathbf{u}} + \alpha Q\nabla^T\mathbf{N}\overline{\mathbf{w}}) - j\omega\Phi G(\omega)\rho_f\mathbf{N}\overline{\mathbf{w}} + \omega^2\rho_f\mathbf{N}\overline{\mathbf{u}} + \omega^2\frac{\rho_f}{\phi}\alpha_\infty\mathbf{N}\overline{\mathbf{w}}]d\Omega - \\ &\qquad \textstyle\int_{\Gamma_N^w} \mathbf{N}^\mathbf{T}(\pi - \underline{\pi})d\Gamma = \mathbf{0} \end{aligned} \tag{3.5-31}$$

Where $\underline{\pi}$ and $\underline{\sigma_\mathbf{n}}$ are boundary conditions.
By partial integration we get:

$$\begin{aligned} &-\int_\Omega \mathbf{L}^\mathbf{T}\mathbf{N}^\mathbf{T}(\mathbf{D} + \alpha^2 M\mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{L}\mathbf{N}d\Omega\overline{\mathbf{u}} - \int_\Omega \nabla\mathbf{N}^\mathbf{T}\alpha M\nabla^T\mathbf{N}d\Omega\overline{\mathbf{w}} + \\ &\qquad \omega^2\int_\Omega \rho\mathbf{N}^\mathbf{T}\mathbf{N}d\Omega\overline{\mathbf{u}} + \omega^2\int_\Omega \rho_f\mathbf{N}^\mathbf{T}\mathbf{N}d\Omega\overline{\mathbf{w}} + \\ &\qquad \int_\Gamma \mathbf{N}^\mathbf{T}[(\mathbf{D} + \alpha^2 M\mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{L}\mathbf{N}\overline{\mathbf{u}} + \alpha M\nabla^T\mathbf{N}\overline{\mathbf{w}}]d\Gamma - \\ &\qquad \int_{\Gamma_N^s} \mathbf{N}^\mathbf{T}(\sigma_\mathbf{n} - \underline{\sigma_\mathbf{n}})d\Gamma = 0 \end{aligned} \tag{3.5-32}$$

$$\begin{aligned} &-\int_\Omega \nabla\mathbf{N}^\mathbf{T}\alpha M\nabla^\mathbf{T}\mathbf{N}d\Omega\overline{\mathbf{u}} - \int_\Omega \nabla\mathbf{N}^\mathbf{T}\alpha Q\nabla^T\mathbf{N}d\Omega\overline{\mathbf{w}} - j\omega\int_\Omega \mathbf{N}^\mathbf{T}\Phi G(\omega)\rho_f\mathbf{N}d\Omega\overline{\mathbf{w}} + \\ &\qquad \omega^2\int_\Omega \mathbf{N}^\mathbf{T}\rho_f\mathbf{N}d\Omega\overline{\mathbf{u}} + \omega^2\int_\Omega \mathbf{N}^\mathbf{T}\frac{\rho_f}{\phi}\alpha_\infty\mathbf{N}d\Omega\overline{\mathbf{w}} - \\ &\qquad \int_\Gamma \mathbf{N}^\mathbf{T}(\alpha M\nabla^T\mathbf{N}\overline{\mathbf{u}} + \alpha Q\nabla^T\mathbf{N}\overline{\mathbf{w}}d\Gamma + \int_{\Gamma_N^w} \mathbf{N}^\mathbf{T}(\pi - \underline{\pi})d\Gamma = \mathbf{0} \end{aligned} \tag{3.5-33}$$

$$\left[\begin{bmatrix} \mathbf{k_{uu}} & \mathbf{k_{uw}} \\ \mathbf{k_{wu}} & \mathbf{k_{ww}} \end{bmatrix} + j\omega\begin{bmatrix} \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{c_{ww}} \end{bmatrix} - \omega^2\begin{bmatrix} \mathbf{m_{uu}} & \mathbf{m_{uw}} \\ \mathbf{m_{wu}} & \mathbf{m_{ww}} \end{bmatrix}\right]\begin{bmatrix} \overline{\mathbf{u}} \\ \overline{\mathbf{w}} \end{bmatrix} = \begin{bmatrix} \mathbf{f_u} \\ \mathbf{f_w} \end{bmatrix}$$

Stiffness, mass and damping matrices

$$\mathbf{k_{uu}} = \int_{\Omega_b} \mathbf{B_u^T}(\mathbf{D} + \alpha^2 M \mathbf{m}\mathbf{m}^\mathbf{T})\mathbf{B_u} d\Omega \tag{3.5-34}$$

$$\mathbf{k_{uw}} = \mathbf{k_{wu}^T} = \int_{\Omega_b} \mathbf{B_u^T} \alpha M \mathbf{m} \mathbf{B_w} d\Omega \tag{3.5-35}$$

$$\mathbf{k_{ww}} = \int_{\Omega_b} \mathbf{B_w^T} M \mathbf{B_w} d\Omega \tag{3.5-36}$$

$$\mathbf{m_{uu}} = \int_{\Omega_b} \mathbf{N^T} \rho \mathbf{N} d\Omega \tag{3.5-37}$$

$$\mathbf{m_{uw}} = \mathbf{m_{wu}^T} = \int_{\Omega_b} \mathbf{N^T} \rho_f \mathbf{N} d\Omega \tag{3.5-38}$$

$$\mathbf{m_{ww}} = \int_{\Omega_b} \mathbf{N^T} \frac{\rho_f}{\phi} k_s \mathbf{N} d\Omega \tag{3.5-39}$$

$$\mathbf{c_{ww}} = \int_{\Omega_b} \mathbf{N^T} \Phi G(\omega) \mathbf{N} d\Omega \tag{3.5-40}$$

$$\begin{aligned} \mathbf{B_u} &= \mathbf{LN} \\ \mathbf{B_w} &= \nabla^\mathbf{T} \mathbf{N} \end{aligned} \tag{3.5-41}$$

Source terms for the Biot equations:

$$\mathbf{f_u} = \mathbf{N^T} \underline{\sigma} \tag{3.5-42}$$
$$\mathbf{f_w} = \mathbf{N^T} \underline{\pi} \tag{3.5-43}$$

1. layer1 - Layered material, aluminum, elastic porous material.

3.6 ELEM08 - Plate/beam

This element is used for the modelisation of plates (or beam in 2D). It follows the formulation of *thick* or *Reissner-Mindlin* plates, for which shear stresses are taken into account. The plate element is defined by nine nodes (Lagrange interpolation). The numbering of those nodes follows the order of the figure bellow. A clockwise or counterclockwise orientation will only modify the definition of the two rotational axis. The beam element is defined by three nodes

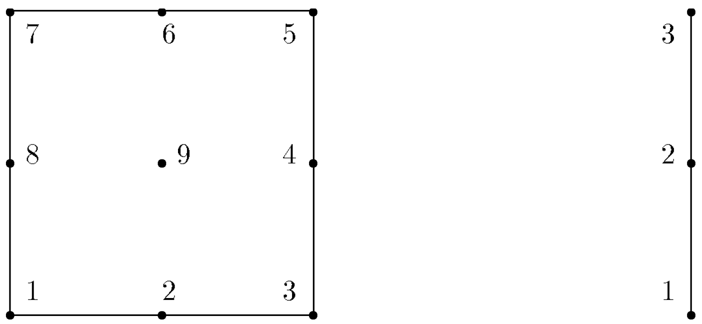


plate element beam element

In 3D, the plate element has three DOF per node. They are defined w.r.t to *local* axis, which are :

- the first local axis (axis I) has the direction defined by nodes 1-3 (in that order).
- the third local axis (axis III) is defined by the vectorial product between the axis I and the axis defined by nodes 3-5.
- the second local axis (axis II) is defined by the vectorial product of axis III and axis I.

For example, in the figure above, as the plate element has a rectangular shape, axis I would be defined by nodes 1-2-3, axis II by nodes 3-4-5, axis III by the vectorial product between axis I and II (it points towards the reader).
In 2D, the beam element has two DOF per node. They are defined w.r.t to *local* axis, which are :

- the first local axis (axis I) has the direction defined by nodes 1-2-3 (in that order).
- the second local axis (axis II) is directly normal to the first axis (i.e. a 90 degree anti-clockwise rotation of axis I gives axis II)
- the third local axis (axis III) is defined by the vectorial product of axis I and II.

For example, in the figure above, axis I is defined by nodes 1-2-3, and axis II is normal to axis I, pointing to the left, axis III is the vectorial product of axis I and II (it points towards the reader).

Dimension	DOF	Comment
2D	w	Normal displacement along axis II
	θ	bending angle about axis III (right-hand rule orientation).
3D	w	Normal displacement along axis III
	θ_1	Bending angle about axis I, (right-hand rule orientation)
	θ_2	Bending angle about axis II, (right-hand rule orientation)

The element matrices are symmetric and have real coefficients.
It may be coupled to other type of elements:

- Acoustic fluid ELEM03. Must use coupling element (ELEM91).
- Porous material, Biot model ELEM05. Must use coupling element (ELEM95).

3.6.1 ELEM08 - Material Parameters

The element has five parameters for the plate, and six for the beam :

1. E - Young's modulus
2. ν - Poisson's ratio
3. - for plane stresses, parameter set to 0.
   - for plane strains, parameter set to 1.
4. ρ - material density
5. h - plate thickness

3.6.2 ELEM08 - Available source options

An excitation may be introduced at one or several nodes of the plate. The exitation is introduce using source type 0. Note that this source is designed for acoustic point source. It has a "buildt-in" frequency dependence.

3.6.3 ELEM08 - Available impedance options

None

3.6.4 ELEM08 - Use of the element

A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the plate. (PREL)
3. Define the prescribed DOF's. That is known displacement or rotation. (COND)
4. Define the nodal connectivity of the element (ELEM)
5. Define external forces at the surface of the plate. (SRCE)
6. Assemble and solve equations. (DYNM)

The points 1,2,3,4 and 6 are mandatory. Usually, all points are included in the definition of the model.

3.6.5 ELEM08 - example of use

As an example, we can calculate the first eigen frequency of a square clamped plate of unit dimension, and thickness 1% of the length. We use a grid of 5x5 elements.
The elastic parameters of the plate are :

1. E=7.1e10 Pa
2. $\nu = 0.3$
3. plane stresses (parameter set to 0.)
4. $\rho = 2700\ kg/m^3$
5. h = 0.001 m

Input file for FEMAK :

```
COMT
     the first five modes of a unit square plate
     thickness : 0.001 m
     clamped boundaries
     grid : 5x5 elements (ELEM08)

COOR
121,3,3
    1     0.0     0.0     0.0
    :      :       :       :
    :      :       :       :
  121     0.4     1.0     0.0
0
PREL
1,5
1,7.1e10,0.3,0.,2700.,0.001
0
COND
111
1
1,3,5,......
............
0
0
ELEM
25,25,0,9,8
    1    1    1    8    1    1    1    2   23    ...
    :    :    :    :    :    :    :    :    :     :
    :    :    :    :    :    :    :    :    :     :
   25    1    1    8    1    1   87   89   88    ...
0
SOLR
VALP
5,20,0.001,0.0,10,0,12,1.D-12
STOP
```

Running this input file with FEMAK, gave the first five frequencies, to be compared with the values from Leissa [18]

- $\lambda_1 = 36.01$ (from [18] : $\lambda_1 = 35.99$)
- $\lambda_2 = 73.99$ (from [18] : $\lambda_2 = 73.40$)
- λ_{2bis} same frequencies for this double mode
- $\lambda_3 = 108.96$ (from [18] : $\lambda_3 = 108.22$)
- $\lambda_4 = 136.68$ (from [18] : $\lambda_4 = 131.64$)

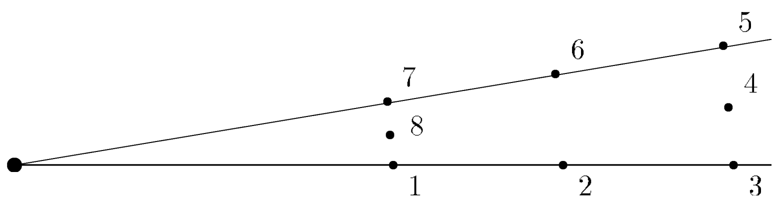
NB : the frequencies are given in terms of λ (non-dimensional), where λ is defined as :
$\lambda = \omega\sqrt{\frac{\rho}{D}}$
with $D = \frac{Eh^3}{12(1-\nu^2)}$
and $\omega = 2\pi f$, f is the natural frequency of the plate.

3.6.6 ELEM08 - Theory of the element

There are two formulations for the modelisation of plates. The first one states that, for thin plate, the rotation angle in one direction is directly connected to the slope of the deflection in that direction. The second one, considers that for thick plates, shearing through the thickness can not be neglected, and the relations between rotation angles and slopes have to take shear angles into account. This last formulation is the most satisfactory for computational purposes. However, it leads to one numerical difficulty. The stiffness matrix of such a plate element is composed of two terms : a pure bending stiffness and a shearing stiffness. For rather thin plates, the second term tends to increase disproportionally, and the problem degenerates. A reduced integration has to be performed on this term, to relax the system.
A full presentation of this element can be found in [19].

3.7 ELEM11 - Wave Envelope Element for Acoustic Fluid

This element is used for terminating an acoustic field meshed by finite elements towards infinity. The element normally has a length which is many wavelengths long. As it is not mapped to infinity, it must be terminated by a suitable impedance. Normally this impedance will be characteristic impedance of the fluid, "ρc". In the element the waves are assumed to have basically the character of outgoing waves. The element must therefore be oriented as shown in the figure below with respect to the source region. Local side 2 towards the source region and side 3 is the one that must be given a terminating impedance.



Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)

The element must always be oriented like this. The 3-dimensional version of the element has a numerotation like for element 3 , and must also be oriented with side 2 towards the source and side 3 towards infinity.
The element has only one DOF for each node, the complex acoustic pressure.

Dimension	DOF	Comment
2D, 2D axisym., 3D	p	Acoustic pressure

The element matrixes are non symmetric and have complex coefficients. It may be coupled to one other type of element

- Acoustic fluid ELEM03

3.7.1 ELEM11 - Material Parameters

The element has two material parameters that must be described

1 ρ - fluid density

2 c - speed of sound in the fluid

3.7.2 ELEM11 - Available source options

None

3.7.3 ELEM11 - Available impedance options

side 3 can be given an impedance. Note that it is possible to join several elements of this type after each other in the direction away from the source. This might increase accuracy. The elements facing the unmeshed infinite domain <u>must</u> be given a terminating impedance. For the two dimensional case, element side 4 might be given an impedance. This feature was included for studies of sound propagation above an impedance surface.

3.7.4 ELEM11 - Theory of the element

The idea of "infinite elements" is to assume certain characteristics of the acoustic propagation within the element. In our case, the characteristics of outwardly propagating waves. Astley [20] adopts the shape functions

$$N_i(r,\theta)\frac{r_i}{r}exp[-ik(r-r_i)] \tag{3.7-1}$$

for wave propagation in three dimensions.where $N_i(r,\theta)$ denotes the conventional shape function assosiated with node i. r_i is the radius to the node i and r and θ are polar coordinates. In two dimensions, the exponentials are replaced by Hankel functions. Conventional infinite elements use Galerkin weighting, but in the present wave envelope elements, the suggestion by Astley is followed and the complex conjugate of the shape function used as weighting. This produces expressions which are easy to integrate, but also matrixes which are non-symmetric.

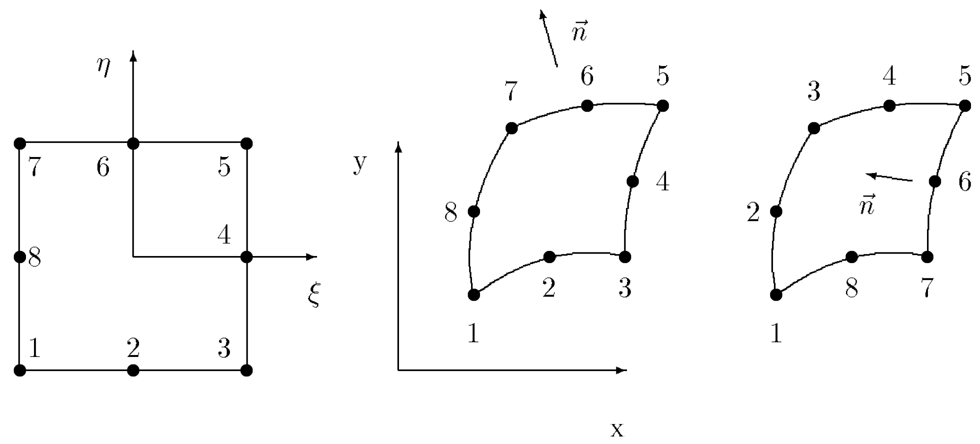
3.7.5 ELEM11 - example of use

List of examples in appendix.

1. stmpwee1, stmpwee5, stmpwee7 - Radiation impedance using wave envelope elements

3.8 ELEM20 - Elastic solid with loss

This element is used for isotropic elastic solids with loss. It is similar to ELEM02. It is 2-dimensional. It can be used for wave propagation in these types of materials The loss is described using the "Rayleigh modell".
Second order shape functions are used. 8 nodes for the 2-dimensional case.
The material is described using Young's modulus, Poisson's coefficient, specific mass and two loss coefficients.
In 2D it can handle plane stress.



Reference element. Nodenumbering 2 dimensional version. Two examples on global elements, one with normal pointing out of element and one with normal pointing into the element.
Each side of the element has a number:

Side 2 ξ = -1, (node 7, 8, 1)

Side 3 ξ = 1, (node 3, 4, 5)

Side 4 η = -1, (node 1, 2, 3)

Side 5 η = 1, (node 5, 6, 7)

The element has two DOF's for each node, the global displacements.

Dimension	DOF	Comment
2D	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
2D axisym.	u_z	Global displacement z-axis
	u_r	Global displacement r-axis

The element matrices are symmetric and have complex coeffients. It may be coupled to other media.

- Acoustic fluid (ELEM03). Must use coupling element (ELEM93).
- Porous materials with elastic frame (ELEM05). Must use coupling element (ELEM94).

