Chapter: 3

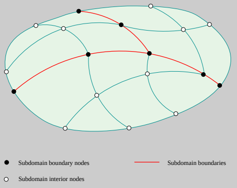
MEK4560
The Finite Element Method in Solid Mechanics II

(January 31, 2008)

TORGEIR RUSTEN

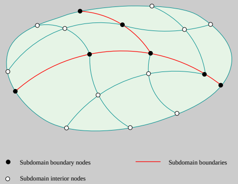
(E-post:torgeiru@math.uio.no)

Department of Mathematics
University of Oslo



Contents

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

3. *Variational crimes* [Strang and Fix, 1973]

In this chapter we introduce some "tricks of the trade" used to improve some finite elements. First *selective reduced integration* is motivated by an example. Then an application of *uniform reduced integration* is mentioned, see [Cook et al., 2002][1], Chapter 6.8.

Another method to improve elements for bending dominated analysis is incompatible modes, see [Cook et al., 2002][1], Chapter 6.6.

We also mention some "improved" element formulations, *elements with rotational degrees of freedom* and *hybrid elements*, [Cook et al., 2002][1] 3.10, 4.10.

All the above mentioned methods are called *Variational crimes* in Strang and Fix [Strang and Fix, 1973][3].

Since the basic requirements for convergence are violated an alternative criteria is introduce, named *the patch test*, [Cook et al., 2002] 6.13.

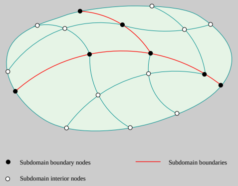
Element evaluation are mentioned briefly, see [Cook et al., 2002][1] 6.11.

---

[1] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt. *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition, October 2002.

[3] G. Strang and G. J. Fix. *An Analysis of the Finite Element Method*. Prentice-Hall, Englewood Cliffs, New Jersey, 1973.

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

Contents
◁◁ ▷▷
◁ ▷
Page 3 of 35
Go Back
Close
Quit

3.1. Full integration

If an integration rule are sufficiently accurate to integrate the stiffness coefficients exactly the rule is called a *full integration* rule, see [Cook et al., 2002][1], Chapter 6.8. Quadrilateral and hexahedral elements are undistorted if they are rectangular and mid side nodes, if present, are uniformly spaced along straight edges.

3.2. Selective reduced integration

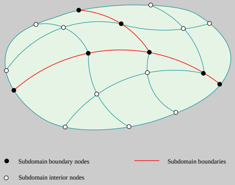
Selective integration of the shear term was introduce in the begin of the seventies in order to improve the accuracy of lower order square elements applied to bending dominated problems. To motivate it we consider a plane stress model and a bending dominated problem.

Pure bending is defined by $\sigma_{xx} \propto y$. The displacement field is given by

$$u = x\,y$$
$$v = -\frac{1}{2}\left(x^2 + \nu y^2\right)$$

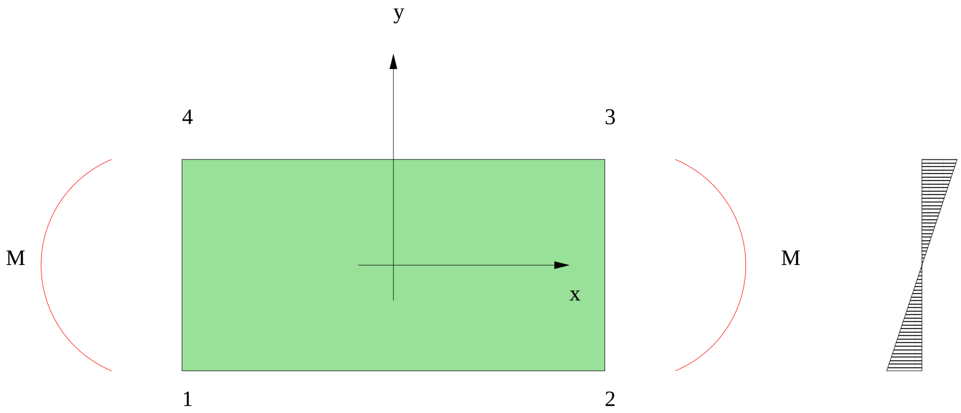
[1] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt. *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition, October 2002.

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

This is illustrated in the figure:



In this case the strains are:

$$\boldsymbol{\varepsilon} = \begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{pmatrix} = \begin{pmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{pmatrix} \begin{pmatrix} u \\ v \end{pmatrix} = \begin{pmatrix} y \\ -\nu y \\ 0 \end{pmatrix}$$

Inserting this in the stress strain relation result in

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{pmatrix} = \frac{E}{1-\nu^2} \begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1}{2}(1-\nu) \end{pmatrix} \begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{pmatrix} = E \begin{pmatrix} y \\ 0 \\ 0 \end{pmatrix} \tag{3.1}$$

and the strain energy becomes

$$\frac{E}{2}\int_{-1}^{1}\int_{-1}^{1} y^2\,dx\,dy = \frac{2}{3}E \tag{3.2}$$

Then, consider a bilinear element with nodes in $(-1,-1)$, $(1,-1)$, $(1,1)$ and $(-1,1)$. The bilinear basis functions are