3.8.1 ELEM20 - Material parameters

The fluid is described by only four parameters.

1. Young's modulus
2. Poisson's coefficient
3. Specific mass
4. Loss parameter α
5. Loss parameter β

3.8.2 ELEM20 - Available source options

source type 25,26 - Specified forces.

3.8.3 ELEM20 - Available impedance options

None

3.8.4 ELEM20 - Use of the element

In this description no coupling to other media is involved. For coupled problems look in the sections for coupling elements (ELEM93 and ELEM94).
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of material. (PREL)
3. Define the prescribe DOF's. *i.e.* the known displacements.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Define sources. (SRCE)
6. Assemble and solve
    - Dynamical problem (DYNM)

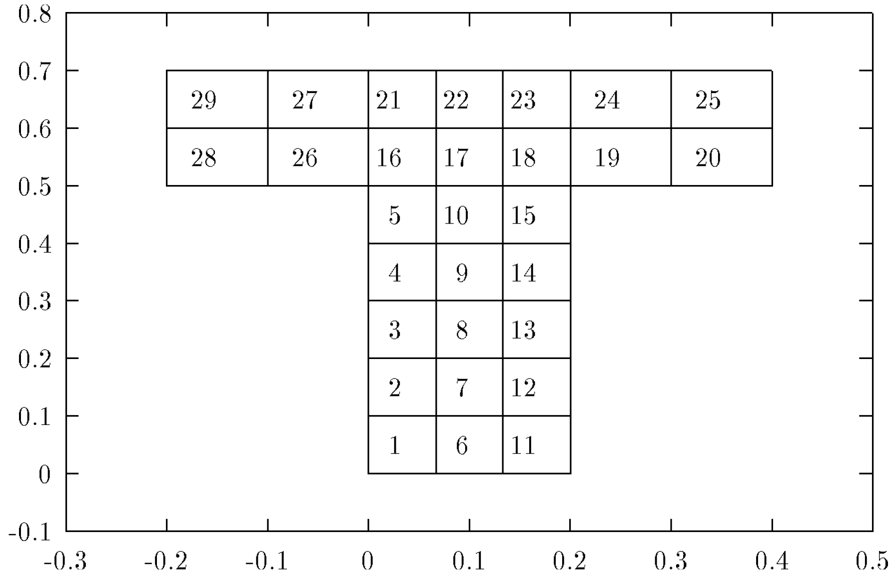
The points 1, 2, 3, 4 and 6 are mandatory.

Example: Forced vibration T-shaped device.

A simple T-shaped geometry is investigated. The material is steel. The lower part of the object is clamped ($u_x = u_y = 0$) for node 1-7 (nodes with y=0.0).
Material parameters:

E-modul $[Pa]$	Poisson's ratio	Density $[\frac{kg}{m^3}]$
$1.95 \cdot 10^1 1$	0.28	7700

The geometry is shown in the figure below.




```
COND
11                                ; dofs ux and uy set to zero
2                                 ;       start loop over nodes
1,7                               ;                          nodes 1 to 7 forced to zero
0                                 ; end loop of nodes
0                                 ; end COND-block
PREL
1,5                               ; NGPE=1, NPRE=5
1,2.0e6,0.3,2.3,0.0,0.0           ; IGPE=1, Material data
0                                 ; end PREL-block
.
.
.
SRCE
1,1,2                             ; NGSRC=1, NFREQ1=1, NSRC=2
1,26                              ; IGSRC=1, SRCPAR=26 (Source type 26)
27,5,1                            ; IEL0=27, IFL=5, IGSRC=1
0.0,1.0                           ; Fx=0.0, Fy=1.0 (force pointing upwards)
-1.0,0.0,-1.0,0.0,-1.0,0.0        ; Nodal coefficients
29,5,1                            ; IEL0=29, IFL=5, IGSRC=1
0.0,1.0                           ; Fx=0.0, Fy=1.0 (force pointing upwards)
-1.0,0.0,-1.0,0.0,-1.0,0.0        ; Nodal coefficients
.
.
.
```

The input-file can be found among the elastic examples. The name of the file is elaex12.inp. Some results for this example can be found in appendix.

3.8.5 ELEM20 - Theory of element

This element use the socalled Rayleigh model for loss mechanisms in an elastic solid, see e.g. Smith [21]. In this model it is assumed that it is possible to find the loss matrix from a linear relationship between the stiffness- and mass-matrix.
The stress-strain relation for the elastic material is:

$$\sigma = \mathbf{D}\epsilon = \mathbf{DLu} \tag{3.8-1}$$

$$\epsilon = \mathbf{Lu} \tag{3.8-2}$$

Where σ is the stress tensor, $\mathbf{D}$ is the elasticity matrix , $\mathbf{u}$ is the displacement of the solid, and ϵ is the strain tensor.

$$\mathbf{L}^T\sigma + \omega^2\rho\mathbf{u} = \mathbf{0} \tag{3.8-3}$$
$$\mathbf{L}^T\mathbf{DLu} + \omega^2\rho\mathbf{u} = \mathbf{0} \tag{3.8-4}$$

When introducing FEM discretization, $\mathbf{u} = \mathbf{N}\ \bar{u}$, using Galerkin's method and Green's theorem:

$$\int_\Omega \mathbf{L^T N^T DLN} d\Omega\overline{\mathbf{u}} - \omega^2 \int_\Omega \rho\mathbf{N^T N} d\Omega\overline{\mathbf{u}} + \int_{\Gamma_N^s} \mathbf{N^T}(\sigma - \underline{\sigma})d\Gamma = 0 \tag{3.8-5}$$

$$\left[\mathbf{k} - \omega^2\mathbf{m}\right]\bar{\mathbf{u}} = \mathbf{f}$$

Stiffness, mass and damping matrices

$$\mathbf{k} = (1 + j\alpha)\int_{\Omega_b} \mathbf{B^T DB} d\Omega \tag{3.8-6}$$

$$\mathbf{m} = (1 + j\beta)\int_{\Omega_b} \mathbf{N^T}\rho\mathbf{N} d\Omega \tag{3.8-7}$$

$$\mathbf{B} = \mathbf{LN} \tag{3.8-8}$$

Source terms:

$$\mathbf{f} = \mathbf{N^T}\underline{\sigma} \tag{3.8-9}$$

The stress-strain relationships used in this element are:
for 2-dimensional case:

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x_1} & 0 \\ 0 & \frac{\partial}{\partial x_2} \\ \frac{\partial}{\partial x_2} & \frac{\partial}{\partial x_1} \end{bmatrix}$$

for axisymmetrical case:

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x_1} & 0 \\ 0 & \frac{\partial}{\partial x_2} \\ 0 & \frac{1}{x_2} \\ \frac{\partial}{\partial x_2} & \frac{\partial}{\partial x_1} \end{bmatrix}$$

$\mathbf{u}^T = [u_1 u_2]$, $\sigma^T = [\sigma_{11}, \sigma_{22}, \sigma_{12}]$,

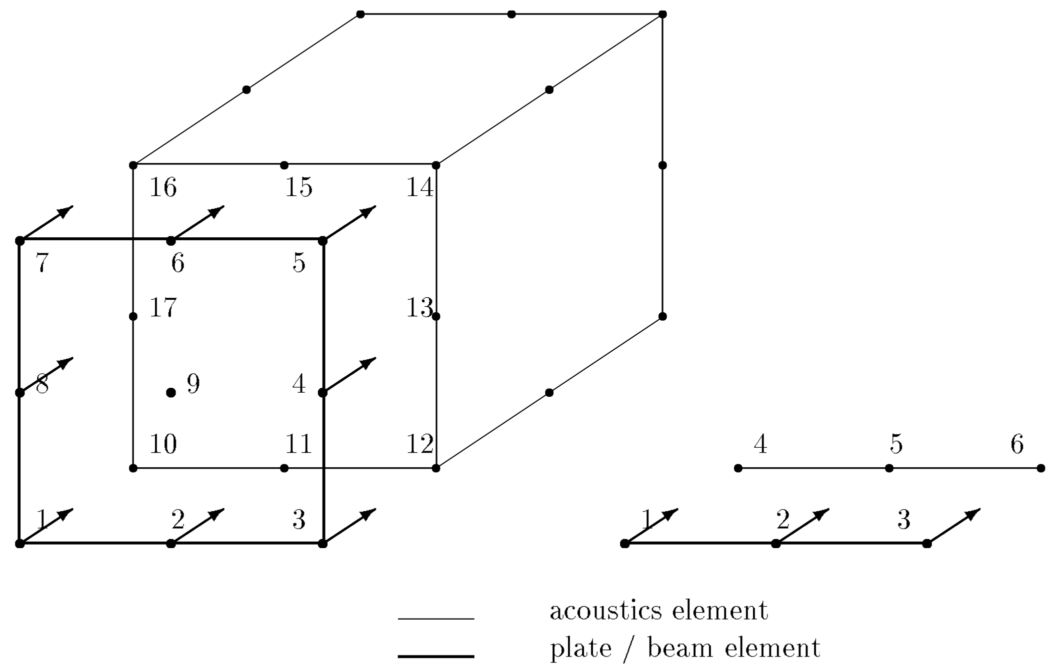
List of examples in appendix.

1. elaex12 - Forced vibration.

2. layer1 - Layered material, aluminum, elastic porous material.

3.9 ELEM91 - Coupling element plate/beam - acoustic elements

This element is to be used when a plate (or beam) element ELEM08 and an acoustic element ELEM03 are superposed. The junction between those two elements is performed by the introduction of a virtual coupling element, which insures the continuity of stress and displacement through the two elements. The coupling element is defined by two sets of nodes at the interface: first, the 9 nodes for the plate, then the 8 acoustic nodes of the interface (in 2D, 3 nodes for the beam, 3 nodes for the acoustics). The orientation of the numbering of the "plate part" of the element must be the same than the orientation of the numbering of the "acoustic part" of the element, as shown in the figure below.



Nodes 1,2,...,8 and nodes 10,11,...,17 have respectively the same coordinates. In 2D, it would be nodes 1,2,3 and nodes 4,5,6.
The element has three (2 in 2D) DOF per plate node (one deflection, and two rotations (1 in 2D) defined just as for ELEM08), and one DOF per acoustic node (the pressure).

Dimension	DOF	Comment
2D, beam	w	Normal displacement along beam-axis II
	θ	Rotation about beam-axis III
2D, acoustic	p	Acoustic pressure
3D, plate	w	Normal displacement along plate-axis III
	θ_1	Rotation about plate-axis I
	θ_2	Rotation about plate-axis II
3D, acoustic	p	Acoustic pressure

The total number of DOF is 35 (9 in 2D).
The element matrices are non symmetric and have real coefficients.

3.9.1 ELEM91 - Material Parameters

The element has one material parameter :

1. ρ - fluid density
2. Parameter for definition of the normal of the element

   VPREE(2)= 0.0 the plate (or beam) normal (cf. ELEM08 description) points towards the porous element

   VPREE(2)= 1.0 the plate (or beam) normal points outside the porous element.

3.9.2 ELEM91 - Available source options

None

3.9.3 ELEM91 - Available impedance options

None

3.9.4 ELEM91 - Use of the element

Here the use of this coupling element is briefly described. Both elements type 3 and type 8 must be present when using this element.
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the density of the acoustic medium. (PREL)
3. Define the nodal connectivity of the element. (ELEM)

All points are mandatory.

3.9.5 ELEM91 - example of use

In this example, we test the coupling between a simply supported plate of dimension : 0.2x0.5 m^2 and thickness 0.001 m, and a rectangular acoustic cavity of length 0.5 m, behind this plate. The grid is composed of 4x4x5 elements : 16 plate elements (ELEM08), 80 acoustic elements (ELEM03), and for the junction 16 added elements : the coupling elements (ELEM91). We shall compare the resonance frequency of the plate alone, and of the plate coupled with this cavity.
The material properties of the plate are the same as in the example of the ELEM08 description, and for the fluid, the density is 1.25 kg/m^3, and the speed of sound is 340 m/s.
The input file is written below

```
COMT
     coupling a plate and an acoustic cavity
     plate : 0.2x0.5 / thickness 0.001 / simply supported
     acoustic cavity : 0.2x0.5x0.5
     grid : 4x4 plate elements (ELEM08)
            4x4x5 acoustic elements (ELEM03)
            4x4 coupling element (ELEM91)

COOR
596,3,3
    1     0.00      0.00      0.00
    :      :         :         :
    :      :         :         :
```

```
  596     0.05     0.25    -0.50
0
DLPN
3,1,2,3,...................
     76,77,78,79,80,81
1,82,83,84,................
     ......................
     592,593,594,595,596
0
PREL
3,5
1,1.25,340.
2,7.1e10,0.3,0.,2700.,0.001
3,1.25
0
COND
1
1,3,5,...........
0
0
ELEM
112,96,16,20,3,,1
    1    1    1    3    1    1   82   83   ....
    :    :    :    :    :    :    :    :    :
    :    :    :    :    :    :    :    :    :
   80    1    1    3    1    1  465  468   ....
  ...  ...  ...  ...  ...  ...  ...  ...
   81    1    1   91    3    1    2   19   ....
    :    :    :    :    :    :    :    :    :
    :    :    :    :    :    :    :    :    :
   96    1    1   91    3    1   55   57   ....
  ... ...   ...  ...  ...  ...  ...
   97    1    1    8    2    1    1    2   ....
    :    :    :    :    :    :    :    :    :
    :    :    :    :    :    :    :    :    :
  112    1    1    8    2    1   82   83   ....
0
SRCE
1,1,1
1,0
0.,10.
6,129,1
DYNM
1
0
79.
STOP
```

The plate here is excited by a harmonic load centered on the DOF number 129.
The excursion in frequency shows resonance frequencies at 79.5 Hz, 96.9 Hz, 252.9 Hz, 281.9 Hz (we are only concerned with the first mode of each class of symmetry).
The natural frequencies of the plate alone are : 70.2 Hz, 99.5 Hz, 255.8 Hz, 284.8 Hz.
Strong coupling only occurs for the first mode.

3.9.6 ELEM91 - Theory of the element

As mentioned earlier, the coupling element between the plate and the acoustic volume has to insure the continuity of stress and displacement at the interface.

1. *continuity of stress*: the acoustic pressure at the surface of the plate acts as an external load on it. Writing the work done by this force, we obtain :

   $\delta W = \int_A (p\mathbf{n}).(\delta \mathbf{w})dA$

   with $\mathbf{n}$, the outward normal to the plate.

   Writing the Hamilton's principle with this expression leads to the governing equation of a plate loaded by a pressure distribution at its surface.

2. *continuity of displacement*: the vibrating plate acts on the acoustic volume as a flexible surface with normal velocity $v_n = \mathbf{v}.\mathbf{n}$. Writing this as a boundary condition for the acoustic volume, we obtain :

   $\partial p/\partial n = \rho \partial v_n/\partial t$

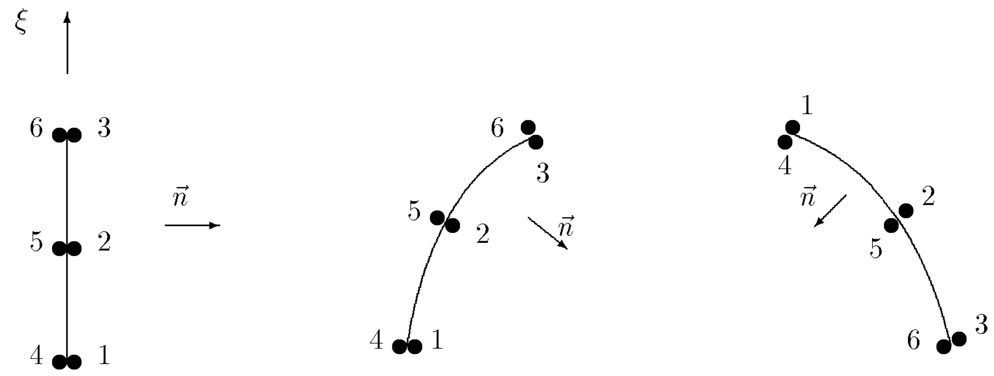
   with $\mathbf{n}$, the outward normal to the surface, and ρ, the fluid density.

   Writing the velocity as the derivative on time of the displacement, and introducing this boundary condition into the weighted residual formulation of the wave equation with boundary conditions, we end up with the equation of motion of the acoustic volume submitted at one surface to the displacement of the plate.

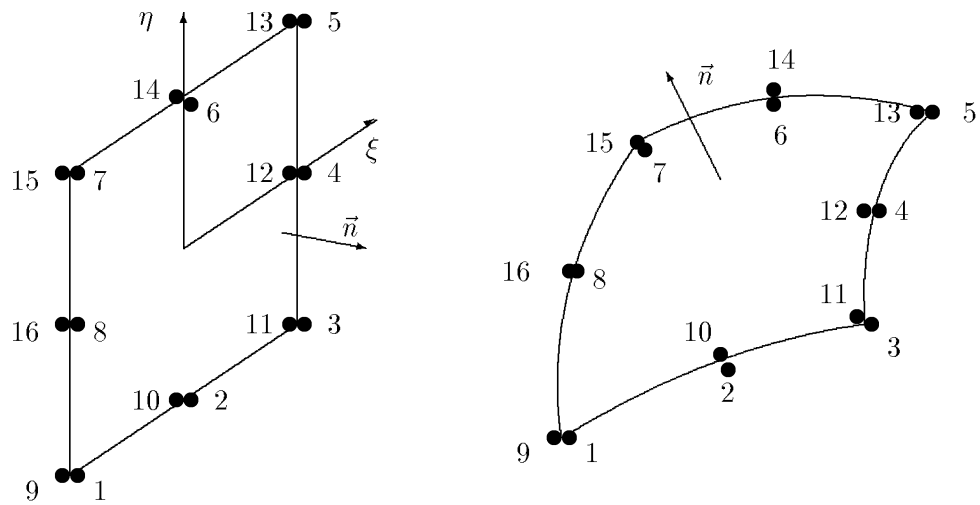
A full presentation of the coupling between a plate and an acoustic volume can be found in [7] or in [19].