$$N_1(x,y) = \frac{1}{4}(1-x)(1-y) \tag{3.3}$$

$$N_2(x,y) = \frac{1}{4}(1+x)(1-y) \tag{3.4}$$

$$N_3(x,y) = \frac{1}{4}(1+x)(1+y) \tag{3.5}$$

$$N_4(x,y) = \frac{1}{4}(1-x)(1+y) \tag{3.6}$$

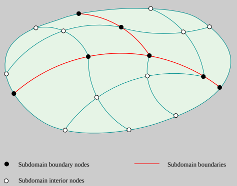
The finite element interpolation of u is denoted u_h and $u_h = xy$, i.e. is equal to u. The value of v at all the nodes are $-\frac{1}{2}(1+\nu)$, hence $v_h = -\frac{1}{2}(1+\nu)$. Thus, the strains becomes

$$\varepsilon = \begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \tau_{xy} \end{pmatrix} = \begin{pmatrix} y \\ 0 \\ x \end{pmatrix} \tag{3.7}$$

Inserting into the stress strain relations result in

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ xy \end{pmatrix} = \frac{E}{1-\nu^2} \begin{pmatrix} y \\ \nu y \\ \frac{1}{2}(1-\nu)x \end{pmatrix} \tag{3.8}$$

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

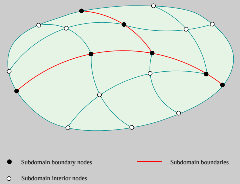
and the strain energy becomes

$$\frac{E}{2(1-\nu^2)}\int_{-1}^{1}\int_{-1}^{1} y^2 + \frac{1}{2}(1-\nu)x^2\,dx\,dy = \frac{2}{3}\frac{E}{1-\nu^2} + \frac{2}{3}G \tag{3.9}$$

Comparing this to (3.2) we note the appearance of the shear term, in addition to the membrane term. The unphysical shear therm absorb part of the energy, and the finite element solution appears to stiff. Note that if the shear term is integrated numerically by a one point integration rule, where the evaluation point is at the origin the shear term vanish.

This motivates the use of reduced shear integration. Note that this does not prove that reduced integration improve the computed results! It is an indication that it is worth trying. The experience in computation mechanics, however, is that reduced integration of the shear term improve the numerical results.

Department of Mathematics
University of Oslo



Contents
Page 7 of 35
Go Back
Close
Quit

3.3. Uniform reduced integration

In *Uniform reduced integration* the integration rule for all the terms are of lower order than *full integration*.

This has two effects:

1. The element becomes softer.
2. The computational time is reduced.

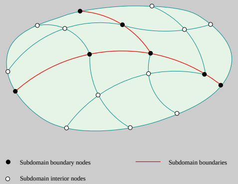
Sometimes this has some advantages, however it is important that the rank of the stiffness matrix is correct. This may not be the case with reduced integration.

3.4. Mindlin-Reissner beams

In Chapter 2 methods for avoiding shear locking was discussed. One efficient method to achieve this is *uniform reduced integration*.

Basis functions	Linear	Quadratic	Cubic
Integration rule (Gauss)	One Point	Two point	Three point

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

In all cases the stiffness matrix has full rank. Note that all therms in the stiffness matrix are integrated exactly except the transverse shear terms.

Residual Bending Flexibility: In [H.MacNeal, 1978][2] *Residual Bending Flexibility* is introduced as a method to improve two node beam elements combined with one point integration. He set

$$\gamma A^s \quad \xrightarrow{\text{med}} \quad \left(\frac{1}{\gamma A^s} + \frac{h^2}{12EI} \right)^{-1}$$

and obtain exactly the same expressions as in beam theory. The last term above is called *Residual Bending Flexibility*.

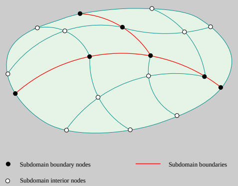
3.5. Incompatible modes

Incompatible modes, or *incompatible displacemets*, is a method used to improve an element's properties with respect to bending dominated analysis. The name comes from the fact that the displacements are not continuous across element boundaries.

The goal is to be able to deform an element into an element with curved sides. A four node

[2] R. H.MacNeal. A simple quadrilateral shell element. *Computer & Structurer*, 8:175—183, 1978.

Department of
Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

isoparametric membrane element has a displacement field given by:

$$\begin{pmatrix} u \\ v \end{pmatrix} = \begin{pmatrix} \boldsymbol{N}_0 & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{N}_0 \end{pmatrix} \begin{pmatrix} \boldsymbol{d}_u \\ \boldsymbol{d}_v \end{pmatrix} + \begin{pmatrix} (1-\xi^2) & (1-\eta^2) & 0 & 0 \\ 0 & 0 & (1-\xi^2) & (1-\eta^2) \end{pmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \end{pmatrix}$$

where a_i are "nodeless", or generalized, degrees of freedom. Note that the parameters $a_1, \cdots, a_4$ are associated to the given element and that the terms $1-\xi^2$ and $1-\eta^2$ result in discontinuous displacement across inter element boundaries. The element is frequently called Q6 in the literature. We will comment on convergence later.

The advantage is that provided the element is rectangular it can reproduce bending around the element axes exactly.

The derivation of the element matrices are as usual.

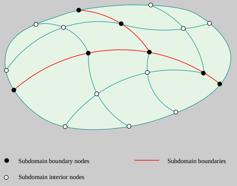
Displacements: ...is given by:

$$\boldsymbol{u} = \boldsymbol{N}\boldsymbol{d} + \boldsymbol{N}_a\boldsymbol{a}$$

Strains: ...is found from the displacement fields:

$$\boldsymbol{\varepsilon} = \boldsymbol{\partial}\boldsymbol{u} = \boldsymbol{\partial}\boldsymbol{N}\boldsymbol{d} + \boldsymbol{\partial}\boldsymbol{N}_a\boldsymbol{a} = \boldsymbol{B}\boldsymbol{d} + \boldsymbol{B}_a\boldsymbol{a}$$

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

Stiffness matrix: ...as usual. Start by introducing the extended strain matrix:

$$\bar{\boldsymbol{B}} = \begin{pmatrix} \boldsymbol{B} & \boldsymbol{B}_a \end{pmatrix} \qquad \text{and displacements} \qquad \bar{\boldsymbol{d}} = \begin{pmatrix} \boldsymbol{d} \\ \boldsymbol{a} \end{pmatrix}$$

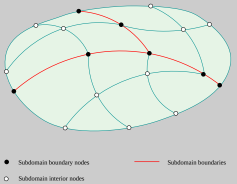
The stiffness matrix from the extended system is found directly from:

$$\bar{\boldsymbol{k}} = \int_{A^e} h \bar{\boldsymbol{B}}^T \boldsymbol{E} \bar{\boldsymbol{B}} \, dA = \begin{pmatrix} \boldsymbol{k} & \boldsymbol{G}^T \\ \boldsymbol{G} & \boldsymbol{H} \end{pmatrix}$$

where

$$\boldsymbol{k} = \int_{A^e} h \boldsymbol{B}^T \boldsymbol{E} \boldsymbol{B} \, dA$$
$$\boldsymbol{G} = \int_{A^e} h \boldsymbol{B}_a^T \boldsymbol{E} \boldsymbol{B} \, dA$$
$$\boldsymbol{H} = \int_{A^e} h \boldsymbol{B}_a^T \boldsymbol{E} \boldsymbol{B}_a \, dA$$

Note that the added degrees of freedom are local to the matrix, thus can be eliminated locally using *static condensation*.

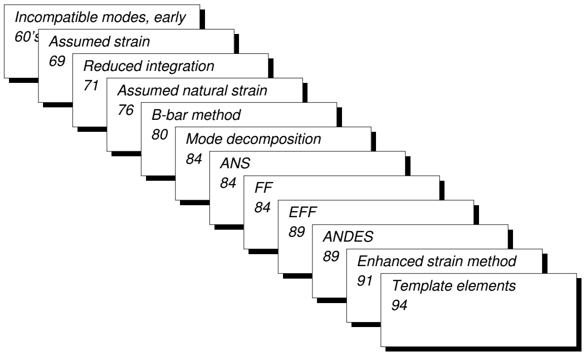


3.6. Membrane elements

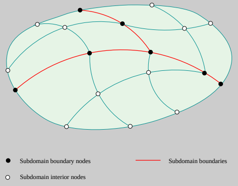
A large number of membrane elements are found in the literature, mostly to improve the accuracy of the low order elements for coarse meshes. We might call them

> *. . . high performance elements.*

Some elements are listed in the figure below:



Note that with the amount of memory available on todays computers combined with modern algorithms from numerical linear algebra, use of higher order elements together with a modest mesh refinement is a viable alternative to low order high performance elements.



3.6.1. Rotational degrees of freedom

In addition to the usual translation degrees of freedom one use rotations in the corner nodes as nodal degrees of freedom. We mention two interesting aspects of this:

1. Improved accuracy of three and four node elements
2. Useful in shell formulations

In[Cook et al., 2002][1] a modification of the quadratic elements for plane elasticity is considered. Here the six displacements degrees of freedom at the mid side nodes are replaced with three rotational degrees of freedom at the corner nodes.

3.6.2. Hybrid formulations

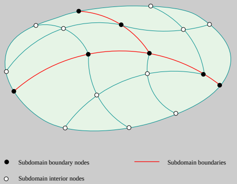
Hybrid formulations are not based on the principle of *complementary strain energy*.

In some cases the hybrid formulation is build on the assumptions:

1. the stress field $\boldsymbol{\sigma}$ is in equilibrium in the element
2. the displacements $\boldsymbol{u}_d$ are defined on inter element boundaries

---

[1] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt. *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition, October 2002.



3.7. The Patch test and convergence

3.7.1. Background

In order for the finite element method to be useful the linear systems must be nonsingular, i.e. solvable, and the finite element approximation $\mathbf{u}_h$ defined using the basis functions and the computed nodal degrees of freedom must "converge" to the exact solution $\mathbf{u}$ when the mesh is refined. The subscript h is a parameter measuring the mesh size of the elements in the mesh, typically the smallest length of an element edge or the radius of the smallest inscribed circle/ball. One say that $\mathbf{u}_h$ "converge" towards $\mathbf{u}$ if $\mathbf{u}_h$ becomes "close" to $\mathbf{u}$ whenever the mesh size h approach zero. One way to measure the difference of two functions is to introduce a *norm*. One way to introduce a norm is to integrate the square of the function:

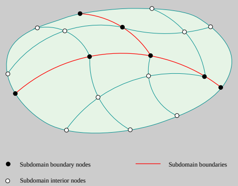
$$\|u^2\| = \int_V u^2 \, dV \tag{3.10}$$

In order to measure error in finite element methods derivatives are often introduced int the integral, since on would like to control certain derivatives of the functions too.