3.10 ELEM92 - Coupling element, porous material, Biot model - fluid

This element is used for coupling between an acoustic fluid and a porous material described with Biot's theory. It is used for solution of the wave equation, typically with harmonic time variation assumed. This element is one of the coupling elements used in FEMAK. It needs a double set of nodes in each point at the interface between two media. One node for each medium [3]. 2-dimensional, 3-dimensional and axisymmetrical problems may be solved. Second order shape functions are used. 2x3 nodes for the 2-dimensional and axisymmetrical case and 2x8 nodes for the 3-dimensional case. See figure below.



Reference element. Nodenumbering 2-dimensional version. Two examples on global elements. The nodes in the two materials are marked with separate circles, although they should have the same coordinates.



Reference element. Nodenumbering 3-dimensional version. Two examples on global elements. The nodes int the two materials are marked with separate circles, although they should have the same coordinates.

Note that the first nodes always are in the porous material. These nodes have 4 DOF each in 2D and axisymmetric case and 6 DOF each in 3D case. The last nodes have 1 DOF per node, the acoustic pressure, p.

[3] When using external preprocessors it may difficult to define elements without any volume(3D), area(2D, axi). Use a very thin element (very small compared to the wavelength in the two media).

Dimension	DOF	Comment
2D porous mat.	u_x u_y w_x w_y	Global displacement x-axis Global displacement y-axis Global weighted relative displacement x-axis Global weighted relative displacement y-axis
2D, acoustic	p	Acoustic pressure
2D axisym. porous mat.	u_z u_r w_z w_r	Global displacement z-axis Global displacement r-axis Global weighted relative displacement z-axis Global weighted relative displacement r-axis
2D, axisym acoustic	p	Acoustic pressure
3D porous mat.	u_x u_y u_z w_x w_y w_z	Global displacement x-axis Global displacement y-axis Global displacement z-axis Global weighted relative displacement x-axis Global weighted relative displacement y-axis Global weighted relative displacement z-axis
3D, acoustic	p	Acoustic pressure

It is important to ensure a consistent definition of the normal at the boundary between the two media. Preferably the normal should point into the porous material.
The element matrices are nonsymmetric and have real coefficients.

3.10.1 ELEM92 - Material parameters

The fluid is described by two parameters.

1. Density of the fluid.
2. Wave velocity of the fluid.

These are the same parameters as the fluid, in the acoustic part or in the pores. Therefore both the group of material parameters describing the fluid or the porous material may be used.

3.10.2 ELEM92 - Available source options

None.

3.10.3 ELEM92 - Available impedance options

None.

3.10.4 ELEM92 - Use of the element

Here the use of this coupling element described briefly. Both elements type 3 and 5 must be precent when using this element.
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the acoustic fluid (fluids). (PREL)
3. Define the prescribe DOF's. (COND)

4. Define nodal connectivity of element. (ELEM)
5. Assemble and solve equations. (DYNM)

All points are mandatory.

Example: Kundt's tube

Here some details of an example is shown. The example is carried out in more detail in the appendix. The example is a calculation on a Kundt's tube. The reason for the choice is the possibility to compare with analytical models (plane wave assumptions).
The geometry is shown below. The tube is axisymmetrical, radius is 0.1 m, the air- column is 0.30 m, while the thickness of the porous material is 0.1 m. Material parameter of the porous material is,

- density of the porous material, $\rho_1 = 35.0 kg/m^3$, VPREE(6),
- shear modulus $G = 10^5 + j10^3$ Pa (VPREE(7)=1.0e5, VPREE(8)=1.0e3),
- Poissons rations 0.3 (VPREE(9)=0.3, VPREE(10)=0.0),
- porosity: $\phi = 0.9$ (VPREE(11)=0.9),
- structure factor: $k_s = 1.5$, (VPREE(12)=1.5),
- flow resisitivy: $\Phi = 10000 Ns/m^4$ (VPREE(13)=10000),
- characteristic length vicous loss $\Lambda = 2.0 \cdot 10^{-4}$ m,VPREE(14)=2.0e-4,
- characteristic length thermal loss $\Lambda' = 3.0 \cdot 10^{-4}$ m VPREE(15)=3.0e-4 .

Here only the material definition block of the input file and the definition of the coupling element is shown:

```
.
.
PREL                                  ; Definition of material properties
2,15
1,1.205,343.4                         ; material 1 - air
2,1.0,1.205,343.4,10000.0,0.90,1.5    ; material 2 - porous material
0
.
.
```

3.10.5 ELEM92 - Theory of element

Equation of continuity

Assume that the porous material is surrounded by a fluid. The fluid is the same as in the pores. The boundary conditions will be, Allard [8] p.138-139.
Resultant force equal to zero on the boundary:

$$\begin{aligned} p^a &+ \sigma_n^s + \sigma_n^f = 0 \\ p^a &+ \sigma_n = 0 \end{aligned} \tag{3.10-1}$$

Where p^a is the acoustic pressure in the fluid, and σ_n is the normal component of the stress. From continuity of pressure:

$$p^a = -\pi \tag{3.10-2}$$

Conservation of volume of fluid and frame:

$$\begin{aligned} \mathbf{u}^a &= \phi \mathbf{u}^f + (1-\phi)\mathbf{u}^s \\ \mathbf{u}^a &= \mathbf{u} + \mathbf{w} \end{aligned} \tag{3.10-3}$$

Finite element formulation

The coupling between the two media is obtained, using the continuity conditions, and Galerkin's method.
In the fluid the finite element formulation is:

$$\left[\mathbf{k_a} - \omega^2 \mathbf{m_a} + j\omega \mathbf{c_a}\right] [\overline{\mathbf{p}}] = -j\omega \mathbf{s_a} \overline{\mathbf{v}} \tag{3.10-4}$$

See e.g. Petyt [7], p.77-81, or the section describing ELEM03.
$\mathbf{v}$ is the normal velocity at the boundary may be exchanged using the continuity conditions $\mathbf{v} = j\omega \mathbf{u^a} = j\omega[\mathbf{u} + \mathbf{w}]$. Which gives a coupling matrix:

$$\mathbf{C_1} = -\omega^2 \begin{bmatrix} \mathbf{c_{1u}} & \mathbf{c_{1w}} \end{bmatrix} \begin{bmatrix} \overline{\mathbf{u}} \\ \overline{\mathbf{w}} \end{bmatrix} \tag{3.10-5}$$

In the porous material the finite element formulation is, i.e. [16], and in the section describing ELEM05:

$$\left[\begin{bmatrix} \mathbf{k_{uu}} & \mathbf{k_{uw}} \\ \mathbf{k_{wu}} & \mathbf{k_{ww}} \end{bmatrix} + j\omega \begin{bmatrix} \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{c_{ww}} \end{bmatrix} - \omega^2 \begin{bmatrix} \mathbf{m_{uu}} & \mathbf{m_{uw}} \\ \mathbf{m_{wu}} & \mathbf{m_{ww}} \end{bmatrix} \right] \begin{bmatrix} \overline{\mathbf{u}} \\ \overline{\mathbf{w}} \end{bmatrix} = \begin{bmatrix} \mathbf{f_u} \\ \mathbf{f_w} \end{bmatrix} \tag{3.10-6}$$

Here the continuity conditon: $p^a = \sigma$ and $p^a = -\pi$ can be introduced to the forcing terms $\mathbf{f_u}$ and $\mathbf{f_w}$. Which gives a coupling matrix:

$$\mathbf{C_2} = \begin{bmatrix} \mathbf{c_{2u}} \\ \mathbf{c_{2w}} \end{bmatrix} \begin{bmatrix} \overline{\mathbf{p}} \end{bmatrix} \tag{3.10-7}$$

Where:

$$\mathbf{c_{1u}} = \mathbf{c_{1w}} = \int_{\Gamma_{ab}} \mathbf{N^T} \rho_f \mathbf{N} \cdot \vec{n} d\Gamma \tag{3.10-8}$$

$$\mathbf{c_{2u}} = \mathbf{c_{2w}} = \int_{\Gamma_{ab}} \mathbf{N^T} \mathbf{N} \cdot \vec{n} d\Gamma \tag{3.10-9}$$

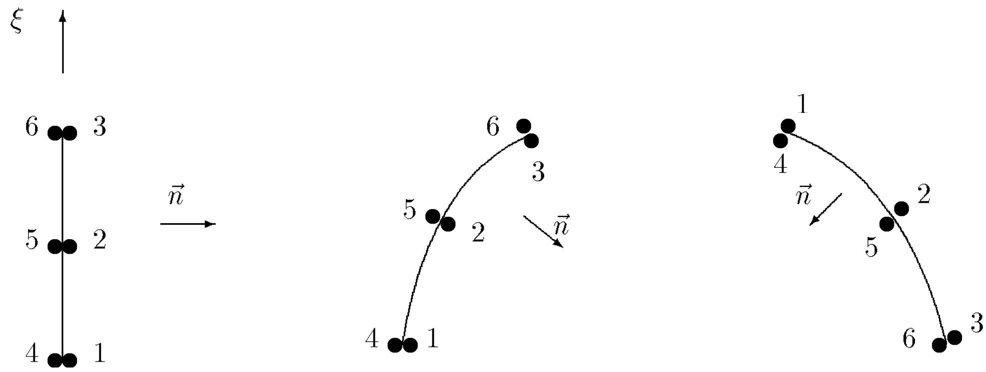
Giving a total set of equations for the elements at the interface between fluid and porous material (note that only $\mathbf{c_{1u}}$, $\mathbf{c_{1w}}$, $\mathbf{c_{2u}}$, and $\mathbf{c_{2w}}$ are contributions from ELEM92):

$$\left[\begin{bmatrix} \mathbf{k_{uu}} & \mathbf{k_{uw}} & \mathbf{c_{2u}} \\ \mathbf{k_{wu}} & \mathbf{k_{ww}} & \mathbf{c_{2w}} \\ 0 & 0 & \mathbf{k_a} \end{bmatrix} + j\omega \begin{bmatrix} \mathbf{0} & \mathbf{0} & 0 \\ \mathbf{0} & \mathbf{c_{ww}} & 0 \\ 0 & 0 & \mathbf{c_a} \end{bmatrix} - \omega^2 \begin{bmatrix} \mathbf{m_{uu}} & \mathbf{m_{uw}} & 0 \\ \mathbf{m_{wu}} & \mathbf{m_{ww}} & 0 \\ \mathbf{c_{1u}} & \mathbf{c_{1w}} & \mathbf{m_a} \end{bmatrix} \right] \begin{bmatrix} \overline{\mathbf{u}} \\ \overline{\mathbf{w}} \\ \overline{\mathbf{p}} \end{bmatrix} = \begin{bmatrix} \mathbf{f_u} \\ \mathbf{f_w} \\ \mathbf{s_a} \end{bmatrix} \tag{3.10-10}$$

A problem with this equation is that the matrices are not symmetric. This means higher storage requirements and also more computer time to assemble the matrices and solve the equations than compared to an equation with symmetric matrices.

3.11 ELEM93 - Coupling element, elastic solid - fluid

This element is used for coupling between an acoustic fluid and elastic solid with or without loss. It is used for solution of the wave equation, typically with harmonic time variation assumed. This element is one of the coupling elements used in FEMAK. It needs a double set of nodes in each point at the interface between two media. One node for each medium [4]. Only 2-D and axisymmetrical problems may be solved. Second order shape functions are used. 2x3 nodes for the 2-dimensional and axisymmetrical case.



Reference element. Nodenumbering 2-dimensional version. Two examples on global elements. The nodes in the two materials are marked with separate circles, although they should have the same coordinates.
Note that the first nodes always are in the elastic solid. These nodes have 2 DOFs for each node in 2D and axisymmetric case; the global displacements, The last nodes have 1 DOF per node, the acoustic pressure, p.

Dimension	DOF	Comment
2D elastic mat.	u_x u_y	Global displacement x-axis Global displacement y-axis
2D, acoustic	p	Acoustic pressure
2D axisym. elastic mat.	u_z u_r	Global displacement z-axis Global displacement r-axis
2D, axisym acoustic	p	Acoustic pressure

It is important to ensure a consistent definition of the normal at the boundary between the two media. Preferably the normal should point into the elastic solid.
The element matrices are nonsymmetric and have real coeffients.

3.11.1 ELEM93 - Material parameters

The fluid is described by two parameters.

1. Density of the fluid.

2. Wave velocity of the fluid.

These are the same parameters as the fluid. Therefore both the group of material paramters describing the fluid may be used.

[4]When using external preprocessors it may difficult to define elements without any volume(3D), area(2D, axi). Use a very thin element (very small compared to the wavelength in the two media).

3.11.2 ELEM93 - Available source options

None.

3.11.3 ELEM93 - Available impedance options

None.

3.11.4 ELEM93 - Use of the element

Here the use of this coupling element described briefly. Both elements type 3 and 20(or 2) must be precent when using this element.
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the acoustic fluid (fluids). (PREL)
3. Define the prescribe DOF's. That is known pressure.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Assemble and solve equations. (DYNM)

All points are mandatory

Example: Simple layered material

Here some details on an example on a layered material is shown. The layered material is two aluminum plates of thickness 1 cm. Between the plates is a layer of porous material of thickness 20 cm. The material is put in a duct with air on both sides.
Here the coupling between the air, the plate and also the porous material is shown. Numbers in circles are element-numbers.

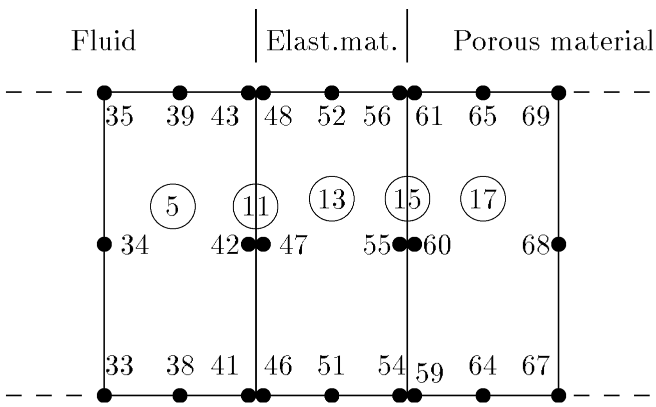
Element 5 is of type 3 (acoustic element)

Element 11 is of type 93 (coupling between elastic solid and acoustic element)

Element 13 is of type 20 (elastic solid with loss)

Element 15 is of type 94 (coupling between porous material and elastic solid)

Element 17 is of type 5 (porous material, Biot model)



This model use three groups of materials, one for the porous material, one for the air, and one for the elastic solid. Below the definition of the material parameters and definition of two elements of type 93 are shown.
More details and results about this model can be found in the example, layer1 in the appendix.

```
.
.
PREL
3,15                                        ; NGPE=3, NPRE=15
1,1.205,340.9,0.71,1.84e-5,1.4,30.0,1.80e5  ;--
1.80e4,0.40,0.0,0.93,3.2,10000.,1.0e-4      ;  | Porous material
1.0e-4                                      ;--
2,1.205,340.9,0.0,0.0,0.0,0.0,0.0           ;--
0.0,0.0,0.0,0.0,0.0,0.0,0.0                 ;  |    Air
0.0                                         ;--
3,7.1E10,0.3,2700.0,0.0,0.0,0.0,0.0         ;--
0.0,0.0,0.0,0.0,0.0,0.0,0.0                 ;  | Elastic solid
0.0                                         ;--
0
.
.
ELEM
40,28,12,8,5,,1                             ;NELT=40,NELTR=28,NELTC=12
.                                           ;NNEL=8,NTPE=5,NSYM=5
.
11,2,2,93,2,1,46,47,48,41,42,43             ; Generate element no.11 and 12 (type 93)
.
.
```

3.11.5 ELEM93 - Theory of element

Equation of continuity

Assume that the elastic material is surrounded by a fluid.
Resultant force equal to zero on the boundary:

$$p^a + \sigma_n = 0 \tag{3.11-1}$$

Where p^a is the acoustic pressure in the fluid, and σ_n is the normal component of the stress.
Conservation of volume of fluid and frame:

$$\mathbf{u}^a \quad = \quad \mathbf{u} \tag{3.11-2}$$

Finite element formulation

The coupling between the two media is obtained, using the continuity conditions, and Galerkin's method.
In the fluid the finite element formulation is:

$$\left[\mathbf{k_a} - \omega^2\mathbf{m_a} + j\omega\mathbf{c_a}\right][\overline{\mathbf{p}}] = -j\omega\mathbf{s_a}\overline{\mathbf{v}} \tag{3.11-3}$$

See e.g. Petyt [7], p.77-81 or the section describing ELEM03:
$\mathbf{v}$ is the normal velocity at the boundary may be exchanged using the continuity conditions $\mathbf{v} = j\omega\mathbf{u^a} = j\omega\mathbf{u}$. Which gives a coupling matrix:

$$\mathbf{C_1} = -\omega^2\mathbf{c_1}\overline{\mathbf{u}} \tag{3.11-4}$$

In the elastic material the finite element formulation is,

$$[\mathbf{k_u} - \omega^2 \mathbf{m_u}]\overline{\mathbf{u}} = \mathbf{f_u} \tag{3.11-5}$$

see e.g. the section describing ELEM20:
Here the continuity conditon: $p^a = \sigma$ can be introduced to the forcing terms $\mathbf{f_u}$. Which gives a coupling matrix:

$$\mathbf{C_2} = \mathbf{c_2}\overline{\mathbf{p}} \tag{3.11-6}$$

Where:

$$\mathbf{c_1} = \int_{\Gamma_{ab}} \mathbf{N^T} \rho_f \mathbf{N} \cdot \vec{n} d\Gamma \tag{3.11-7}$$

$$\mathbf{c_2} = \int_{\Gamma_{ab}} \mathbf{N^T} \mathbf{N} \cdot \vec{n} d\Gamma \tag{3.11-8}$$

Giving a total set of equations for the elements at the interface between fluid and elastic material (note that only $\mathbf{c_1}$, and $\mathbf{c_2}$ are contributions from ELEM93):

$$\left[\begin{bmatrix} \mathbf{k_u} & \mathbf{c_2} \\ 0 & \mathbf{k_a} \end{bmatrix} + j\omega \begin{bmatrix} \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{c_a} \end{bmatrix} - \omega^2 \begin{bmatrix} \mathbf{m_u} & \mathbf{0} \\ \mathbf{c_1} & \mathbf{m_a} \end{bmatrix} \right] \begin{bmatrix} \overline{\mathbf{u}} \\ \overline{\mathbf{p}} \end{bmatrix} = \begin{bmatrix} \mathbf{f_u} \\ \mathbf{f_w} \\ \mathbf{s_a} \end{bmatrix} \tag{3.11-9}$$

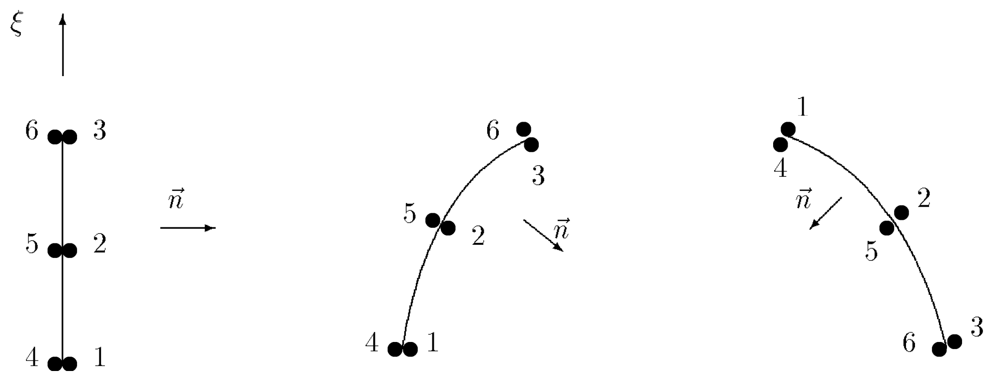
A problem with this equation is that the matrices are not symmetric. This means higher storage requirements and also more computer time to assemble the matrices and solve the equations than compared to an equation with symmetric matrices.