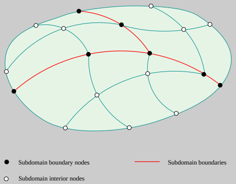
Basic requirement:

1. The basis functions must have complete polynomials of degree m. Note: for membrane elements linear polynomials are required.
2. Compatibility functions across inter element boundaries, i.e. $(m-1)$-continuity. Note: for membrane elements the displacements must be continuous.

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten



3. $m-$th derivatives of field variables must be represented when the element size becomes small. Note: for membrane elements constant strains must be represented exactly.

In addition one would like elements to be:

4. Geometric invariant. I.e. the solution should not depend on the orientation of the global coordinate system.

In the next Chapter methods where the compatibility condition can be relaxed.

3.7.2. The Patch test

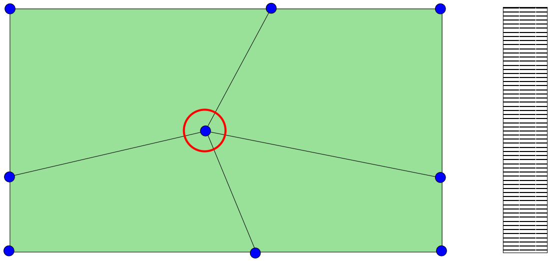
Irons, [Bazeley et al., 1965], in 1965 suggested a numerical test to check the validity of an element formulation. If this test, called the *Patch test*, is satisfied it is verified that the solution convergence to the correct solution. This statement is, and has been, controversial but the test is accepted in the engineering community.

The Patch test (procedure): . . . is a necessary and sufficient for convergence [Taylor et al., 1986][4]. We check that all rigid-body motions and constant strains are exactly represented for a set of

[4] R. L. Taylor, O. C. Zienkiewicz, J. C. Simo, and A. C. H. Chan. The patch test — A condition for assessing FEM convergence. *International Journal for Numerical Methods in Engineering*, 22:39–62, 1986.

Contents
◁◁ ▷▷
◁ ▷
Page 15 of 35
Go Back
Close
Quit

elements, called a *patch*.



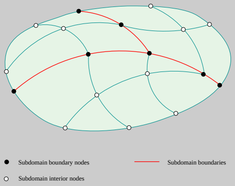
In a patch of elements at least one node must be in the interior of the patch.

The patch test can be performed in two ways:

1. Specify displacements on the boundary such that the displacement field in the patch consist of:
   (a) *rigid-body motions*, and
   (b) *constant strains*.
2. Specify a load giving:
   (a) constant stresses and strains, $(\sigma_c, \varepsilon_c)$
   (b) boundary conditions that prevent rigid-body motion (since most equation solvers do not handle singular matrices.)

The patch test is a consistency check, together with the stability(solvability) of the linear

Department of Mathematics University of Oslo



Contents
Page 16 of 35
Go Back
Close
Quit

system convergence is shown:

$$
Convergence \begin{cases} Consistence \\ Stability \end{cases}
$$

A stronger stability requirement is: *The element stiffness matrices must have full rank.* I.e. the rank of the matrix must be the number of degrees of freedom, n_e minus the number of rigid-body motions n_r:

$$
\text{rang}(\boldsymbol{k}) = n_e - n_r
$$

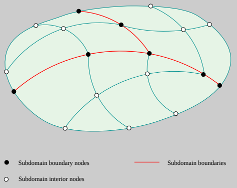
The patch test as described check the geometry of one patch, if the patch is arbitrary it is assumed that it is also valid for other patches.

In 1976 a simple element test was introduced in order to check that an element satisfy the patch test, *the individual element test*, [Bergan and Hanssen, 1976].

The weak patch test: ...a weak form of the patch test has also been introduced:

...a patch where all the elements are parallelograms.

Elements that do not pass the patch test: ...should not be used in structural analysis.



3.8. Element quality

The goal is to obtain sufficient accuracy of the analysis as efficiently as possible. Usually one think of efficiency as computational efficiency, but it also makes sense to consider the man-time involved.

Note: The quality of a finite element analysis depend on the chosen finite element and also an the geometry of the elements. An a-priori, i.e. without knowledge of the solution, a qualitative assessment of element quality is impossible.

Fortunately, in connection to structural analysis a-priori knowledge of the solution exist and experience in the choice of elements are gained over the years. The comments below are related to structural analysis.

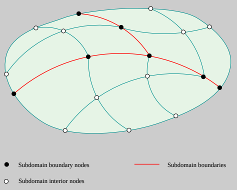
1. Square elements is better than triangular elements.
2. Geometricly "distorted" elements result in a stiffer element.

For lower order elements:

1. Shear locking can be a problem and bending is poorly represented.

For nine node Lagrangian and eight node Serendipity elements:

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

1. Rectangular elements give identical results.
2. 3×3 integration is stiffer than 2×2.
3. 8 node element are more sensitive to the geometry.

Conclusion: The ideal element is

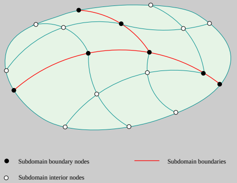
1. Quadratic,

Use of "bad" elements:

... concentrate the "bad" elements.

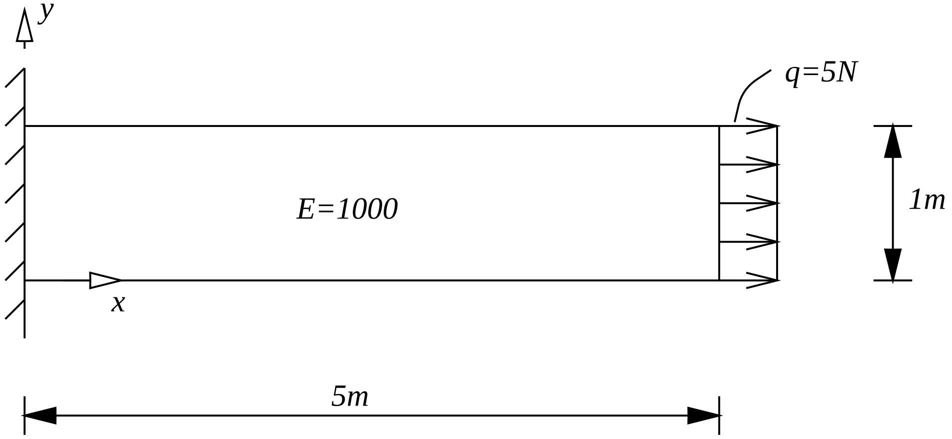
... in doubt? Compute on several different meshes, mesh refinement. (Adaptive methods?).

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

3.9. Bjelke (PLANE42)



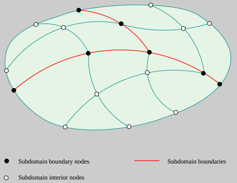
Problem: Den fast innspente bjelken som er vist i figuren skal modelleres med 4-knutepunkts membranelementer som har 2 frihetsgrader pr. knutepunkt. Bjelken skal diskretiseres slik at arealene til alle elementer skal være $0.25 \times 0.25 m^2$ (dvs. 80 elementer).

Foreta elementmetodeanalyse der lasten påføres som:

i) vist i figuren (flatestrekk).

ii) 5 punktlaster ($P = 1kN$) på de fem endeknutepunktene.

Department of Mathematics University of Oslo



Hvilken av disse to belastninger modellerer det som figuren viser mest nøyaktig?

Hva er σ_x og ϵ_x ved $x = 2.5\,m$, $y = 0.5\,m$ og $x = 4.5\,m$, $y = 0.5\,m$? Kommenter nøyaktigheten av disse verdiene.

Løsning:

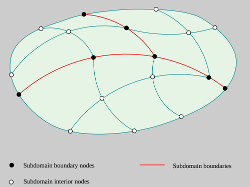
```
/BATCH,LIST
/FILNAM,ex413
/TITLE,Lineær statisk analyse av en bjelke

/PREP7               ! Preprossessoren
ET,1,42              ! PLANE42 membranelementer (plan spenning)
MP,EX,1,1e3          ! E-modulen

!Geometri ("solid modelling")
K,1                  ! origo
K,2,0,1e3
K,3,5e3,1e3
K,4,5e3,0
A,1,2,3,4            ! Areal lages (linjer lages automatisk)

!Inndeling i elementer
LESIZE,ALL,250
AMESH,1
FINISH

/SOLU
ANTYPE,STATIC
NSEL,S,LOC,X,0               ! Velger alle knutepunkt med x-koord lik null
D,ALL,ALL                    ! Opplagring ved innspenningen
NSEL,S,LOC,X,5e3             ! Velger alle knutepunkt med x-koord lik 5m
ESLN,S,0                     ! Velger alle elementer med minst ett knutepunkt i valgt sett
```

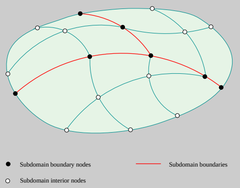
```
SFE,ALL,2,PRES,,-5
ALLSEL
LSWRITE,1
SFEDEL,ALL,ALL,ALL          ! Få bort den jevnt fordelte lasten
NSEL,S,LOC,X,5e3
F,ALL,FX,1e3                ! Punktbelastningen
ALLSEL
LSWRITE,2
LSSOLVE,1,2                 ! Løsningsprosedyren
FINISH

/POST1
SET,1,1             ! Last inn analyseresultatene for jevnt fordelt last
PLDISP,1            ! Deformert geometri
NSEL,S,NODE,,NODE(5000,500,0)
PRNSOL,S,X
SET,2,1             ! Resultat for punktbelastningen
PRNSOL,S,X
ALLSEL
FINISH
```

Kommentarer:

- Her er det også brukt `SELECT` logikk ved belastning av elementkantene og knutepunktene på enden av bjelken.
- Bruk av kommandoen `LSWRITE` gjør at de to belastningstilfellene kunne kjøres uten å måtte gå inn og ut av `/SOLU`.

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

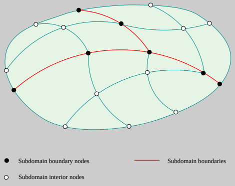
Svar på spørsmålene Fra den deformerte geometrien til de to lasttilfellene ser man at belastningen med flatestrekk er å foretrekke fordi endetversnittet forblir rett.

Belastningene ga følgende σ_x og ϵ_x verdier:

		Belastning	
		Flatestrekk	*Punktlast*
Kn. pkt.	$2.5, 0.5$	$\sigma_x = 5.000 \quad \epsilon_x = 0.5 \times 10^{-2}$	$\sigma_x = 4.999 \quad \epsilon_x = 0.4999 \times 10^{-2}$
koord.	$4.5, 0.5$	$\sigma_x = 5.000 \quad \epsilon_x = 0.5 \times 10^{-2}$	$\sigma_x = 4.219 \quad \epsilon_x = 0.4069 \times 10^{-2}$

Unøyaktigheten av σ_x og ϵ_x med punktbelastningen viser tydelig St. Venants effekten - jo lengre bort fra enden, så blir spennings- og tøyningsfordelingen mer lik de riktige verdier.