List of examples in appendix.

1. layer1 - Layered material, aluminum, elastic porous material.

3.12 ELEM94 - Coupling element, elastic solid - porous material

This element is used for coupling between a porous material using Biot's theory and an elastic solid with loss. It is used for solution of wave problems, typically with harmonic time variation and coupled to acoustic fluids. This element is one of the coupling elements used in FEMAK. It needs a double set of nodes in each point at the interface between two media. One node for each medium. This element use a very simple penalty method for continuity of displacements. We ask the users to look for and report trouble with this scheme. [5] Only 2-D and axisymmetrical problems may be solved. Second order shape functions are used. 2x3 nodes for the 2-dimensional and axisymmetrical case.



Reference element. Nodenumbering 2-dimensional version. Two examples on global elements. The nodes in the two materials are marked with separate circles, although they should have the same coordinates.
Note that the first nodes always are in the porous material. These nodes have 4 DOFs for each node in 2D and axisymmetric case. The last nodes have 2 DOF per node, the displacements in the elastic solid.

Dimension	DOF	Comment
2D	u_x	Global displacement x-axis
porous mat.	u_y	Global displacement y-axis
	w_x	Global weighted relative displacement x-axis
	w_y	Global weighted relative displacement y-axis
2D	u_x	Global displacement x-axis
elastic mat.	u_y	Global displacement y-axis
2D axisym.	u_z	Global displacement z-axis
porous mat.	u_r	Global displacement r-axis
	w_z	Global weighted relative displacement z-axis
	w_r	Global weighted relative displacement r-axis
2D axisym.	u_z	Global displacement z-axis
elastic mat.	u_r	Global displacement r-axis

It is important to ensure a consistent definition of the normal at the boundary between the two media. Preferably the normal should point into the porous material.
The element matrices are symmetric and have real coeffients.

[5] When using external preprocessors it may difficult to define elements without any volume(3D), area(2D, axi). Use a very thin element (very small compared to the wavelength in the two media).

3.12.1 ELEM94 - Material parameters

The elastic material is described by one parameter.

1. Young's modulus

Which is the first parameter for the elastic solid. Therefore the group of material paramteres describing the solid may be used.

3.12.2 ELEM94 - Available source options

None.

3.12.3 ELEM94 - Available impedance options

None.

3.12.4 ELEM94 - Use of the element

Here the use of this coupling element is described briefly. Both elements type 5 and 20 must be precent when using this element.
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the elastic solid. (PREL)
3. Define the prescribe DOF's. That is known pressure.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Assemble and solve equations. (DYNM)

All points are mandatory.

Example: Simple layered material

Here some details on an example on a layered material is shown. The layered material is two aluminum plates of thickness 1 cm. Between the plates is a layer of porous material of thickness 20 cm. The material is put in a duct with air on both sides.
Here the coupling between the air, the plate and also the porous material is shown. Numbers in circles are element-numbers.

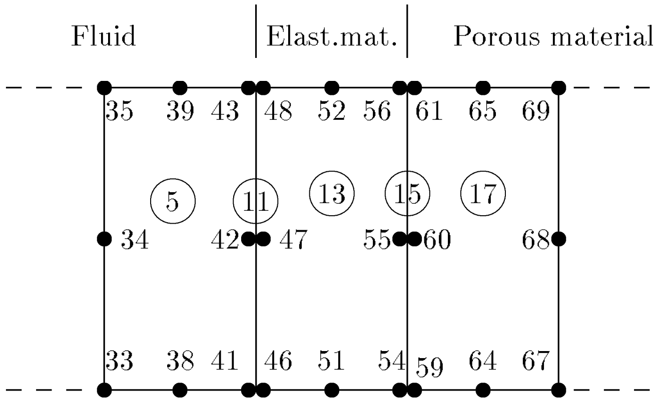
Element 5 is of type 3 (acoustic element)

Element 11 is of type 93 (coupling between elastic solid and acoustic element)

Element 13 is of type 20 (elastic solid with loss)

Element 15 is of type 94 (coupling between porous material and elastic solid)

Element 17 is of type 5 (porous material, Biot model)



This model use three groups of materials, one for the porous material, one for the air, and one for the elastic solid. Below the definition of the material parameters and definition of two elements of type 94 are shown.
More details and results about this model can be found in the example, layer1 in the appendix.

```
.
.
PREL
3,15                                          ; NGPE=3, NPRE=15
1,1.205,340.9,0.71,1.84e-5,1.4,30.0,1.80e5    ;--
1.80e4,0.40,0.0,0.93,3.2,10000.,1.0e-4        ;  | Porous material
1.0e-4                                        ;--
2,1.205,340.9,0.0,0.0,0.0,0.0,0.0             ;--
0.0,0.0,0.0,0.0,0.0,0.0,0.0                   ;  |    Air
0.0                                           ;--
3,7.1E10,0.3,2700.0,0.0,0.0,0.0,0.0           ;--
0.0,0.0,0.0,0.0,0.0,0.0,0.0                   ;  | Elastic solid
0.0                                           ;--
0
.
.
ELEM
40,28,12,8,5,,1                               ;NELT=40,NELTR=28,NELTC=12
.                                             ;NNEL=8,NTPE=5,NSYM=5
.
15,2,2,94,3,1,59,60,61,54,55,56               ; Generate element no.15 and 16 (type 94)
.
.
```

3.12.5 ELEM94 - Theory of element

Equations of continuity

We assume here that the porous material is glued to the elastic solid.
Resultant force equal to zero on the boundary:

$$\begin{aligned} \sigma^e + \sigma^s + \sigma^f &= 0 \\ \sigma^e + \sigma &= 0 \end{aligned} \qquad (3.12\text{-}1)$$

Where σ^e is the stress in the elastic solid, σ^s is the the stress in the elastic frame of the porous material, σ^f is the the pressure in the fluid of the porous material.
Continuity of displacements:

$$\mathbf{u}^e = \mathbf{u}^s \qquad (3.12\text{-}2)$$

$$\mathbf{u}^e \cdot \mathbf{n} = \mathbf{u}^s \cdot \mathbf{n} \qquad (3.12\text{-}3)$$

$$\mathbf{w} \cdot \mathbf{n} = 0 \qquad (3.12\text{-}4)$$

Where $\mathbf{u}^e$ is the displacement of the elastic solid, $\mathbf{u}^s$ us the displacement of the frame of the porous material, and $\mathbf{w}$ is the weighted relative displacement of the porous material. The last equations means that the frame of the porous material will be identical to the displacement of the elastic solid. The fluid will have the same displacement as the frame normal to interface between the two materials, while the displacement of the fluid is free in the direction tangential to the interface.

Finite element formulation

This coupling element introduce extra continuity equations for the global set of equations using a penalty function and functional, as described by Zienkiewicz, [6], ch.9.14 [6].

$$\bar{\Pi} = \alpha \int C^T(u)C(u)\, d\Omega \tag{3.12-5}$$

The functions C(u)=0 are the continuity equations.

$$C = \begin{bmatrix} N\bar{\mathbf{u}^s} - N\bar{\mathbf{u}^e} \\ \vec{n}N\bar{\mathbf{w}} \end{bmatrix} = \begin{bmatrix} N & 0 & 0 & 0 & -N & 0 \\ 0 & N & 0 & 0 & 0 & -N \\ 0 & 0 & n_x N & n_y N & 0 & 0 \end{bmatrix} \begin{bmatrix} u_x^s \\ u_y^s \\ w_x \\ w_y \\ u_x^e \\ u_y^e \end{bmatrix} \tag{3.12-6}$$

$$C_i^T \cdot C_j = \mathbf{u^T} \begin{bmatrix} N_iN_j & 0 & 0 & 0 & -N_iN_j & 0 \\ 0 & N_iN_j & 0 & 0 & 0 & -N_iN_j \\ 0 & 0 & n_{xi}n_{xj}N_iN_j & n_{xi}n_{yj}N_iN_j & 0 & 0 \\ 0 & 0 & n_{xj}n_{yi}N_iN_j & n_{yi}n_{yj}N_iN_j & 0 & 0 \\ -N_iN_j & 0 & 0 & 0 & N_iN_j & 0 \\ 0 & -N_iN_j & 0 & 0 & 0 & N_iN_j \end{bmatrix} \mathbf{u} \tag{3.12-7}$$

Where

$$\mathbf{u} = \begin{bmatrix} u_x^s \\ u_y^s \\ w_x \\ w_y \\ u_x^e \\ u_y^e \end{bmatrix} \tag{3.12-8}$$

Instead of carrying out the integrations over the elements, we utilize that N_i is 1 in node no.i and zero in the other and demand continuity in the nodes.
The penalty number α is set to $VPREE(1) \cdot 10^7$.
Variation of $\bar{\Pi}$ gives the additional equations to the global set of equations:

$$\mathbf{k} = 2\alpha \begin{bmatrix} N_iN_j & 0 & 0 & 0 & -N_iN_j & 0 \\ 0 & N_iN_j & 0 & 0 & 0 & -N_iN_j \\ 0 & 0 & n_{xi}n_{xj}N_iN_j & n_{xi}n_{yj}N_iN_j & 0 & 0 \\ 0 & 0 & n_{xj}n_{yi}N_iN_j & n_{yi}n_{yj}N_iN_j & 0 & 0 \\ -N_iN_j & 0 & 0 & 0 & 0 & N_iN_j \\ 0 & -N_iN_j & 0 & 0 & 0 & N_iN_j \end{bmatrix} \mathbf{u} \tag{3.12-9}$$

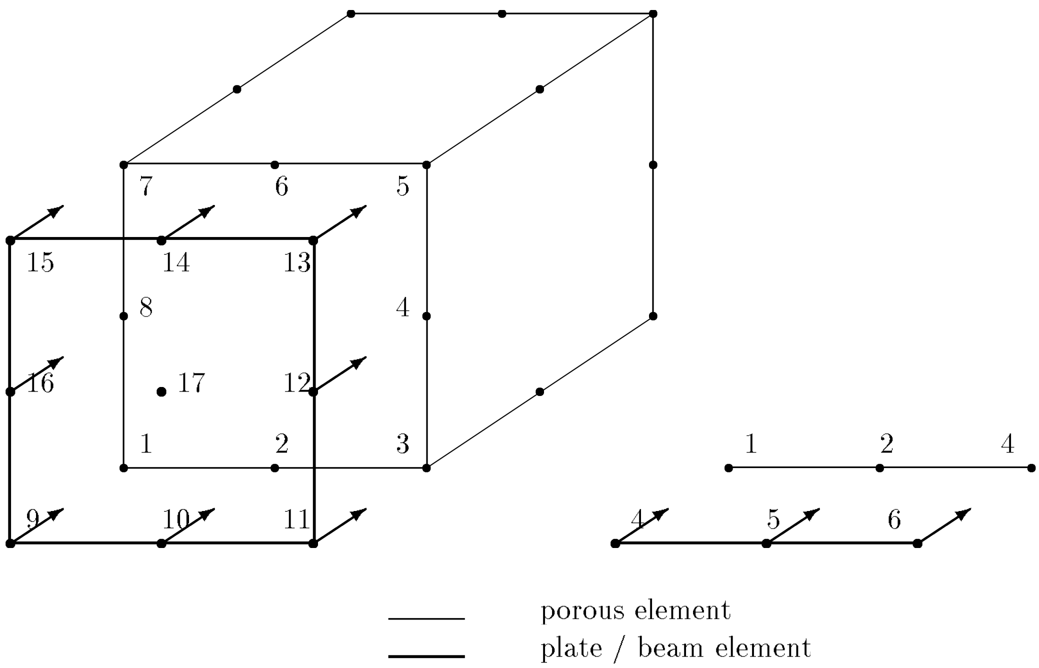
[6]Note that we have not introduced any continuity on the stresses.

List of examples in appendix.

1. layer1 - Layered material, aluminum, elastic porous material.

3.13 ELEM95 - Coupling element, plate/beam - porous material

This element is to be used when a plate (or beam in 2D) element ELEM08 and a porous element ELEM05 are superposed. The junction between those two elements is performed by the introduction of a virtual coupling element, which insures the continuity of displacements in all direction between the plate and the frame of the porous material, and the continuity of normal displacements only between the plate and the fluid pore. The coupling element is defined by two sets of nodes at the interface: first, the 8 nodes of the porous material which are at the interface, then the 9 nodes of the plate (in 2D, 3 nodes for the beam, 3 nodes for the porous material). The orientation of the numbering of the "plate part" of the element must follow the same than the orientation of the numbering of the "porous part" of the element, as shown in the figure below.



Nodes 1,2,...,8 and nodes 9,11,...,16 have respectively the same coordinates. In 2D, it would be nodes 1,2,3 and nodes 4,5,6.
The element has six DOF (4 in 2D) per porous nodes (defined as in ELEM05) and three DOF (2 in 2D) per plate node (one deflection, and two rotations given in the local coordinate system defined when the element is writen as an ELEM08 element in FEMIN.DAT).

Dimension	DOF	Comment
2D, beam	w	Normal displacement along beam-axis II
	θ	Rotation about beam-axis III
2D, porous	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
	w_x	Global weighted relative displacement x-axis
	w_y	Global weighted relative displacement y-axis
3D, plate	w	Normal displacement along plate-axis III
	θ_1	Rotation about plate-axis I
	θ_2	Rotation about plate-axis II
3D, porous	u_x	Global displacement x-axis
	u_y	Global displacement y-axis
	u_z	Global displacement z-axis
	w_x	Global weighted relative displacement x-axis
	w_y	Global weighted relative displacement y-axis
	w_z	Global weighted relative displacement z-axis

The total number of DOF is 75 (50 in 2D).
The element has symmetric matrices with real coefficients.

3.13.1 ELEM95 - Material Parameters

The element has six material parameters :

1. E - Young's modulus
2. ν - Poisson's ratio
3. h - Plate thickness
4. Parameter for definition of the normal of the plate

   VPREE(4)= 1.0 normal of the beam/plate towards the x direction

   VPREE(4)=-1.0 normal of the beam/plate towards the-x direction

   VPREE(4)= 2.0 normal of the beam/plate towards the y direction

   VPREE(4)=-2.0 normal of the beam/plate towards the-y direction

   VPREE(4)= 3.0 normal of the beam/plate towards the z direction

   VPREE(4)=-3.0 normal of the beam/plate towards the-z direction

5. Parameter for first plate angle action on points at the interface

   VPREE(5)= 1.0 or 2.0 or 3.0 : a positive rotation of angle θ_1 induces a displacement of the interface nodes in x or y or z respectively

   VPREE(5)=-1.0 or-2.0 or-3.0 : a positive rotation of angle θ_1 induces a displacement of the interface nodes in-x or-y or-z respectively

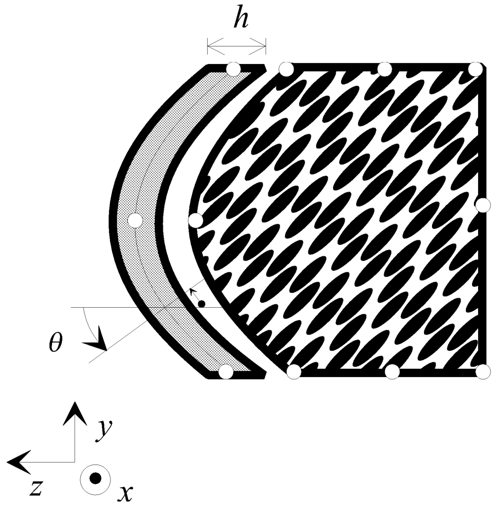
6. Parameter for the second plate angle action at the interface

   VPREE(6)= 1.0 or 2.0 or 3.0 : a positive rotation of angle θ_2 induces a displacement of the interface nodes in x or y or z respectively

   VPREE(6)=-1.0 or-2.0 or-3.0 : a positive rotation of angle θ_2 induces a displacement of the interface nodes in-x or-y or-z respectively

N.B1 : as mentioned above, the normal of the plate (or beam) as well as its rotation angles are defined as they are in ELEM08 from FEMIN.DAT. This means that no matter if in ELEM95, the numbering of the plate (or beam) nodes is different from that used to define the plate (or beam) as an ELEM08 element in FEMIN.DAT, the direction of the DOF of the plate (or beam) nodes are still in accordance with the local orientation defined from ELEM08, not ELEM95.

N.B2 : VPREE(5) and VPREE(6) may be best explained by the figure below. The continuity of displacement has to be satisfied *at the interface*. However, the plate nodes (circles at the middle line of the beam in the figure) are in the middle plane of the plate, not at the interface. That is the reason why we have to get information on the positioning of the porous element w.r.t the plate. If the porous element is on the right or on the left of the plate, the normal displacement at the interface remains unchanged, but the transverse displacements at the interface are opposite. The figure below gives an example in 2D on how to use VPREE(5) (and VPREE(6) in 3D). A positive rotation of θ (see ELEM08 to understand what a positive angle means) induces a transverse displacement at the interface in the direction +y (the black round in the figure is moving upward because of the shrinking of the beam at its lower part). Therefore VPREE(5)=2.0.



3.13.2 ELEM95 - Available source options

None

3.13.3 ELEM95 - Available impedance options

None

3.13.4 ELEM95 - Use of the element

Here the use of this coupling element is briefly described. Both elements type 3 and type 8 must be present when using this element.

A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)

2. Define the density of the acoustic medium. (PREL)

3. Define the nodal connectivity of the element. (ELEM)

All points are mandatory.

3.13.5 ELEM95 - example of use

3.13.6 ELEM95 - Theory of the element

As mentioned earlier, the coupling element between the plate and the porous element has to insure the continuity of displacement at the interface. We can write the continuity requirements as :

$$\mathbf{C}(\mathbf{u}) = \mathbf{0}$$

$\mathbf{u}$ is a vector with coordinates representing the frame displacements, the relative fluid-frame displacements, and the plate deflection and rotations. This continuity requirement may be taken into account in the variational principal governing the coupled system by adding an other functional Π defined as :

$$\Pi = \alpha/2 \int_{interface} \mathbf{C}(\mathbf{u}).\mathbf{C}(\mathbf{u})$$

For a very large value of α, the continuity requirement will be satisfied.
A full presentation of the penalty function theory can be found in [6]

3.14 IMP100 - Coupling procedure, circular hole in baffle

This is not an ordinary element but rather a procedure for coupling of acoustic waves between an axisymmetrical volume and a semi-infinite domain, through a hole in an infinite baffle. The volume must have rigid walls close to the opening.
The number of nodes in the circular hole depends on the mesh, so the user must specify the number of nodes for this "element". See example for description of geometry.
The nodes in the elements has only one degree of freedom, the acoustic pressure.

Dimension	DOF	Comment
2D axisym.	p	acoustic pressure

The element matrices are symmetric and have real coefficients. It may be coupled to other type of elements

- Acoustic element(ELEM03)

The procedure results in a impedance type matrix, but this matrix is not connected to the common type elements in the circular hole, but rather to all the nodes in the hole in the baffle. Therefore, a new element is defined, containing all the nodes in the hole. This element will not contribute to the system-matrices as all the usual element, but only give an impedance type contribution. This element must be defined in the ELEM-block, and used through reference in the CIMP- block.
The element is described by Kagawa et.al.[5]. The idea of the element is as follows: the field in the semi-infinite part is described using the Rayleigh's integral, while the field in the closed domain is described using a modal decomposition. The two descriptions are matched in the hole. This is done in such a manner that the contributions to the matrices are symmetrical and use no other degrees of freedom than the acoustical pressure in the nodes. See theory-section.
The user must supply the number of nodes,n_{100}, the number of modes used,m_{100}, and the radius of the hole. We have found that $m_{100} \leq (n_{100} - 1)/2$. [7] To find the pressure in external points, the user must input the coordinates of these points and set VELPAR to 100 in the DYNM-block .

3.14.1 IMP100 - Material parameters

The fluid is described by two parameters.

1. Density of the fluid.
2. Wave velocity of the fluid.

The number of modes used in the description of the wave in the volume must be specifyed in the CIMP-block.

3.14.2 IMP100 - Use of the element

Here the use of this coupling procedure is described briefly.
A description of typical use of the element is as follows:

1. Define the grid by the coordinates of each node. (COOR)
2. Define the material properties of the acoustic fluid (fluids). (PREL)

[7] There has been some problems in the use of this element when the mesh has been made with PATRAN. Be sure that the normal of the acoustic fluid elements points outwards.

3. Define the prescribe DOF's. That is known pressure.(COND)
4. Define nodal connectivity of element. (ELEM)
5. Define impedance type 100. (CIMP)
6. Assemble, solve equations, and calculate external pressure. (DYNM)

All points are mandatory. See also the subsection describing the use of the CIMP-block.

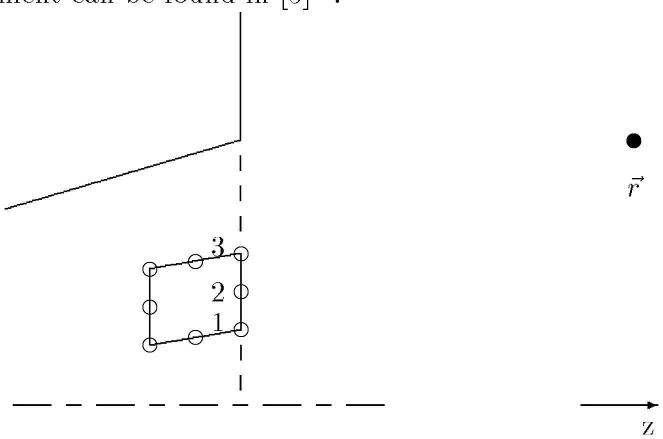
Example: Horn loudspeaker ending in an infinite baffle.

A simple horn with two conical sections in a baffle will be investigated. The throat of the horn has radius .1 m, the inner section is narrow, 36.9^o, and 0.3 m long. The outer section is wider, 90^o, and 0.3 m long. The mesh is shown below. The full input file is found in horn1.inp. here some details will be shown. Some results can be found in the appendix. Note that often the element defining IMP100 is the element with the largest number of nodes.

```
ELEM
19,19,0,11,3
.
.
19,1,1,100,1,1,61,62,63,64,65,66,67,68,69,70    ; special coupling/impedance
71                                              ; element no.100
0
CIMP
1,29,1                               ; only one impedance
1,100
3,11,0.5                             ; 3 modes, 11 nodes, rad=0.5 m
19,1,1                               ; identify the impedance-element
```

3.14.3 IMP100 - Theory of element

The details of the element can be found in [5] [8].



Circular hole in rigid baffle. Definition of nodes used in coupling element.
The pressure in inside the opening may be expressed as:

$$p_i(r,z) = \sum_{m=1}^{\infty} (A_m e^{-jk_m z} + R_m e^{jk_m z}) w_m(r) \tag{3.14-1}$$

[8]Here the acoustic pressure is used instead of the velocity potential for description of the waves. In [5] the velocity potential is used

where $k_m^2 = k^2 - (\gamma_m/a)^2$ and $w_m = J_0(\gamma_m/ar)/(\sqrt{\pi}aJ_0(\gamma_m))$ for a circular hole with no spinning mode present. A_m and R_m is the amplitude of the outgoing and the reflected waves.
The exterior pressure can be expressed:

$$p_o(\mathbf{x}) = j\rho\omega \int_{S'} v_n(\mathbf{x}')g(\mathbf{x}|\mathbf{x}')d\Gamma(\mathbf{x}') \tag{3.14-2}$$

$$g(\mathbf{x}|\mathbf{x}') = \frac{e^{-jk|\mathbf{x}-\mathbf{x}'|}}{2\pi|\mathbf{x}-\mathbf{x}'|} \tag{3.14-3}$$

Where $\mathbf{x}$ is coordinate of point in exterior domain, while $\mathbf{x}'$ is coordinate in the baffle plane.
Continuity:

$$p_i(\mathbf{x}) = p_o(\mathbf{x}) \tag{3.14-4}$$

$$\frac{\partial p_i(\mathbf{x})}{\partial z} = \frac{\partial p_o(\mathbf{x})}{\partial z} \tag{3.14-5}$$

After truncating the modal decomposition to M members, using the continuity conditions, weighting with w(r) and integrate over the opening, S, one find [5]:

$$\sum_{n=1}^{M}(A_m + R_m)\int_S w_m(\gamma)w_n(r)dS = \sum_{n=1}^{M}(A_m - R_m)jk_m\int_S w_m(r')w_n(r)g(\mathbf{x}|\mathbf{x}')dS'dS \tag{3.14-6}$$

Which may be rewritten in matrix form using w_m's orthogonality:

$$\mathbf{A} = [\mathbf{E}-\mathbf{I}]^{-1}[\mathbf{E}+\mathbf{I}]\mathbf{R} = \mathbf{TR} \tag{3.14-7}$$

$$\mathbf{E}_{n,m} = jk_m\int_S w_m(r')w_n(r)g(\mathbf{x}|\mathbf{x}')dS'dS \tag{3.14-8}$$

$$\mathbf{E}_{n,m} = 2k_m a\int_0^\infty \frac{X^3J_1(X)^2}{\mu_a(X^2-\gamma_m^2)(X^2-\gamma_n^2)}\,dX \tag{3.14-9}$$

$$\mu_a = \begin{cases} \sqrt{(ka)^2 - X^2} & , ka \geq X \\ -j\sqrt{X^2-(ka)^2} & , ka < X \end{cases} \tag{3.14-10}$$

Using Galerkin's method to introduce boundary conditions in the opening gives [9]:

$$\int_\Gamma N^T\frac{\partial p}{\partial n}\,d\Gamma = -j\int_\Gamma N^T WK[A-R]\,d\Gamma = jQK[T-I]R \tag{3.14-11}$$

Where

$$Q = \int_\Gamma N^T W\,d\Gamma \tag{3.14-12}$$

The reflection coefficients, R, can be found using:

$$\int_\Gamma W^T N\overline{p}\,d\Gamma = \int_\Gamma W^T W[A+R]\,d\Gamma = [T+I]R \tag{3.14-13}$$

$$R = [T+I]^{-1}Q^T\overline{p} \tag{3.14-14}$$

Introducing R in the boundary condition expression gives an impedance type matrix, which can be treated as an ordinary impedance-matrix in FEMAK.

$$Y\,\overline{p} = -jQK[T-I][T+I]^{-1}Q^T\overline{p} \tag{3.14-15}$$

[9] In [5] this is done using a variational procedure

In order to find the external pressure the Rayleigh integral is used. First the normal velocity in the circular hole must be calculated. It is found using

$$j\rho\omega v_n(r,z_0) = \sum_{m=1}^{M} -jk_m(A_m e^{-jk_m z} - R_m e^{jk_m z})w_m(r) \tag{3.14-16}$$

List of examples in appendix.

1. horn1 - Horn loudspeaker

Appendix A

Examples.

In this appendix some examples are collected. The examples are chosen to show the use of the different elements, the different methods and also how to couple different types of elements. In some of the examples, results are discussed. We encourage the users to support this manual with new examples. The files are put in directories under the /femak/docu directory.

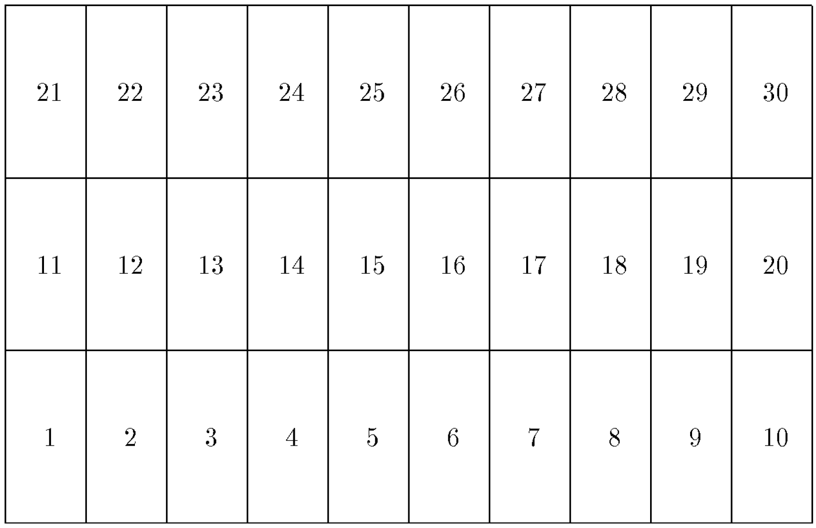
Subject	Filename	Short Description
Simple infile ELEM03	acouex1	Model of small tube, velocity source and impedance
Transmission loss ELEM03	acouex3	Duct, reactive muffler, incoming plane wave
Radiation impedance ELEM03	acouex21	Radiation impedance of a circular piston in infinite baffle using ELEM03 and impedance for termination of calculation domain.
Radiation impedance ELEM03 and ELEM11	stmpwee1 stmpwee5 stmpwee7	Radiation impedance of a circular piston in infinite baffle using ELEM03 and ELEM11 for termination of calculation domain.
Horn loudspeaker ELEM03 and Imp.100	horn1	Analysis of simple horn loudspeaker ending in baffle Axisymmetric case. Exterior pressure
Eigenvalue, elastic ELEM02	elaex11	Eigenvalue analysis of simple T-shaped steel bar
Forced vibrations ELEM20	elaex12	Forced vibrations, same geometry as elaex11
Static plane stress ELEM02	bigex2	Plane stress, static case.
Porous material equivalent fluid ELEM04 and ELEM03	porex1	Simulation of impedance/absorption measurement in Kundt's tube. Different models of the porous material is compared.
Layered material ELEM03, ELEM05 ELEM20, ELEM93 ELEM94	layer1	Layered material buildt of aluminum, elastic porous material. Air on both sides of material.
Heat conduction ELEM01	genex3 genex4	Heat conduction in perforated plate. Both static case (genex3) and transient case (genex 4).

A.1 Pure acoustic examples

Here some examples are shown on pure acoustic problems using elements ELEM03 and ELEM11.

A.1.1 acouex1 - Simple infile

In this example some simple calculations on the sound field in a straight cylinder with circular cross section are shown. The cylinder has length 1.0 m and radius 0.3m. In the left end it has a vibrating piston (v = 1.0 m/s at z=0.0) in the right hand end it is terminated in an impedance equal to 1+j (at z=1.0, the impedance is normalized with ρc). An axisymmetric model is used. The finite element has 30 elements and 117 nodes. See figure below. The input model can be found in the file acouex1.inp

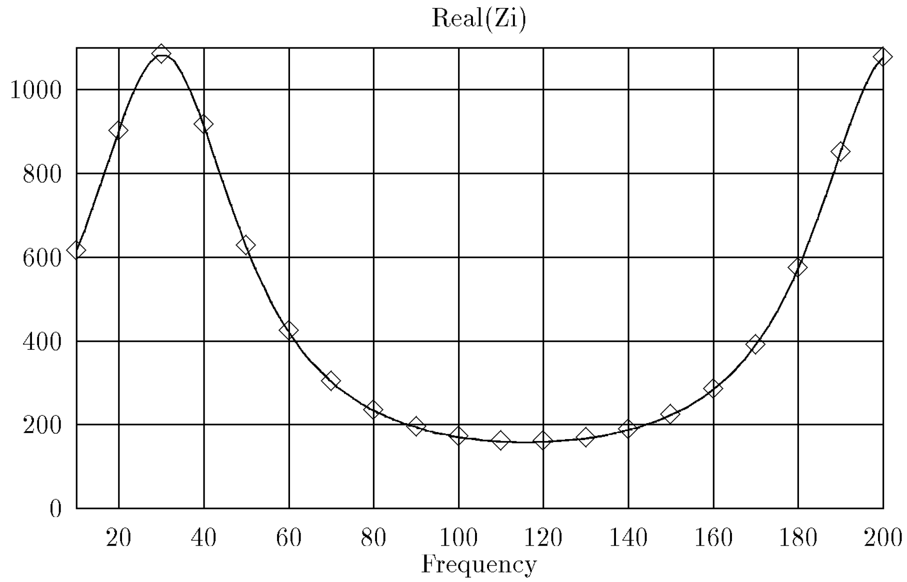


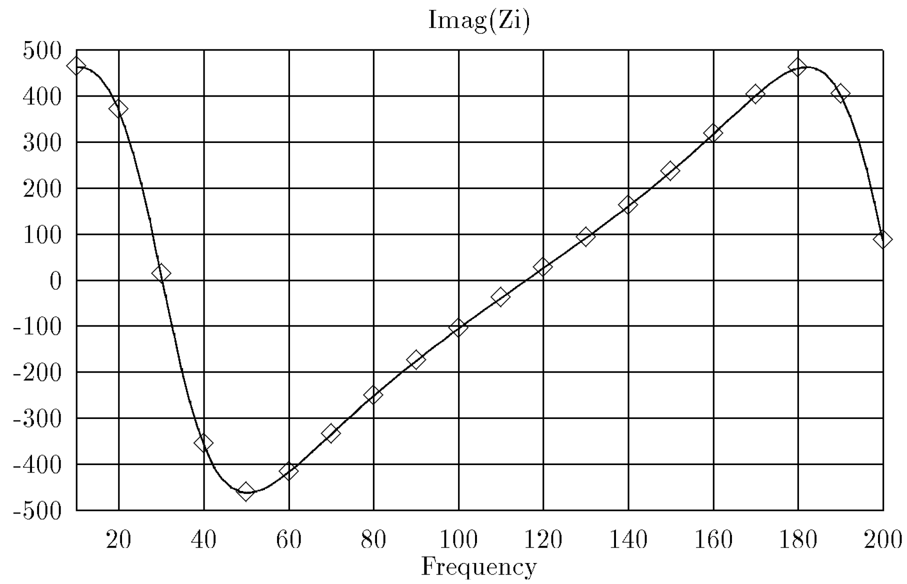
Finite element model acouex1.

The size of the elements are 0.1 m. We need at least four elements for each wavelength, in other words this model can be used for wavelengths longer than about 0.4 m. This gives an upper frequency limit at about 850 Hz. Here results for 0-200 Hz are presented.
The impedance at the vibrating piston is calculated. The pressure at the piston is the impedance because the velocity of the piston is 1.0 m/s. The analytic expression for the impedance is:

$$z_i = \rho c \frac{z_l + j\rho c \tan kl}{\rho c + j z_l \tan kl} \tag{A.1-1}$$

The below graphs of the impedance show no difference between the analytic expression and the results found with femak.