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten

Contents

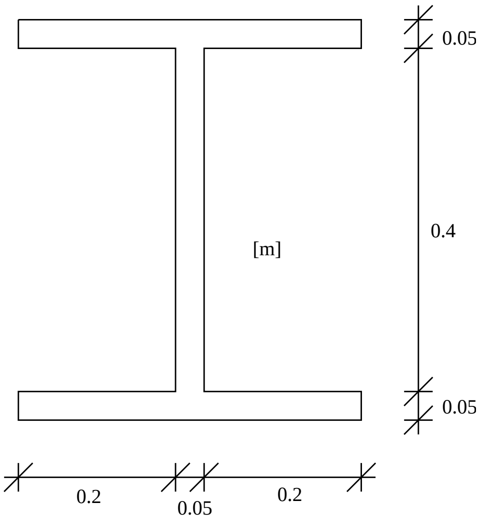
Page 23 of 35

Go Back

Close

Quit

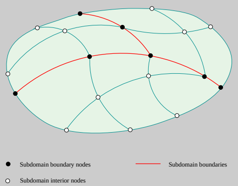
3.10. I-bjelke (SOLID45)



$$E = 2 \times 10^{11} N/m^2$$

$$I = 1.936 \times 10^{-3} m^4$$

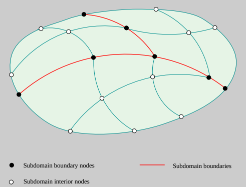
Problem: Bjelken analysert med bjelkeelementer i Kapittel 2 har nå tversnitttet som er vist i figuren.



Foreta en lineær statisk analyse av konstruksjonen med ANSYS. Er vertikalforskyvningen av punkt A (midten av tversnittet) samme som i Kapittel 2? Kommenter en eventuell forskjell mellom de to verdiene.

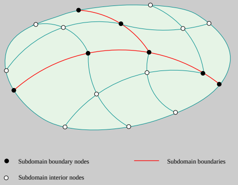
Løsning:

```
/BATCH, LIST
/FILENAME, ex414
/TITLE, Statisk lineær analyse av I-bjelke
/PREP7
ET,1,45
ET,2,4                ! Et "hjelpeelement" for momentbelastningen
R,2,2e3,1e10,150,150
MP,EX,1,2e5
K,1 $ K,2,200         ! I-tverrsnittet
K,3,250 $ K,4,450     ! $-tegnet seperer kommandoene
/PNUM,KP,1            ! "keypts" vises ved "solid modell" plott (kplo,aplo..)
/VIEW,1,1,1,-1        !
/ANGLE,1,-90,YM,1
KGEN,2,ALL,,,,50      ! Generering av "keypoints"
KGEN,2,ALL,,,,450
A,1,2,6,5             ! Arealer
A,2,3,7,6
A,3,4,8,7
A,9,10,14,13
A,10,11,15,14
A,11,12,16,15
A,6,7,11,10
K,,,,20e3             ! Punkt D
L,1,17
VDRAG,1,2,3,4,5,6,23  ! Generering av volum fra arealer
VDRAG,7,,,,,,23
ESIZE,500             ! Maks. element stoerrelse lik 500
```

```
VMESH,ALL
TYPE,2 $ REAL,2       ! "hjelpeelementet"
E,NODE(200,50,12e3),NODE(200,450,12e3)    ! ved punkt C
NUMMRG,ALL            ! To noder paa samme punkt blir en
NUMCMP,ALL            ! Komprimer numerering
NSEL,S,NODE,,108      ! Bjelkeelement med 2 knutepunkter
NSEL,A,NODE,,355
CERIG,355,ALL         ! Kobling av bjelke og volumelementer
NSEL,ALL
FINISH
/SOLU
NSEL,,LOC,Y,0         ! "select" for aa paafoere last og sette
NSEL,R,LOC,Z,4e3      ! Grensebetingelser
D,ALL,UY,,,,,UX,UZ    ! Fastholdt på undersiden av bjelken ved B
NSEL,,LOC,Y,0
NSEL,R,LOC,Z,20e3     !    ved D
D,ALL,UY,,,,,UX
NSEL,ALL
NSEL,S,LOC,Y,460,500
NSEL,R,LOC,Z,0,3900
ESLN,S,0                   ! Oversiden av AB
SFE,ALL,1,PRES,,-.3333333
ALLSEL
F,NODE(200,450,12e3),MX,12e9    ! Momentbelastningen
SOLVE
SAVE
FINISH

/POST1
SET
/SHOW, ex414pl
PLDISP,1
PRERR
FINISH
```

Svar/kommentarer: Et bjelkeelement er brukt som ”hjelpeelement” for påføring av momentbelastningen. Bjelkeelementets frihetsgrader er koblet sammen med volumelementene gjennom kommandoen `CERIG` (som lager koblingsligninger - jfr. kurset *MEK4550*).

Nei, nedbøyningen av punkt A (ca. $95mm$) er ikke den samme som i Kapittel 2 .

Denne konstruksjon er mye stivere (enn den i Kapittel 2). `PRERR` kommandoen gir en tilnærmet verdi (i %) av feilen som dårlig ”mesh” (geometri,type og inndeling av elementene) i konstruksjonen utgjør i resultatet. Denne verdien i dette ekempelet er på 49.645% - som er veldig høy. Feil kan også oppstå ved måten punktbelastningen påføres. Den deformerte bjelken vises i figuren nedenfor.

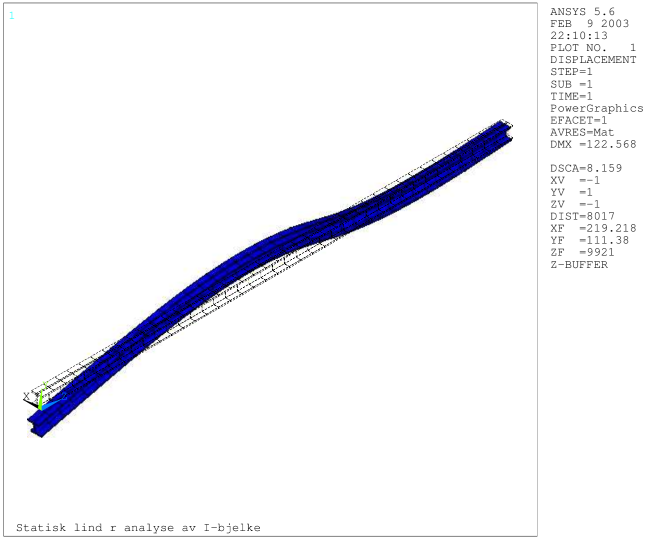
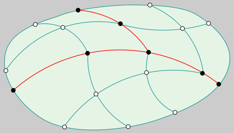


Figure 3.1: Deformert og original geometri av I-bjelken



Øving 3.1

Hva er Patch testen, og hva er formålet med denne testen?

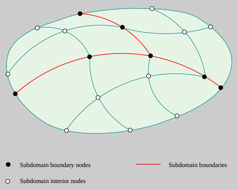
Sett opp et komplett Patch test problem for et stavelement, der alle randkrav settes på som foreskrevne forskyvninger. Angi hva som indikerer om testen er oppfylt eller ikke. Tøyningsrelasjonen, materialloven og den totale potensielle energien for stavproblemet er gitt ved:

$$\varepsilon_{xx} = \frac{du}{dx}$$
$$N_{xx} = EA\varepsilon_{xx}$$
$$\Pi(u) = \frac{1}{2}\int_{\ell} N_{xx}\varepsilon_{xx}\,dx - \int_{\ell} qu\,dx$$

Øving 3.2

Figuren viser et rektangulært element med 8 frihetsgrader.

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

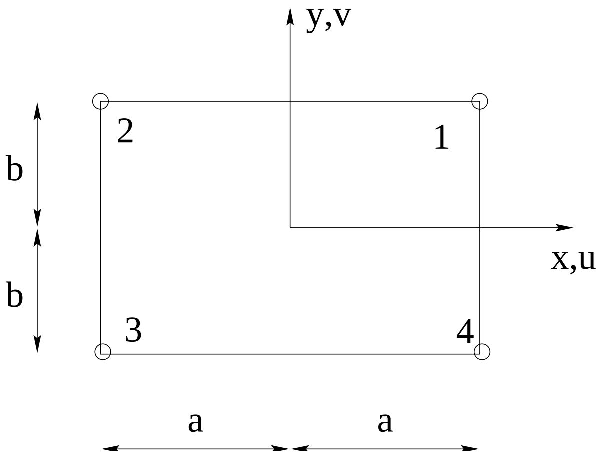


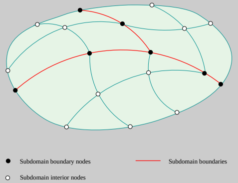
Figure 3.2: Rektangulært skiveelement.

Til forskjell fra det kompatible forskyvningsfelt som vanligvis benyttes, foreslås følgende forskyvningsfelt:

$$\begin{bmatrix} u_x \\ u_y \end{bmatrix} = \frac{1}{2} \begin{bmatrix} 1 & \xi & \eta & \xi\eta & 0 & 0 & 0 & \frac{b}{2a}(1-\eta^2) \\ 0 & 0 & 0 & \frac{a}{2b}(1-\xi^2) & 1 & \xi & \eta & \xi\eta \end{bmatrix} \begin{bmatrix} \mathbf{q}_x \\ \mathbf{q}_y \end{bmatrix}$$

a) Beregn tøyningsvektoren.

b) Hvordan skal en kombinere koeffisientene i $\mathbf{q}$ for å framstille ren rotasjon om origo?



c) Studer elementets egenskaper ved ren bøying om de to aksene. Sammenlign resultatet med et vanlig rektangulært element med 8 frihetsgrader.

d) Beregn stivhetsmatrisene $\mathbf{k}_q$ for de to elementene. Kan man av dette slutte noe om størrelsen av forskyvningene ved beregning av de to forskjellige matrisene?

Øving 3.3

Figuren viser et volumelement med 8 knutepunkter.