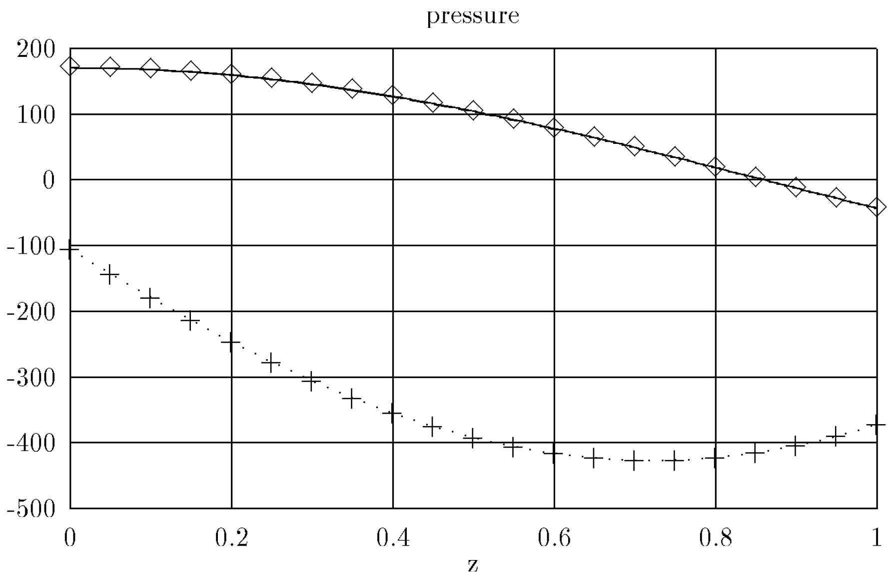


Impedance at vibrating end of tube. Solid line - analytic expression, diamonds - femak.

The pressure as a function of z at 100 Hz is shown. An anlytic expression for the pressure is

$$p(x) = \rho c(e^{jkl} \frac{(1+z)\cos kx}{\cos kl + jzsinkl} - e^{jkx}) \tag{A.1-2}$$

Also for the example the results found using femak is close to the theoretical values.



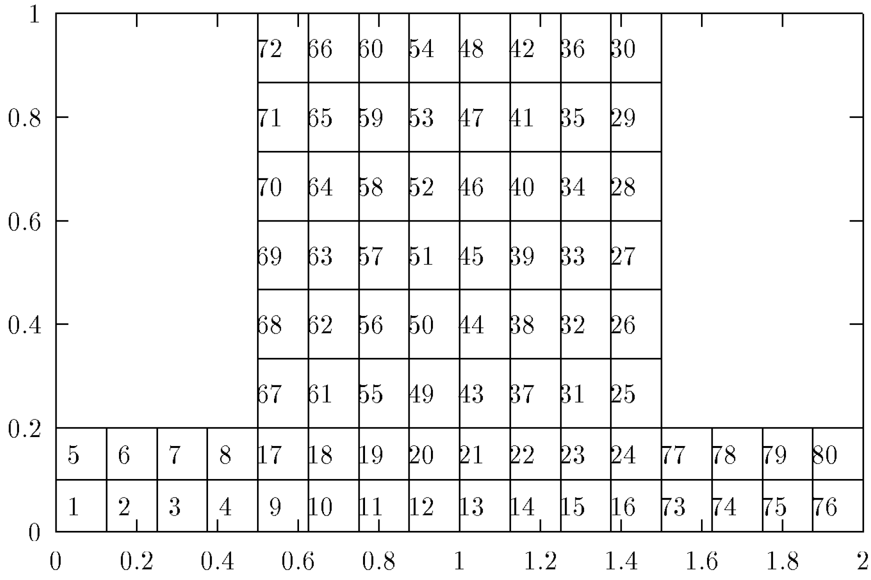
Pressure in center of tube at f=100 Hz.
Re(p): Solid line - analytic expression, diamonds - femak.
Im(p): dotted line - analytic expression, cross - femak.

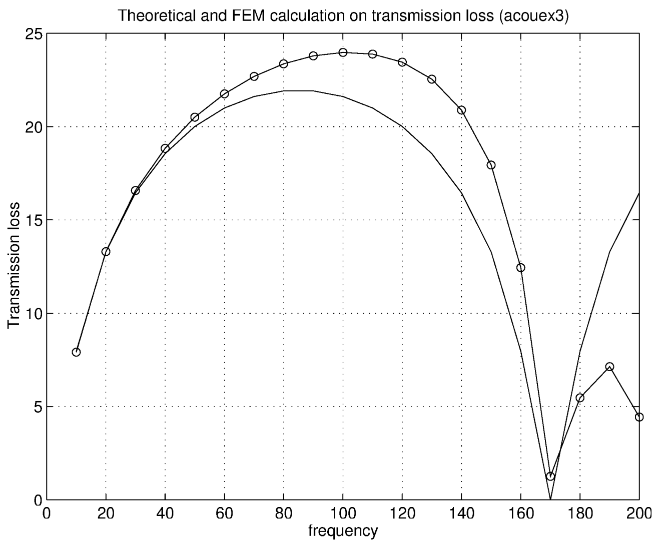
A.1.2 acouex3 - Transmission loss

In this example an incoming plane wave in a duct is used to study a reactive muffler. The muffler is a simple expansion chamber type. The muffler is 1.0 m long with a diameter of 2.0 m. The diameters of the duct is 0.4 m . The length indicates a 1/4 wavelength resonance at 85 Hz. However, the diameters of the system are large, and therefore the plane wave assumption using continuity of pressure and volume velocity over the junctions may not be adequate to describe the transmission loss.
Using plane wave assumptions the transmission loss will be [22]:

$$TL = 10log[1 + \frac{1}{4}(m - \frac{1}{m})^2 sin^2(kl)] \tag{A.1-3}$$

Where m is the ratio between the cross-sectional area of the muffler and that of the duct.

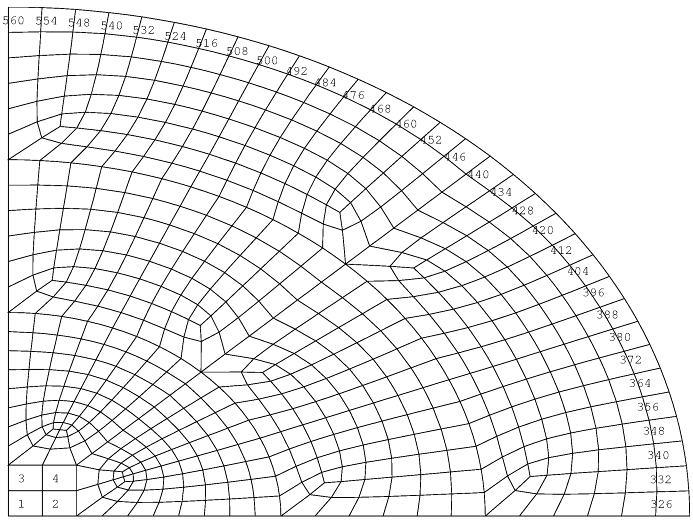




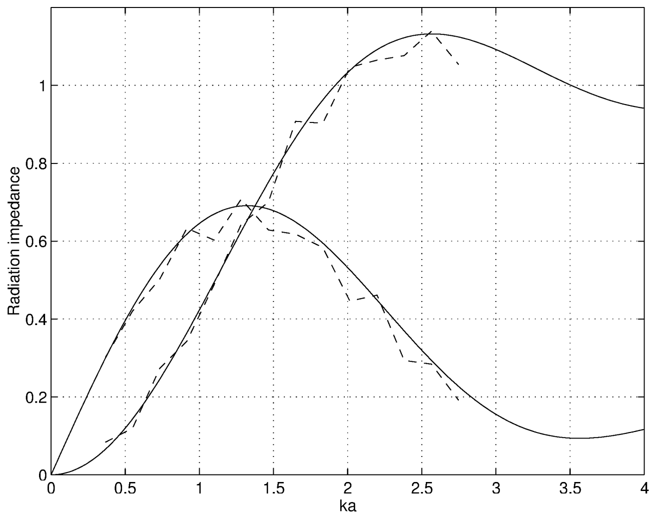
The curve obtained with FEMAK is marked with circles, while the solid curve is found by the formula above.
The two curves can be seen to follow eachother rather closely, although deviations are found at the higher frequencies, as expected.

A.1.3 acouex21 - Radiation impedance

In order to find the radiation impedance of an circular piston in a infinite baffle a semi-infinite domain must be modelled.
The simplest method to model an infinite domain is to define a large domain and terminate it by a suitable impedance. In many cases Z=1 is used (characteristic impedance of the fluid). An example on such use is shown here. It must be remembered that such a calculation domain is not an accurate description of the infinite field. There will be reflections from the outer boundary of the domain. However, by a suitable choice of geometry and terminating impedances a good approximation of the nearfield of the radiating body may be found.
The example is: A circular piston with radius a=1.0 m in an infinite baffle. Frequency range: 10 - 150 Hz.
Here is shown results for a demisphere of radius 10 m for these frequencies using an elementsize of 0.5m. This model has 1757 nodes and 560 elements.
The input file is acouex21.inp .



Geometry of mesh, acouex21.inp. Element number are shown for the elements close to the piston, and those at the outer boundary.



Radiation impedance normalized to area of piston and ρc. Real and imaginary part are shown. The results are shown for normalized wave-number, ka. Solid line - reference, dashed line femak.
The reference impedance is found from [23].

$$Z_r(2ka) = \pi a^2 \rho c(\theta(2ka) + j\chi(2ka)) \quad \text{(A.1-4)}$$

$$\theta(ka) = 1 - \frac{1}{ka} J_1(2ka) \quad \text{(A.1-5)}$$

$$\chi(2ka) = \frac{4}{\pi}\int_0^{\pi/2} \sin(2ka\cos\alpha)\sin^2\alpha d\alpha \quad \text{(A.1-6)}$$

Where Z_r is the radiation impedance, $J_1(x)$, is the Bessel function of 1.order, 1. kind.
As shown in the figure above this simple mesh gives a good approximation to the radiation impedance of the vibrating circular piston in a rigid baffle.

A.1.4 stmpwee1, stmpwee5, stmpwee7 - Radiation impedance using wave envelope element

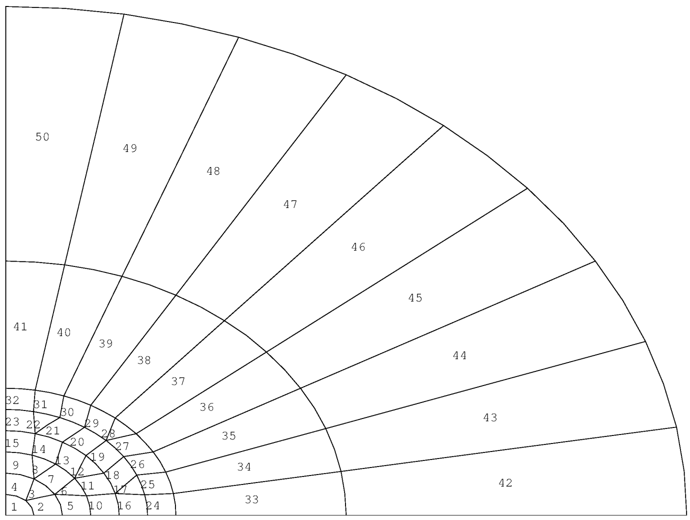
Here, as in the previous example, the radiation impedance of a circular piston in a baffle is computed. In the model used here wave envelope elements, WEE, are use with ordinary finite elements, FE. Using such a model it is possible to reduce the number of nodes. However, it is important to have knowledge on how the WEE work.
The model is shown in the picture below. The piston is 0.4 m in radius, the FE- domain is 0.6 m in radius. Three different "outer domains" modelled using WEE are used.

1. Two layers of WEE's, outer radius 2.4 m 32 FE, 18 WEE, 169 nodes. Geometric center of WEE-domain in the origin (stmpwee1)

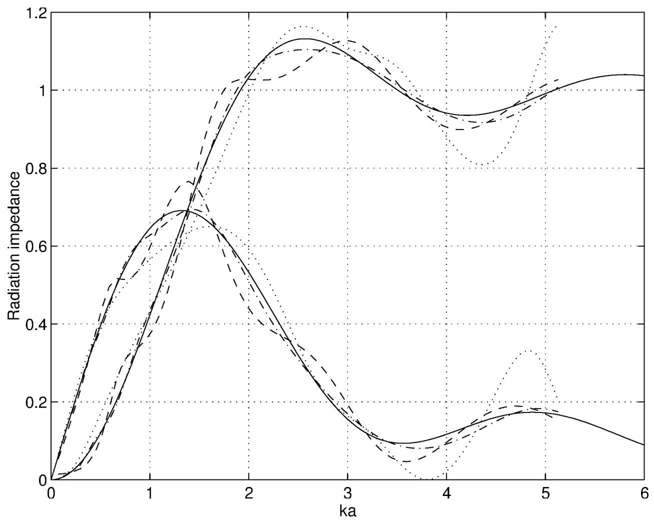
2. Four layers of WEE's, outer radius 9.6 m, 32 FE, 36 WEE, 227 nodes. Geometric center of WEE-domain in the origin (stmpwee7)

3. Two layers of WEE's, outer radius 18.6 m, 32 FE, 36 WEE, 223 nodes. Geometric center of WEE-domain is not in the origin (stmpwee5)

These models results in less DOF's than in the previous example, but we will have unsymmteric matrices because we use WEE.
Here the geometry of the mesh of the smallest modell is shown.



Geometry of the mesh used in case 1 (stmpwee1).



Radiation impedance normalized to area of piston and ρc. Real and imaginary part are shown. The results are shown for normalized wave-number, ka. Solid line - reference, dashed line - case 1, "dash-dotted" line - case 2, dotted line - case 3.
The reference impedance is found from [23].

$$Z_r(2ka) = \pi a^2 \rho c(\theta(2ka) + j\chi(2ka)) \quad (A.1\text{-}7)$$

$$\theta(ka) = 1 - \frac{1}{ka} J_1(2ka) \quad (A.1\text{-}8)$$

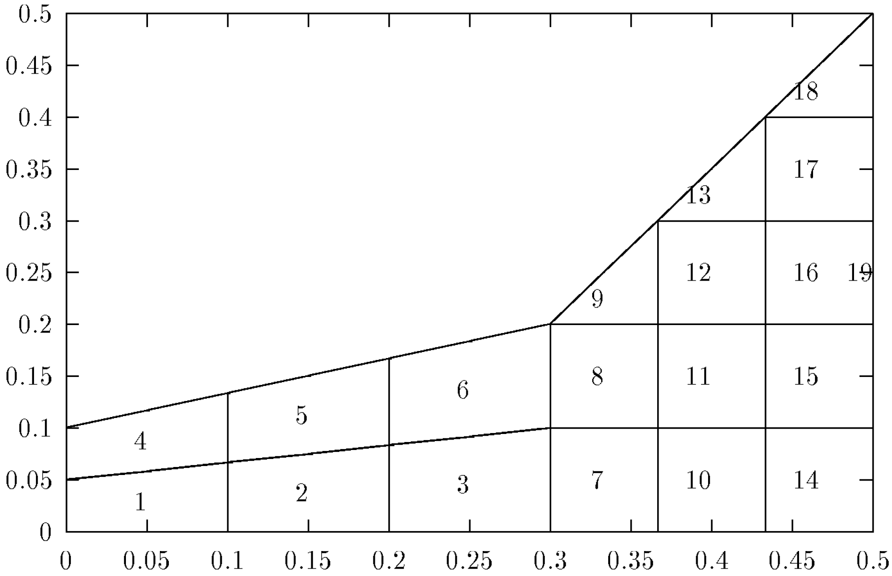
$$\chi(2ka) = \frac{4}{\pi} \int_0^{\pi/2} \sin(2ka\cos\alpha)\sin^2\alpha d\alpha \quad (A.1\text{-}9)$$

Where Z_r is the radiation impedance, $J_1(x)$, is the Bessel function of 1.order, 1. kind.
It can be seen that case 2 has the best approximation to the analytical expression for the radiation impedance. Case 1 has results oscillating about the reference. The results found with case 1 is surpricingly close to the reference. The size of the model is very small compared to the wavelength at the lowest frequencies, and the model is small compared to that in example acouex21.
Case 3 gives less satisfactory results even if this case has the largest outer radius. The background is the geometrical misalignment of the geometrical source and the origin of the wave envelope element.

A.1.5 horn1 - Horn loudspeaker

This example demonstrate the possibility to analyze axisymmetrical horn loudspeakers. The same technique could be used for axisymmetrical musical horn or ducts. Throat impedance, the pressure in the mouth and also farfield directivity are calculated. Impedance type 100 is used to model the horn ending in an infinite baffle.
The horn is a simple one made from two conical sections. The throat of the horn has radius .1 m, the inner section is narrow, 36.9°, and 0.3 m long. The outer section is wider, 90°, and 0.3 m long. The mesh is shown below. The full input file is found in horn1.inp. Element no.19 is of type 100, which is an impedance and not an ordinary element. It has 11 nodes. As mentioned in the section describing IMP100 it uses a modal decomposition of the field in the horn. The approximation of the acoustic field will be better with many modes. However, maximum number of modes is less than number of modes in the hole.
Here maximum number of modes that can be used in the impedance model is 5. Results are presented for 3, 4 and 5 modes. This impedance model also offer the possibility to

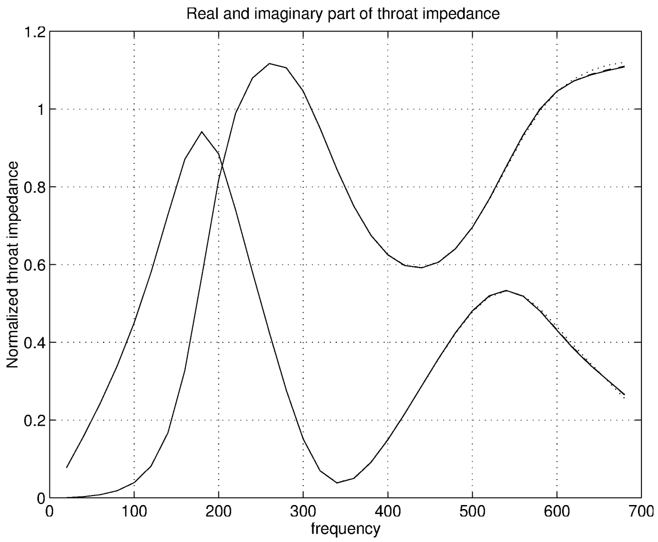
compute the pressure outside the hole. This is done by defining exterior computation points in the DYNM-block.



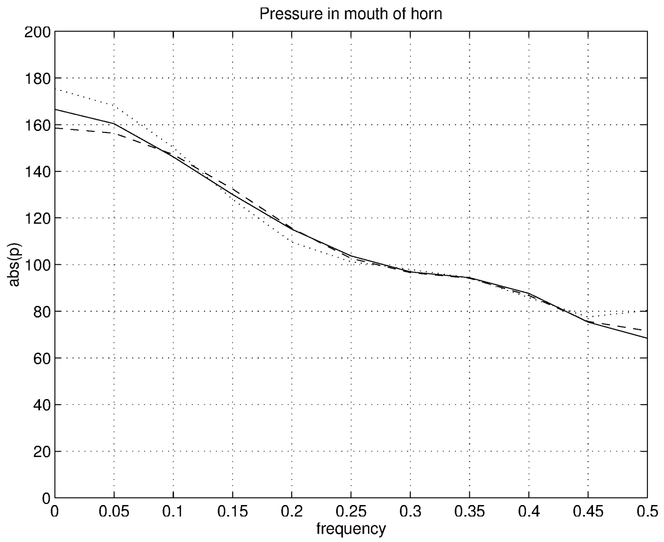
Throat impedance

The velocity in the throat is 1.0 m/s. The throat impedance can therefore be found directly by the pressure in the throat. The throat impedance is found by:

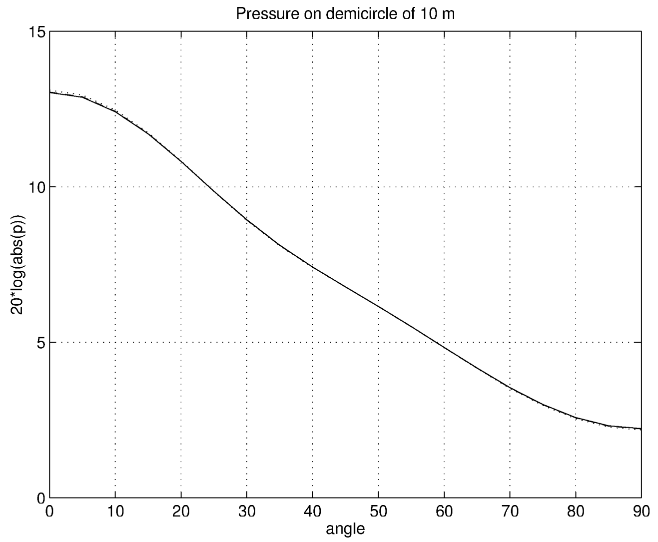
$$z_t = \frac{2\pi \int p(r,0) r dr}{\pi r^2} \tag{A.1-10}$$



Computations for throat impedance. The results are obtained using three different cases: 5 modes are marked with solid line, 4 modes dashed line, 3 modes dotted line.



Computations of pressure in the horn's mouth. Curves are marked as previous curves.



Computations pressure as a function of angle 10 m from the horn. Curves are marked as previous curves.
Note that the directivity plots and the throat impedances shows nearly the same results for the three different calculations. There are clear differences for the pressure in the mouth of the horn (in the opening of the baffle). This is expected because this is the position where the influence of the impedance model is strongest.

A.2 Pure elasticity examples

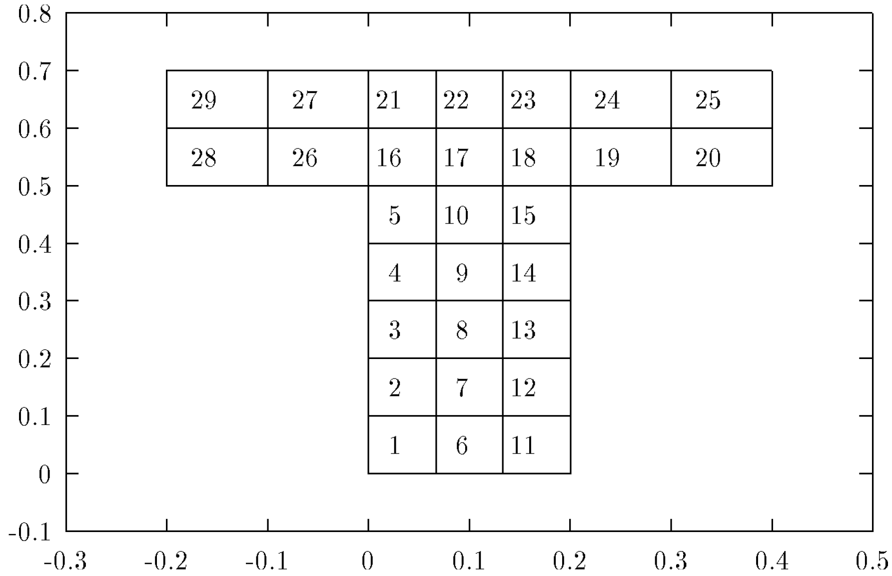
Although FEMAK is primarely made for acoustic waves, it is possible to do simple analysis on elastic materials. There are two models for elastic solids (ELEM02 and ELEM20) and one model for plates/beams (ELEM08). Here some examples are presented with these elements without coupling to acoustic waves.

A.2.1 elaex11 - Eigenvalue elastic material

A simple T-shaped geometry is investigated. The material is steel. The lower part of the object is clamped ($u_x = u_y = 0$) for node 1-7 (nodes with y=0.0).
Material parameters:

E-modul $[Pa]$	Poisson's ratio	Density $[kg/m^3]$
$1.95 \cdot 10^{11}$	0.28	7700

The geometry is shown in the figure below.

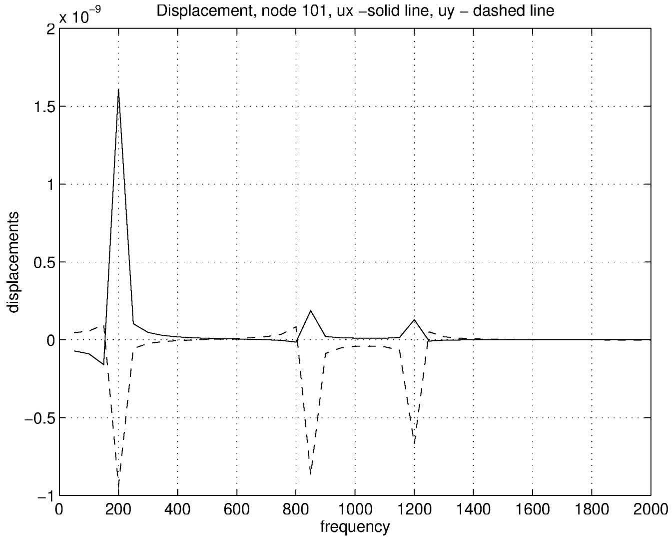


FEMAK gives the results as shown in the table below:

ω_0^2 [(rad/s) ^{2}]	$1.518 \cdot 10^6$	$2.820 \cdot 10^7$	$5.726 \cdot 10^7$	$2.428 \cdot 10^8$	$2.629 \cdot 10^8$
f_0 [Hz]	196.1	845.2	1204.3	2480.0	2580.6

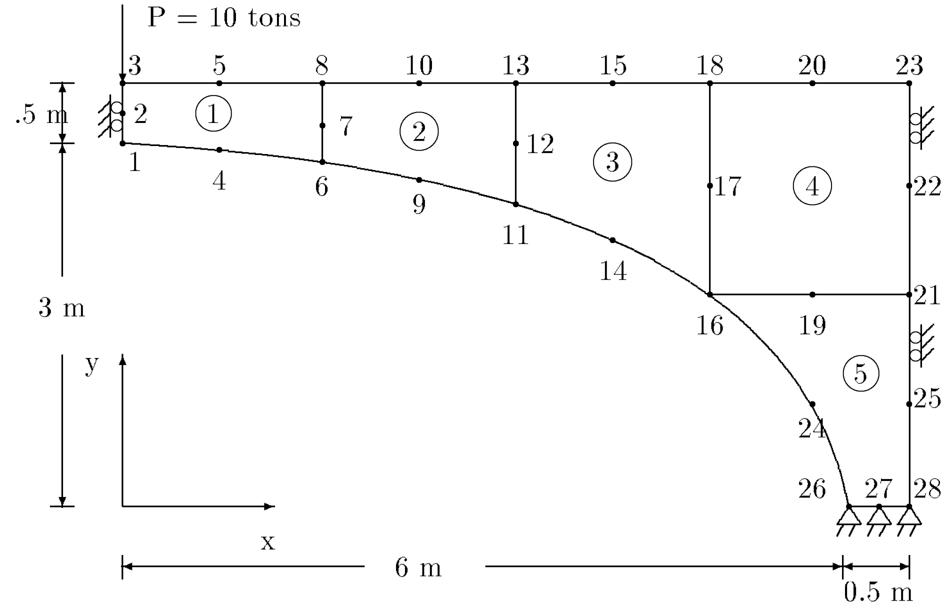
A.2.2 elaex12 - Forced vibrations, elastic material

This example use the same geometry and materials as elaex11. ELEM20 must be used for this analysis. The forces attack at the top of the left part of the T. (y=0.7, $-0.2 \leq x \leq 0.0$). This means that the forces attack at side 5 of elements 27 and 29. See input file for details. Calculations are done for the frequencies 100 - 2000 Hz. Displacement of the right tip of the T (node 101) is shown as a function of frequency. The displacements is largest at the resonance frequencies.



A.2.3 bigex2 - Elliptical concrete arch

See also [1],p.481.



Material constants:

E - $0.2 \cdot 10^7$ tons/m^2

ν - 0.3

ρ - 2.3 tons/m^3

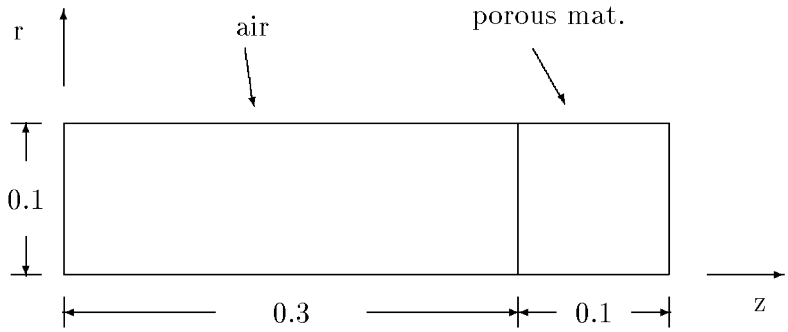
thickness - 1 m

The model use two type of loads, a distributed dead weight (using the SOLR-block), and a consentrated force in node (using the SOLC-block). The input file is shown in section describing ELEM02.

A.3 Coupling fluid and porous material

A.3.1 porex1 - Kundt's tube, equivalent fluid model

Here computations on the acoustic field in the Kundt's tube is used as an example on use on femak with fluid and porous material. The results found with femak are compared with those found with analytical models (plane wave assumptions). The porous material is modelled with ELEM04 , equivalent fluid.
The geometry is shown below. The tube is axisymmetrical, radius is 0.1 m, the air- column is 0.30 m, while the thickness of the porous material is 0.1 m. The source is a vibrating surface at x=0.



Different models have been used for the porous materials.
The fluid, and the fluid in the pores is air which has data:

ρ kg/m^3	c_a (adiabatic) m/s	B^2	η poiseuille
1.205	343.4	0.71	$1.84 \cdot 10^{-5}$

All models use $\sigma = 10000 Ns/m^4$, porosity: $\phi = 0.9$, structure factor: $k_s = 1.5$. These are used for Craggs' model (both revised and old implementation): The Johnson-Allard model take the frequency-dependent viscous and thermal losses into account. This model will deviate from Craggs' model when the pore-size is very small compared to the wavelength. Therefore two sets of characteristic lengths are used:

1. $\Lambda = \Lambda' = 10 \cdot 10^{-2} m$ which indicates large pores, and the models should give similar results.

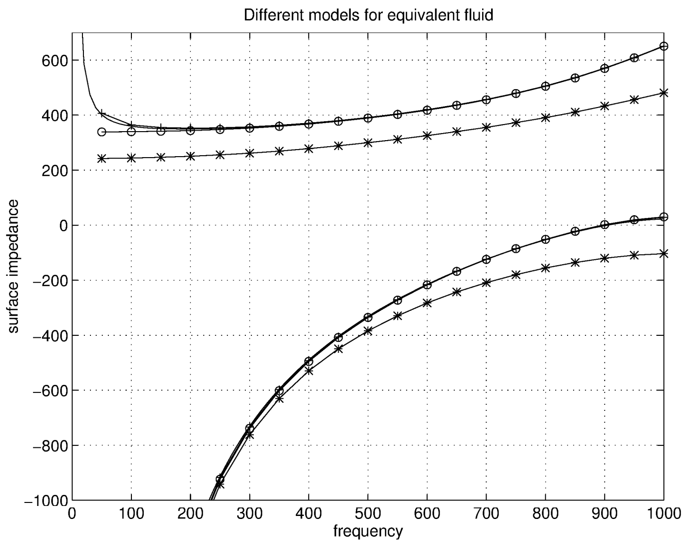
2. $\Lambda = \Lambda' = 10 \cdot 10^{-4} m$ which indicates smaller pores, and the models should give different results.

```
.
.
PREL                                    ; Craggs model, revised
2,6
1,1.205,343.4                           ; material 1 - air
2,1.0,1.205,343.4,10000.0,0.90,1.5      ; material 2 - porous material
0
.
.
.
.
PREL                                    ; Craggs model, old implementation
2,6
1,1.205,343.4                           ; material 1 - air
2,10.0,1.205,343.4,10000.0,0.90,1.5     ; material 2 - porous material
0
.
.
PREL
```

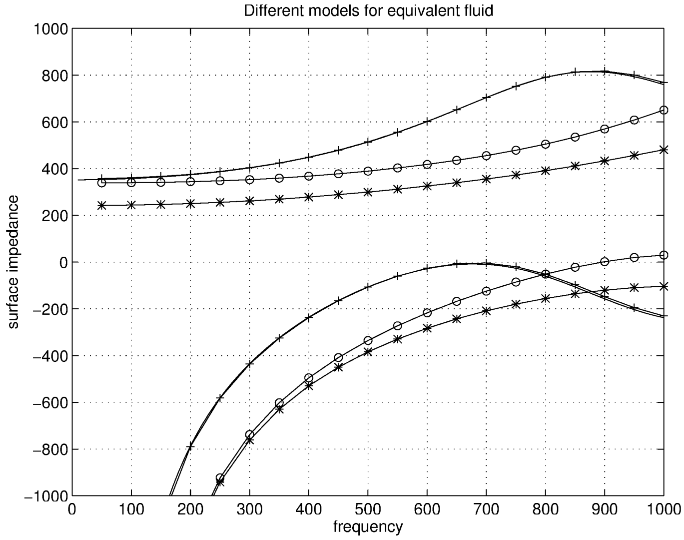
```
2,11                                        ; Johnson-Allard model, large pores
1,1.205,343.4,0.0,0.0,0.0,0.0,0.0           ; material 1 - air
0.0
2,2.0,1.205,343.4,0.71,1.84e-5,1.4,0.90     ; material 2 - porous material
1.5,10000.0,100.0e-4,100.0e-4
0
.
.
PREL
2,11                                        ; Johnson-Allard model, small pores
1,1.205,343.4,0.0,0.0,0.0,0.0,0.0           ; material 1 - air
0.0
2,2.0,1.205,343.4,0.71,1.84e-5,1.4,0.90     ; material 2 - porous material
1.5,10000.0,100.0e-2,100.0e-2
0
.
.
```

To find the surface impedance of the porous material the impedance of the vibrating surface at x=0 (piston) is determined first, z_2, and then the impedance is transformed. The impedance on the piston is found by giving it a velocity 1.0 m/s. Then the pressure on the piston will be equal to the impedance. The surface impedance is:

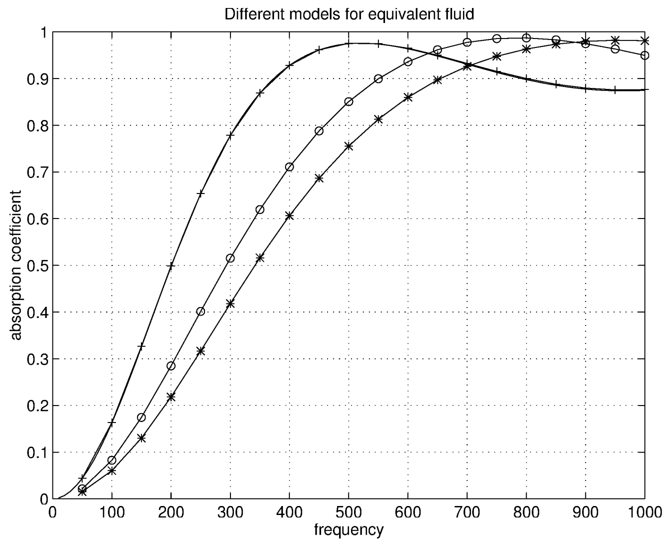
$$z_i = z_c \frac{j z_2 \cot(kd) + z_c}{z_2 + j z_c \cot(kd)} \tag{A.3-1}$$



Surface impedance (real- and imaginary-part). The three models are: circle - Craggs' model revised, star - Craggs' model old implementation, and cross - Johnson-Allard model, large pores. Solid line, analytical model, [24], using using Johnson-Allard model.



Surface impedance (real- and imaginary-part). The three models are: circle - Craggs' model revised, star - Craggs' model old implementation, and cross - Johnson-Allard model, small pores. Solid line, analytical model, [24], using using Johnson-Allard model.



Absorption coefficient. The three models are: circle - Craggs' model revised, star - Craggs' model old implementation, and cross - Johnson-Allard model, small pores. Solid line, analytical model, [24], using using Johnson-Allard model.

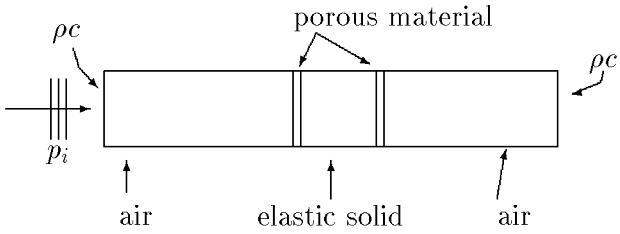
The above figures shows surface impedance (real- and imaginary-part) for the two materials presented above, and also absorption coefficient for the case with smallest characteristic lengths. The three models are circle - Craggs' model revised, star - Craggs' model old implementation, and cross - Johnson-Allard model. Solid line, analytical model, using using Johnson-Allard model.
The curves found with the analytical model, computed with the programme maine, [24], are identical to those found with femak (Johnson-Allard model).
It can be seen that the Craggs' model and the Johnson-Allard model gives similar results for the largest characteristic lengths. At the lowest frequencies differences are found. The Johnson-Allard model gives different bulk modulus and density of the equivalent from those found with Craggs' model in the lower frequency-domain. However, the size of the pores must be accounted for. The characteristic lengths for the material is dependent on

the pore-size.
When using smaller characteristic lengths clear differents between the models are found. The curves found with the old implementation of Craggs' model shows clear differences from the two other models.

A.4 Coupling fluid, porous and elastic material

A.4.1 layer1 - Simple layered material in infinite tube

This is a simple example of a layered material. The layered material is two aluminum plates of thickness 1 cm, with a layer of porous material of thickness 20 cm between the two plates. The material is put in a duct with air on both sides. The model has only two elements in the transversal direction. This is because it is used for comparison with incoming plane waves of normal incidence.
The geometry of the model is shown below.



There is an incoming plane wave from left, p_i. It is modelled using source type 50. The modell is ended in characteristic impedances, ρc. This ensures no reflections from the end of the "tube". In other words, semi-infinite extent of the air-domain to the left and right of the model. At the boundary of the model, all displacements in the transversal direction are set to zero for the porous and the elastic materials.
The results from this model can be compared with those of a method assuming plane waves and infinite extent of the model. See Allard, [8]. The programme maine, [24], which is based on these assumptions is used.
This model use three groups of materials, one for the porous material, one for the air, and one for the elastic solid.
The porous material, material no.1, has the following material parameters:

- density of the porous material, $\rho_1 = 30.0 kg/m^3$, VPREE(6)=30.0,
- shear modulus $G = 1.8 \cdot 10^5 + j1.8 \cdot 10^4$ Pa (VPREE(7)=1.80e5, VPREE(8)=1.80e4),
- Poissons rations 0.4 (VPREE(9)=0.4, VPREE(10)=0.0),
- porosity: $\phi = 0.93$ (VPREE(11)=0.93),
- structure factor: $k_s = 3.2$, (VPREE(12)=3.2),
- flow resisitivy: $\Phi = 10000 Ns/m^4$ (VPREE(13)=10000),
- characteristic length vicous loss $\Lambda = 1.0 \cdot 10^{-4}$ m,VPREE(14)=1.0e-4,
- characteristic length thermal loss $\Lambda' = 1.0 \cdot 10^{-4}$ m VPREE(15)=1.0e-4 .
- Density of pore fluid (air), 1.205 kg/m^3,(VPREE(1)=1.205)
- Speed of sound pore fluid (air), 340.9 m/s,(VPREE(2)=340.9)
- Prandtl's number pore fluid (air), 0.71 ,(VPREE(3)=0.71)
- Viscosity of pore fluid (air), $\eta = 1.84 \cdot 10^{-5}$ poiseuille,(VPREE(4)=1.84e-5)
- Fraction of heat capacities of pore fluid (air), $\gamma = c_p/c_v = 1.4$,(VPREE(5)=1.205)

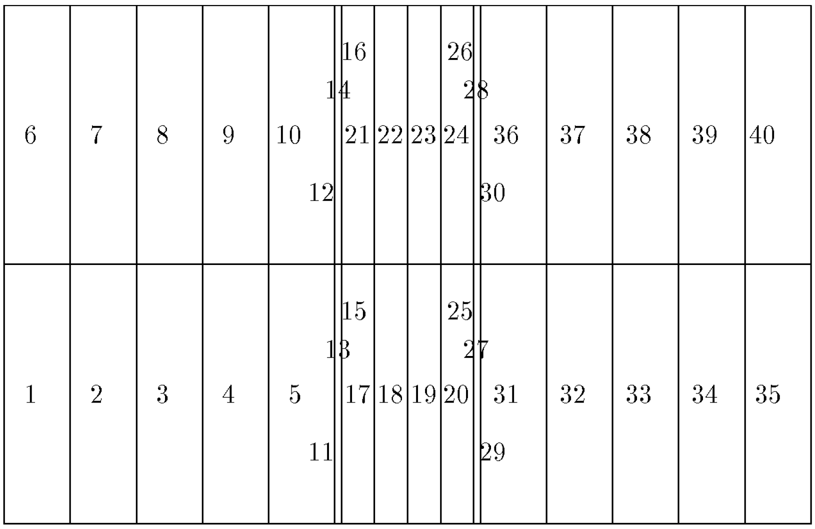
The air, material no.2, has the following material parameters:

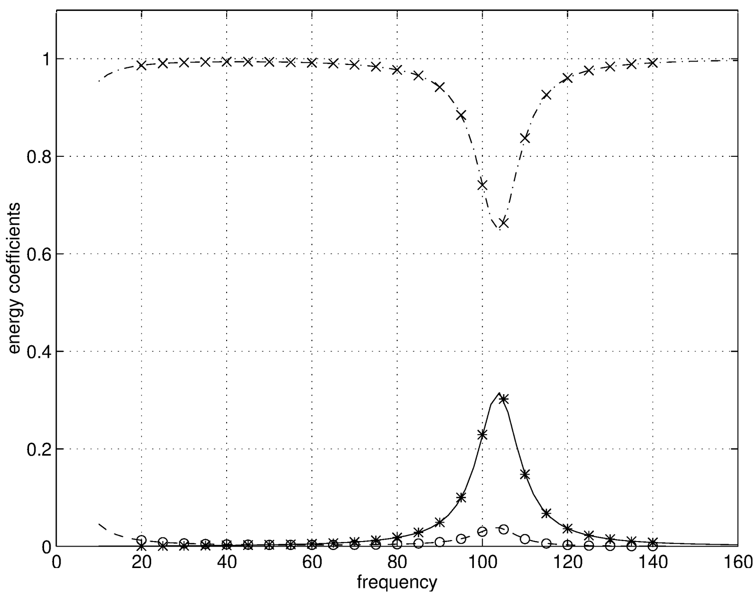
- Density air, 1.205 kg/m^3,(VPREE(1)=1.205)
- Speed of sound of air, 340.9 m/s,(VPREE(2)=340.9)
- 13 zeros to have 15 material parameters

The elastic solid, material no.3, has the following material parameters:

- Young modulus, $7.1 \cdot 10^{10}$ Pa (VPREE(1)=7.1e10)
- Poissons ratio 0.3 (VPREE(2)=0.3)
- Density of elastic material, 2700 kg/m^3,(VPREE(3)=2700)
- Loss parameter $\alpha = 0.0$,(VPREE(4)=0.0)
- Loss parameter $\beta = 0.0$,(VPREE(5)=0.0)
- 10 zeros to have 15 material parameters

The input file for this model can be found in genex/layer1.inp Below, the mesh and some results from calculations are shown. The agreement between femak and transfer-matrix calculations is excellent.





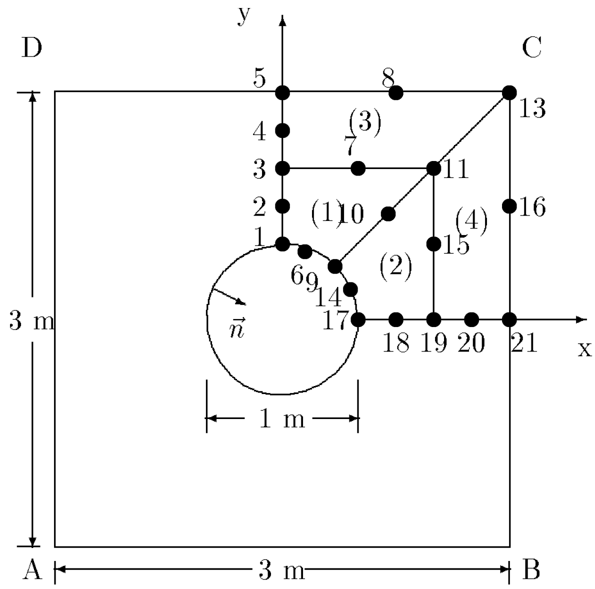
Results from computations using Femak and Maine. Maine: solid line - absorption coefficient, dashed line - transmission coefficient, dash-dot line - reflections coefficient. Femak: star - absorption coefficient, circle - transmission coefficient, x - reflections coefficient.

A.5 Other examples

A.5.1 genex3, genex4 - Heat conduction in perforated plate

genex3 - Steady state case

These examples on heat conduction in a perforated plate can be found in [1], p.447-480. There are minor differences both because FEMAK is sligthly modified from MEF in [1], and because I think there are some errors in that text considering the loads (sources).
The perforated plate is shown below.



Geometry and mesh of heat transfer problem

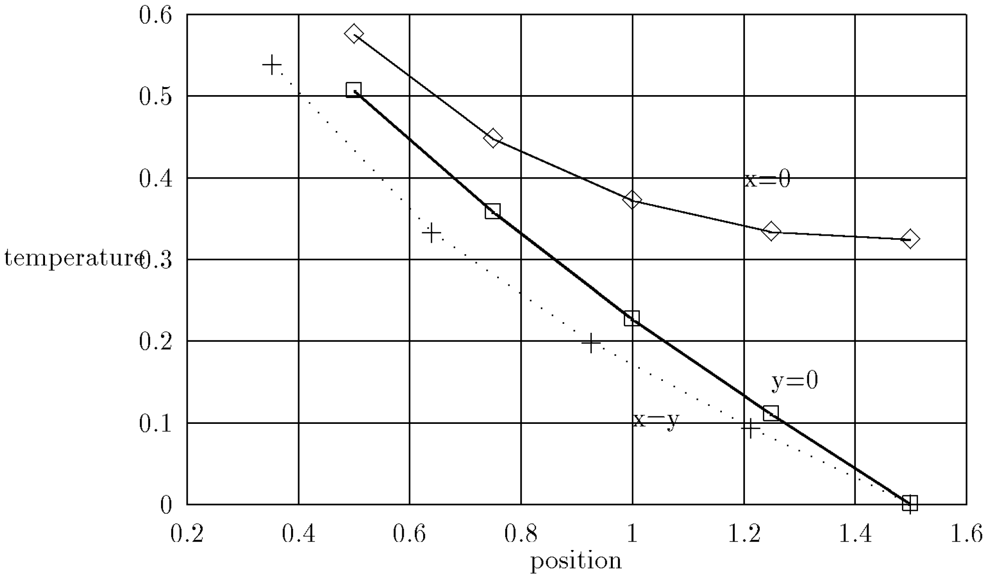
This example is 2-dimensional. The material is concrete, it is isotropic with the thermal conductivity $d = d_x = d_y = 1.4$ w/(m°C) .
The boundary conditions are:

- u=0, on AD and BC (specified temperature). (COND - block)
- $\partial u/\partial n = \partial u/\partial y = 0$ on AB and CD (insulated edges) (Natural boundary condition).
- d $\partial u/\partial n = 1$, Specified heat flux on the inside circle (SOLC block).

The model use a quarter of the plate because of the symmetry in the problem. The heat flux is accounted by concentrated nodal load of values:

- $\pi/48$ at nodes 1 and 17
- $\pi/24$ at nodes 9
- $\pi/12$ at nodes 6 and 14

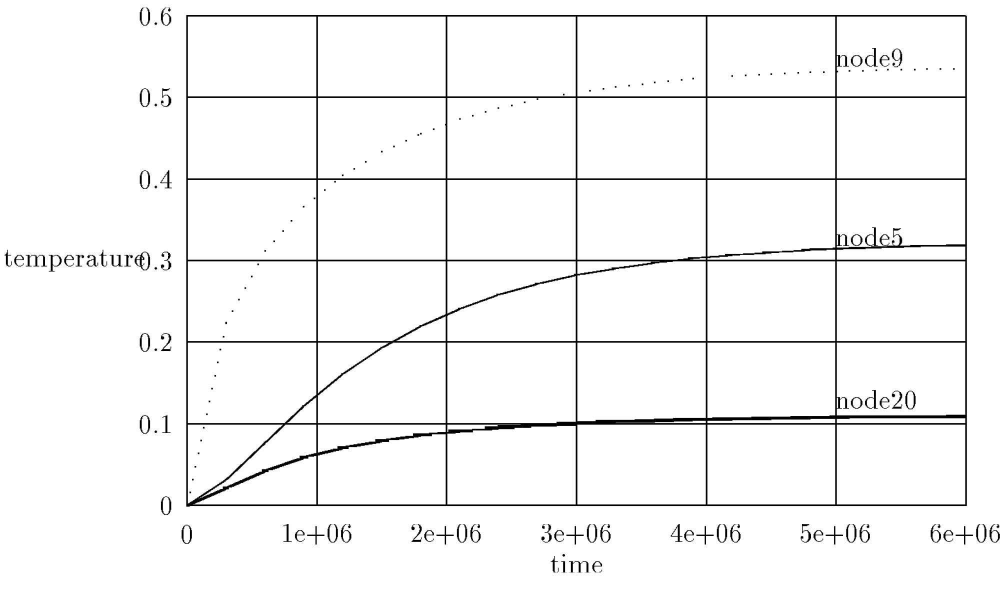
The results are presented below.



Temperature in along along x=0 (nodes no.1, 2, 3, 4, 5) x=y (nodes no.9, 10, 11, 12, 13), and y=0 (nodes no 17, 18, 19, 20, 21).

genex4 - Transient case

The transient case is similar the steady state case.The initial condition is u=0.0 The thermal capacity ρc is $2.03 \cdot 10^6 J/m^3\ °C$. For this case the TEMP block is used(implicit Euler's method).
Some results are presented in the figure below



Temperature in node 9, 5 and 20

Bibliography

[1] G.Dhatt G.Touzot. *The Finite Element Method Displayed.* John Wiley & Sons, Chichester, 1984.

[2] PDA Engineering, PATRAN Division. *Patran Plus, User Manual.*

[3] A.Craggs. A finite element method for damped acoustic systems: An application to evaluate the performance of reactive mufflers. *Jornal of sound and vibration*, 48:377–392, 1976.

[4] J.P.Coyette. Sysnoise seminar, march 1989. Numerical Integration Technologies, Belgium.

[5] Y.Kagawa T.Yamabuchi T.Yoshikawa S.Ooie N.Kyouno T.Shindou. Finite element approach to acoustic transmission-radiation systems and application to horn and silencer design. *Journal of sound and vibration*, 69:207–228, 1980.

[6] O.C.Zienkiewicz R.L.Taylor. *The Finite Element Method*, volume 1. McGraw-Hill, 4.ed., London, 1991.

[7] Petyt. The finite element method in acoustics. In P.Filippi, editor, *Theoretical acoustics and numerical techniques*, pages 51–104. Springer, 1982.

[8] Jean Francois Allard. *Propagation of sound in porous media, Modelling sound absorbing materials.* Elsevier, London, 1993.

[9] A.Craggs. A finite element model for rigid porous absorbing materials. *Journal of Sound and Vibration*, 61:101–111, 1978.

[10] A.Craggs. A finite element model for acoustically linedsmall rooms. *Journal of Sound and Vibration*, 108:327–337, 1986.

[11] Biot M.A. Theory of propagation of elastic waves in a fluid-saturated porous solid. i. low-frequency range. *Journal of acoustical society of America*, 28:168–178, 1956.

[12] Biot M.A. Theory of propagation of elastic waves in a fluid-saturated porous solid. ii. higher frequency range. *Journal of acoustical society of America*, 28:179–191, 1956.

[13] Biot M.A. Generalized theory of acoustic propagation in porous dissipative media. *Journal of acoustical society of America*, 34:1254–1264, 1962.

[14] Biot M.A. Mechanics of deformation and acoustic propagation in porous media. *Journal of applied physics*, 33:1482–1498, 1962.

[15] Willis D.G. Biot M.A. The elastic coefficients of the theory of consolidation. *Journal of Applied mechanics*, 24:594–601, 1957.

[16] Simon B.R. Wu J.S.-S. Zienkiewicz O.C. Paul D.K. Evaluation of u-w and u-π finite element methods for the dynamic response of saturated porous media usine one-dimensional models. *International journal for numerical and analytical methods in geomechanics*, 10:461–482, 1986.

[17] G. Degrande. *A spectral and finite element method for wave propagation in dry and saturated poroelastic media.* PhD thesis, Katholieke Universiteit te Leuven, 1992.

[18] Leissa. *Vibration of plates.* NASA SP-160, 1969.

[19] O.C.Zienkiewicz R.L.Taylor. *The Finite Element Method*, volume 2. McGraw-Hill, 4.ed., London, 1991.

[20] W.Eversman R.J.Astley. Finite element formulations for acoustical radiation. *Journal of sound and vibration*, 88:47–64, 1983.

[21] I.M.Smith. *The Finite Element Method - with applications to geomechanics.* John Wiley & Sons, Chichester, 1982.

[22] T.F.W.Embleton. *Noise and Vibration Control*, chapter Mufflers.

[23] P.M.Morse K.U.Ingard. *Theoretical Acoustics.* Princeton University Press, Princeton, New Jersey, 1986.

[24] B.Brouard. *Maine - User's manual.* Laboratoire d'acoustique de l'Univerist e du Maine, URA CNRS 1101 Av. O Messian, B.P.535, 7217 Le Mans CEDEX, France, 1969.