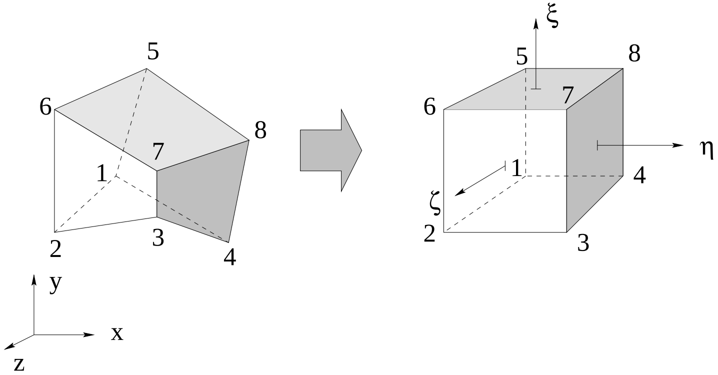
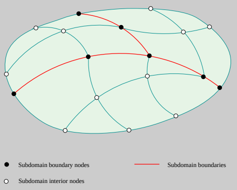


Figure 3.3: Isoparametrisk åttenoderselement.

a) Finn $\boldsymbol{N}$ uttrykket ved η, ξ, ζ

b) Uttrykk forskyvningene i elementet ved hjelp av de generaliserte forskyvningsmønstre $\boldsymbol{N}_q$



c) Finn de tilhørende tøyningene $\boldsymbol{\varepsilon}$.

d) Klassifiser de generaliserte forskyvningsmønsterene i $\boldsymbol{N}_q$ etter:

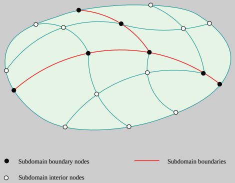
- Stivlegememønstre.
- Konstante tøyningsmønstre.
- Høyere ordens mønstre.

e) Vis hvordan Jakobimatrisen ser ut. $(\Delta_{\eta\xi\zeta} = \boldsymbol{J}\Delta_{xyz})$

Øving 3.4

I denne oppgaven skal vi benytte *COMSOL Multiphysics* og multi fysikk modellering. Vi skal igjen se på utkragebjelken fra Kapittel 1 sammen med redusert skjær integrasjon. Problemet er vist i Figure 3.4.

Department of
Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

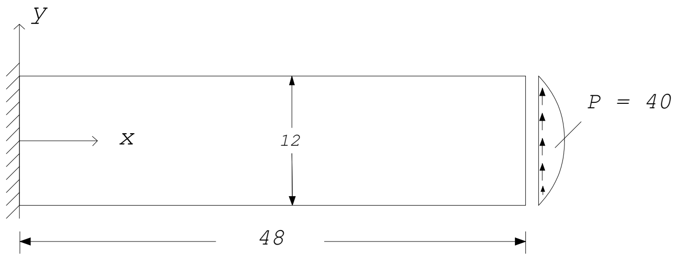


Figure 3.4: Utkragerbjelke med endelast. Geometriske data er gitt i figuren, samt endelastens størrelse. Materialdata: $E = 30000$, $\nu = 0.25$. Tykkelsen, $t = 1$.

Vi skal benytte et lineært Lagrange element sammen med et 4×1 elementnett.

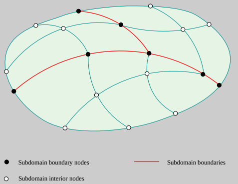
- Start *COMSOL Multiphysics* og velg *Multiphysics* i **Model Navigator**. Legg til to versjoner av samme problem: *Structural Mechanics Plane Stress* (smps).
- Velg en av de to problemene og generer geometri, randkrav og laster.
- For den ene modellen så setter vi skjærstivheten lik null. Må benytte *Anisotropic material*.

- For den andre modellen så setter vi de komponenten som ikke er knyttet til skjær lik null. Her velger vi også en redusert integrasjon for elementet, ettpunkts integrasjon.

- Vi må koble de to lingningssetene sammen. Det gjør vi for hele domenet.

- Hvordan sammenligner resultatene med de for full integrasjon.

Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

Contents
Page 34 of 35
Go Back
Close
Quit

A. References

[Bazeley et al., 1965] Bazeley, G. P., Cheung, Y. K., Irons, B. M., and Zienkiewicz, O. C. (1965). Triangular elements in bending — Conforming and non-conforming solutions. In *Proceedings of Conference on Matrix Methods in Structural Mechanics*, Ohio. Air Force Institute of Technology, Wright Patterson Air Force Base.

[Bergan and Hanssen, 1976] Bergan, P. and Hanssen, L. (1976). A new approach for deriving good finite elements. In Whiteman, J., editor, *The Mathematics of Finite Elements and Applications — Volume II*. MAFELAP II Conference, Brunel University, 1975, Academic Press, London.

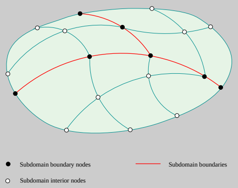
[Cook et al., 2002] Cook, R. D., Malkus, D. S., Plesha, M. E., and Witt, R. J. (2002). *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition.

[H.MacNeal, 1978] H.MacNeal, R. (1978). A simple quadrilateral shell element. *Computer & Structurer*, 8:175—183.

[Strang and Fix, 1973] Strang, G. and Fix, G. J. (1973). *An Analysis of the Finite Element Method*. Prentice-Hall, Englewood Cliffs, New Jersey.

[Taylor et al., 1986] Taylor, R. L., Zienkiewicz, O. C., Simo, J. C., and Chan, A. C. H. (1986). The patch test — A condition for assessing FEM convergence. *International Journal for Numerical Methods in Engineering*, 22:39–62.

Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten