DISPENSA

FEM in MSC. Nastran

components

preprocessing:

mesh generation
material definitions
definition of loads and boundary conditions

solving:

solving the (linear) set of equations

postprocessing:

visualisation and analysis of results
(primary and secondary field variables)

primary	secondary
displacement	stress/strain
temperature	heat flux
acoustic pressure	velocity/intensity
…	…

preprocessing

- co-ordinate systems
- nodes
- elements
- geometrical properties
- material properties
- units
- loads and constraints

illustration for MSC/NASTRAN

preprocessing

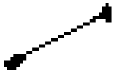
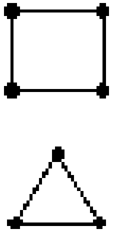
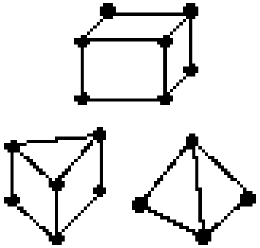
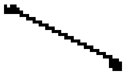
nodes

- nodes are called GRID points in NASTRAN
- grids are defined as points in space that have :
  - a unique number (integer)
  - a certain location X,Y,Z
    - coordinate systems aid in locating point
  - 6 Degrees Of Freedom (DOFs) to move in space
    - coordinate systems aid in interpreting displacement results

- GRID definition statement :
  - GRID ID CP X Y Z CD
  - where
    - ID : identification number
    - CP : reference to coordinate system that was used to position the grid
    - X,Y,Z : co-ordinates
    - CD : reference to coordinate system in which the input (loads, BC) and output (displacements) are defined

preprocessing

elements

Category	Spring Elements	Line Elements	Surface Elements	Solid Elements	Rigid Elements
Physical Behavior	Simple Spring	Rod, Bar, Beam	Membrane, Thin Plate	Thick Plate, Brick	Rigid Bar
MSC/NASTRAN Element Name	CELAS2*	CONROD* CROD CBAR	CQUAD4 CTRIA3	CHEXA CPENTA CTETRA	RBE2*
Associated Property Entry	None Required	PROD PBAR	PSHELL	PSOLID	None Required
					

preprocessing

geometry

3D Solid Elements	2D Surface Elements	1D Line Elements
<none>	Plate/Shell Thickness	Beam orientation (3th point) Beam cross section properties

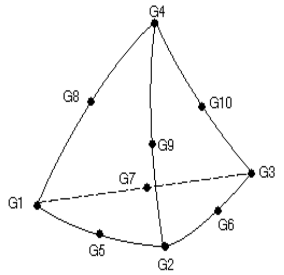
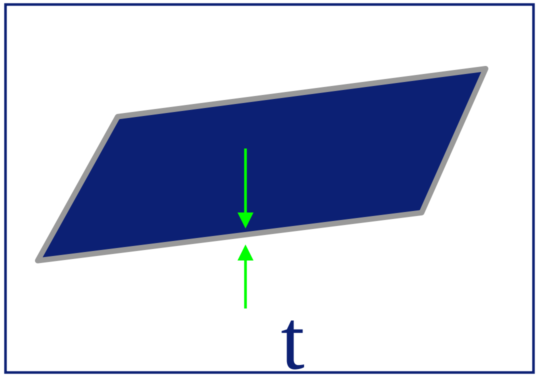
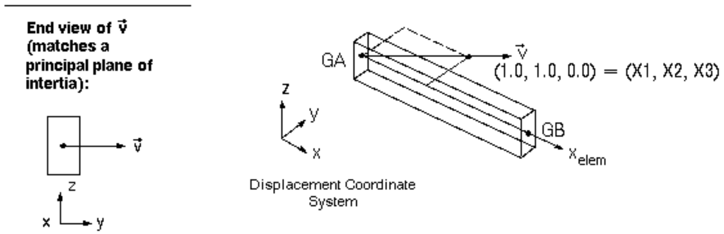
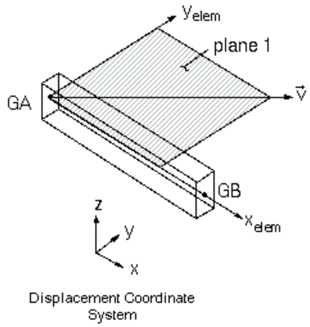


Figure 6-19. CTETRA Element Connection.





STEP 3 The plane formed by the element x-axis and orientation vector $\vec{v}$ is called plane 1. The element y-axis lies in plane 1 and is perpendicular to the element x-axis, as shown below:



preprocessing

material properties

- Basic Material Property Definitions :
  - *linear* : deformation are directly proportional to the applied load
  - *elastic* : an elastic structure returns to its original, undeformed shape when the load is removed
  - *homogeneous* : properties are independent of location within the material
  - *isotropic* : material properties do not change with the direction of the material

- MATERIAL definition statement :

  MAT1 ID E G NU RHO ... … GE
  - where
    - ID : identification number
    - E : Young's modulus
    - G : Shear modulus G = 0.5 * E / (1 + NU)
    - NU : Poisson's ratio
    - RHO : Mass density
    - GE : structural damping coefficient

preprocessing

units

- most FE solvers do not have an explicit notion of physical units.
- it is the user's responsibility to use a consistent set of units.
- popular unit sets : SI, English Units
- If the units are not known, try to estimate them from :
  - the grid coordinates (if you know the dimensions of the structure, you should be able to deduce the length unit)
  - the material definition (for known materials such as steel, aluminum, ….)
- Common mistakes in FE models originate from wrong material values (due to wrong unit conversions) !!

preprocessing

loads

- Static Loads :
  - concentrated loads applied to grid points (FORCE, MOMENT)
  - distributed loads on line elements (PLOAD1)
  - normal uniform pressure loads on surface (PLOAD, PLOAD2)
  - normal pressure load on face of 2D or 3D element (PLOAD4)
  - gravity or acceleration loads (GRAV)
- Enforced displacement (SPCD)

preprocessing

loads

- Dynamic Loads :
  - concentrated loads applied to grid points :

$$P(f) = A.\left[C(f) + i.D(f)\right].e^{i(\theta - 2\pi f \tau)}$$

  - RLOAD1 or RLOAD2 statement that refer to
    DAREA statements (spatial definition of load : A)
    2 TABLED1 statements (spectral definition C(f),D(f)
    real/imag for RLOAD1, amplitude/phase for RLOAD2)
  - Selection of dynamic loading with DLOAD case control statement (reference to RLOAD1 / 2)

preprocessing

constraints

- a constraint is the enforcement of a prescribed displacement on a single grid point or a set of points

- two basic types of constraints :
  - single point constraints (SPCs) :
    - enforces a displacement (for example zero displacement) to a single point
  - multiple point constraints (MPCs)
    - enforces a mathematical constraint relationship between one grid point and a set of grid points

solution sequence for mode calculation

solver

$$\left(\left[K\right]+j\omega\left[C\right]-\omega^2\left[M\right]\right).\left\{X\right\}=\left\{F\right\}$$

- undamped
- no external forces

$$\left[K\right].\left\{\Phi_m\right\}=\omega_m^2\left[M\right].\left\{\Phi_m\right\}$$

ω_m : eigenfrequencies (# modes = total # dofs n)

Φ_m : eigenmodes (each eigenvector has size (nx1))

- mode calculation = standard eigenvalue problem

$$\left[M\right]^{-1}\left[K\right].\left\{\Phi_m\right\}=\lambda_m.\left\{\Phi_m\right\}$$

Lanczos algorithm :

iterative procedure to determine a subset of modes

solution sequence for dynamic response analysis

solver

$$\left([K]+j\omega[C]-\omega^2[M]\right).\{X\}=\{F\}$$

1. direct solution method:

- solving the FE matrix equation directly for the unknown nodal dofs
- dedicated large model solvers that fully benefit from matrix properties
- back-substitution of result vector {p} into field variable approximation

2. modal solution method:

- projecting the original dofs onto a modal base
- that possibly leads to some substantial model size reduction

solution sequence for dynamic response analysis

solver

2. modal solution method:

$$\left(\left[K\right]+j\omega\left[C\right]-\omega^{2}\left[M\right]\right).\left\{X\right\}=\left\{F\right\}$$

2.1. calculating the undamped modes

$$\left[K\right].\left\{\Phi_{m}\right\}=\omega_{m}^{2}\left[M\right].\left\{\Phi_{m}\right\}$$

ω_m : eigenfrequencies (# modes = total # dofs n)

Φ_m : eigenmodes (each eigenvector has size (nx1))

2.2. projection of original dofs onto modal base

solution sequence for dynamic response analysis

solver

2. modal solution method:

2.3. construction of modal model

$$\left(\left[\tilde{K}\right] + j\omega\left[\tilde{C}\right] - \omega^2\left[\tilde{M}\right]\right).\{\phi_m\} = \{Q_m\}$$

$\{\phi_m\}$: *(m_a x1)* vector of unknown modal participation factors

$\left[\tilde{K}\right], \left[\tilde{C}\right], \left[\tilde{M}\right]$ diagonal matrices due to mode orthogonality !

2.4. solving modal model for unknown modal participation factors ϕ_m

2.5. back-substitution of result vector into field variable approximation

solution sequence for dynamic response analysis

solver

2. modal solution method:

- model size reduction: from *(nxn)* to *(m_axm_a)*
- accuracy depends on size of modal base m_a
  - if m_a=n : same accuracy is obtained with direct solution sequence
  - if m_a< n : possible gain in computational effort but loss in accuracy

'rule of thumb':

reasonable accuracy at some frequency ω requires a modal base that contains at least all modes with eigenfrequencies up to 2ω

⇨ *mainly at low frequencies (low modal density) the required number of modes can be substantially smaller than the original number of degrees of freedom*

solution sequence for transient analysis

solver

$$\{F(t)\} = [K]\{d(t)\} + [C]\{\dot{d}(t)\} + [M]\{\ddot{d}(t)\}$$

solver

- overview of some NASTRAN solution sequences:
  - SOL 101 : linear static analysis
  - SOL 103 : normal modes
  - SOL 107 / 110 : complex modes (direct/ modal)
  - SOL 108 / 111 : frequency response (direct/ modal)
  - SOL 109 / 112 : transient response (direct/ modal)
  - SOL 106 : non-linear statics followed by normal modes
  - SOL 200 : Design sensitivity and optimization

postprocessing

visualisation and analysis of results

(primary and secondary field variables)

primary	secondary
displacement	stress/strain
temperature	heat flux
acoustic pressure	velocity/intensity
…	…

! secondary variable approximations are less accurate then primary variable approximations !

How to read a nastran file

```
$ NASTRAN input file created by the MSC MSC.Nastran input file
$ translator ( MSC.Patran 12.0.041 ) on September 21, 2005 at 10:52:43.
$ Normal Modes Analysis, Database
SOL 103
CEND
SEALL = ALL
SUPER = ALL
ECHO = NONE
SUBCASE 1
$ Subcase name : Default
   SUBTITLE=Default
   METHOD = 1


stress=all
spc=2

BEGIN BULK
PARAM    POST    0
PARAM    AUTOSPC YES
PARAM   PRTMAXIM YES
EIGRL    1                       16      0
PSOLID  1       1
$ Pset: Property_1
CTETRA  1       1       5774    2133    6428    6367    24353   24376
        24380   35839   13174   17539
CTETRA  2       1       2196    2185    1526    2172    40246   18845
...............................................................

$ Material : Material_1
MAT1*    1               6.8+10                          .3
*        2900.


$ Nodes of the Entire Model
GRID*   1                               -0.02603926542250.06873768028045*A1
*A1     0.15908825799237
GRID*   2                               -0.02605234924510.07670429191051*A2
*A2     0.14779071433393
GRID*   3                               -0.02791697484020

$ Loads for Load Case : Default
SPCADD   2       1
$ Displacement Constraints of Load Set : vincoli
SPC1     1       123456  183     199     200     201     202     203
         204     225     226     227     228
SPC1     1       123456  264     THRU    277
$ Referenced Coordinate Frames
ENDDATA
```

FE modeller has a determining impact on prediction accuracy

idealization error

choice of underlying mathematical model (avoid singularities)

representative boundary condition modelling

representative load modelling

appropriate material modelling

discretization error

mesh quality: balance between accuracy and computational load (CPU and memory)

solution error

choice of solver

avoid singularities

TYPES OF SINGULARITIES ENCOUNTERED IN FE MODELS

	Type 1	Type 2
Stress	Infinite	Infinite
Strain energy	Finite	Infinite
Displacement	Finite	Infinite
Examples	Sharp re-entrant corner in 2-D Sharp re-entrant edge in 3-D Point load in 2-D Line load in 3-D	Point support in 2-D Edge support in 3-D

stress singularities

e.g. stress at sharp re-entrant corners

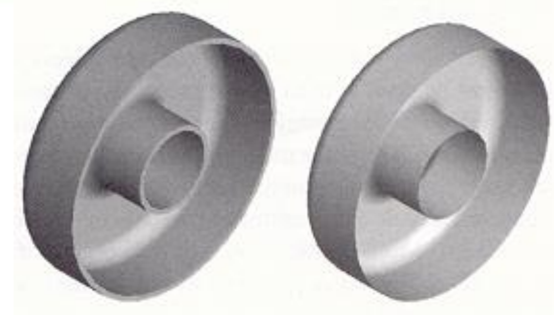
displacement singularities

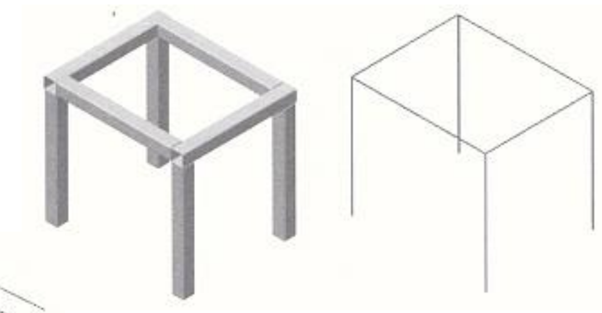
point (in 2D) or edge (in 3D) connections cannot withstand reaction forces (e.g. spotwelds !)

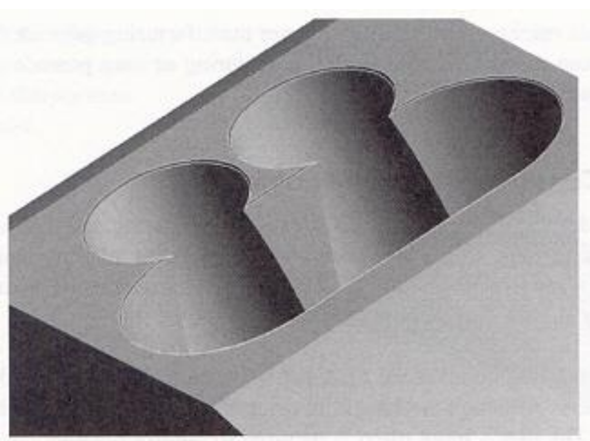
CAD geometry ≠ FE geometry

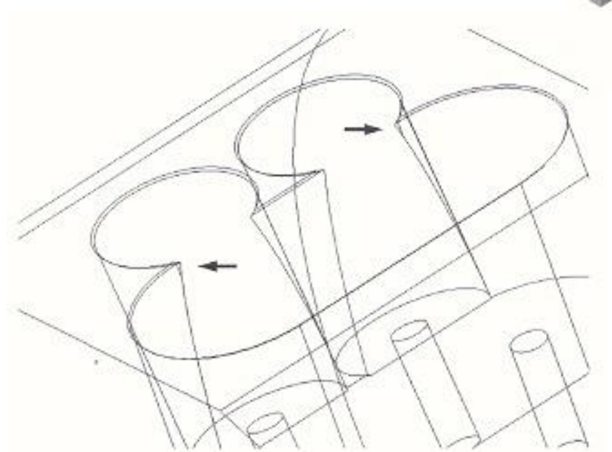
prior to meshing

- defeaturing
- idealization (e.g. shell versus solid)
- clean-up









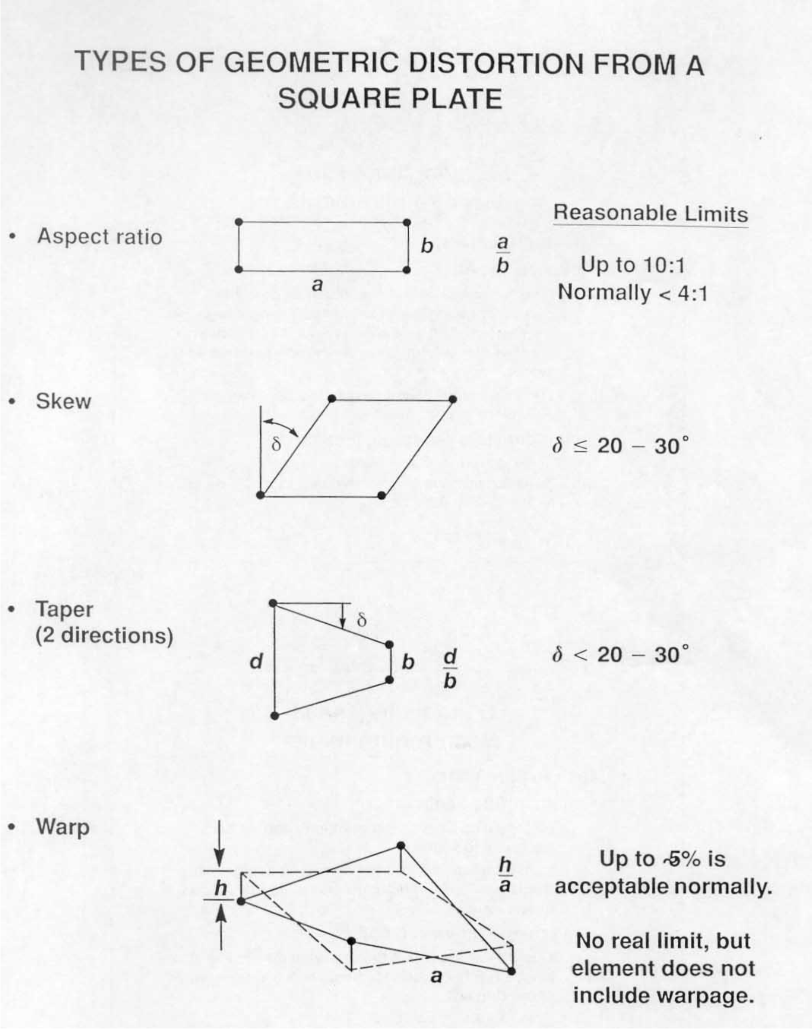
appropriate meshing

avoid distorted elements

low order p …
low distortion allowed

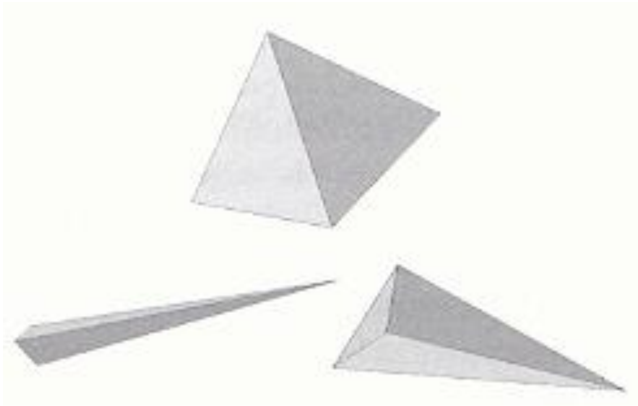
watch out with automatic meshers

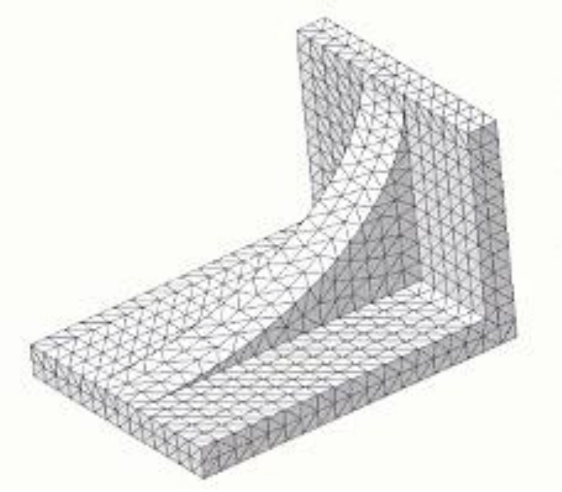
watch out with morphing



appropriate meshing

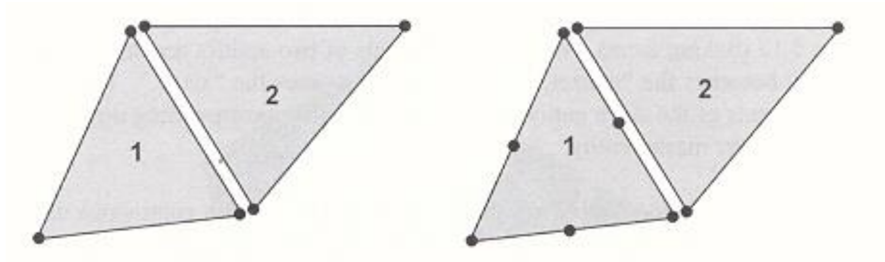
avoid distorted elements

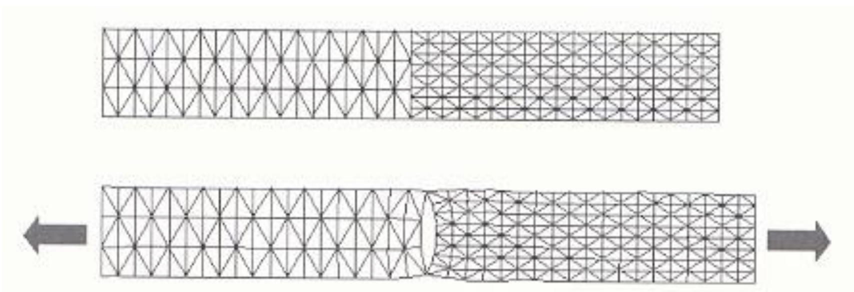


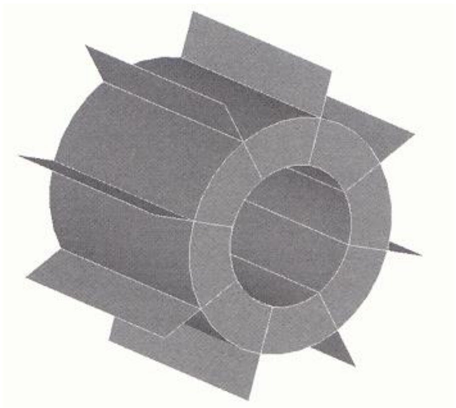


appropriate meshing

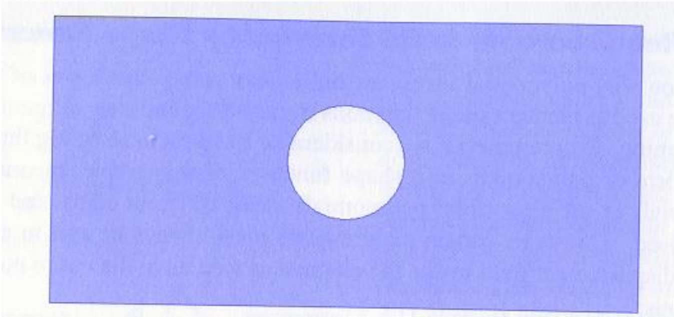
- avoid mesh incompatibilities



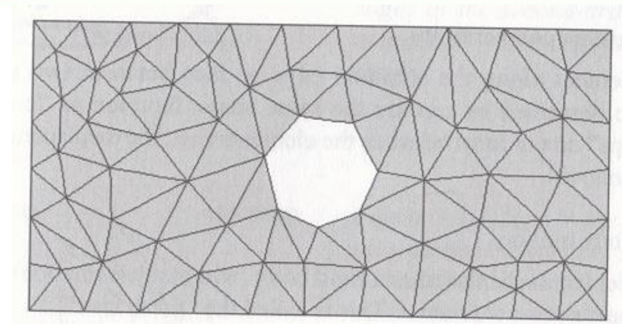




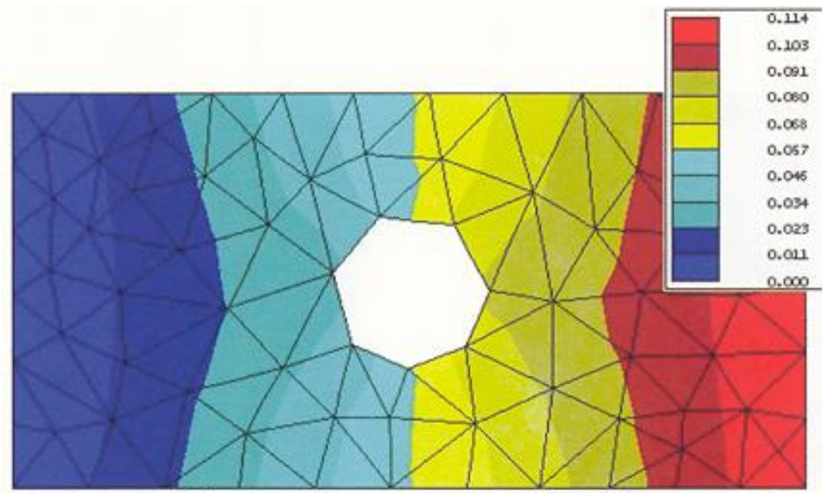
induce master/slave relations … no reliable stress evaluation



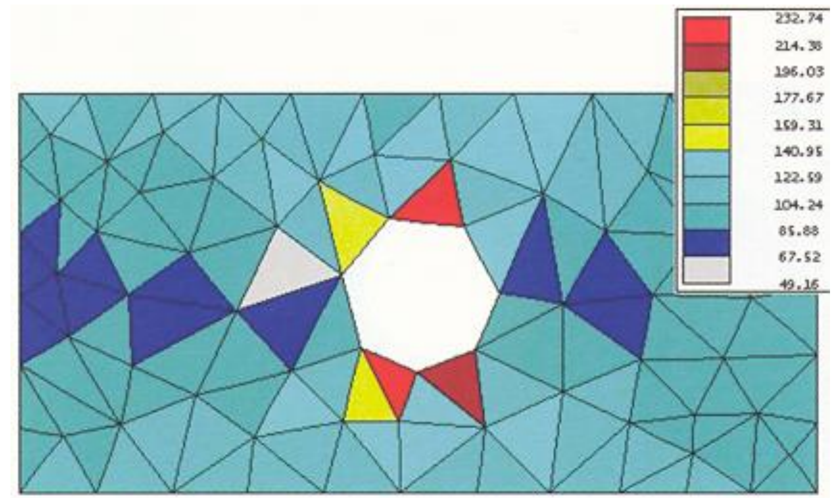
thin hollow plate
horizontal tensile load – constrained at left side



first-order triangular elements

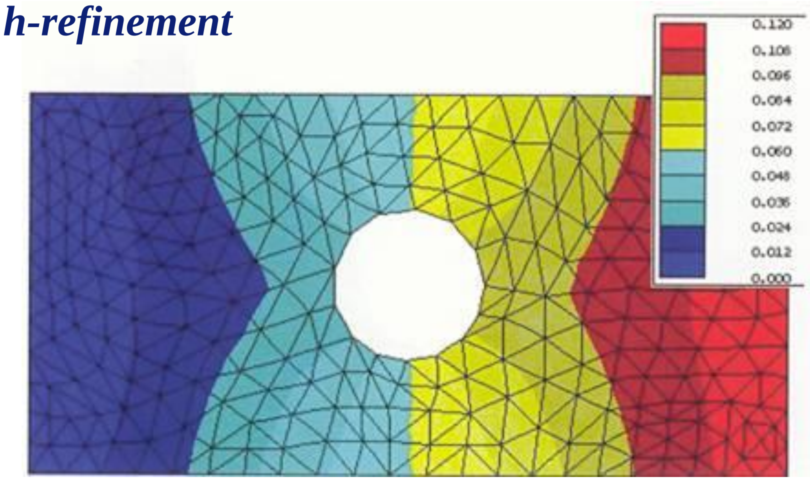


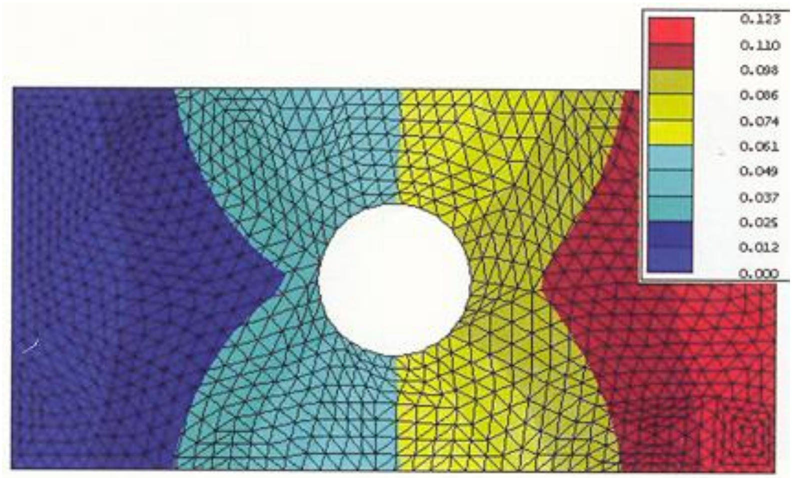
horizontal displacement (p=1)



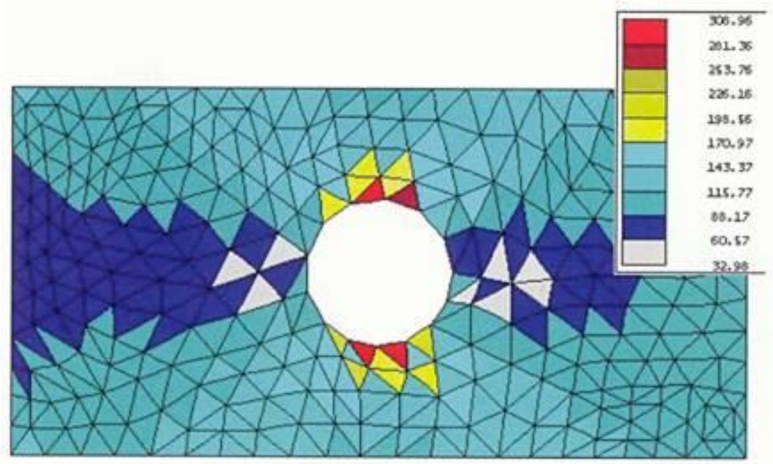
von Mises stress (p=1)

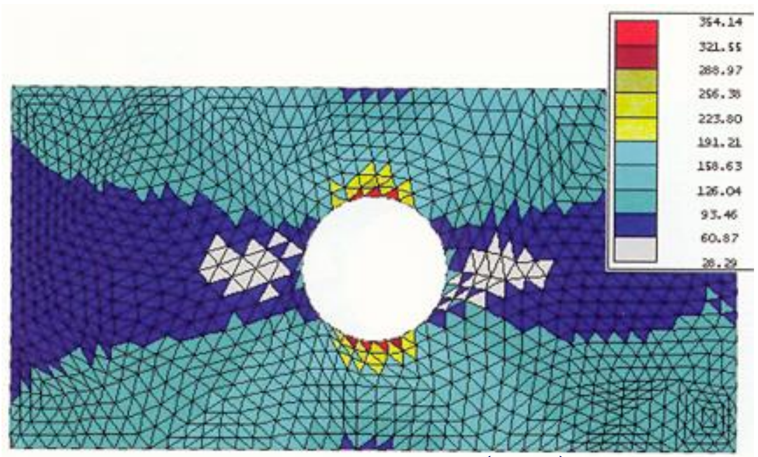
h-refinement





horizontal displacement (p=1)

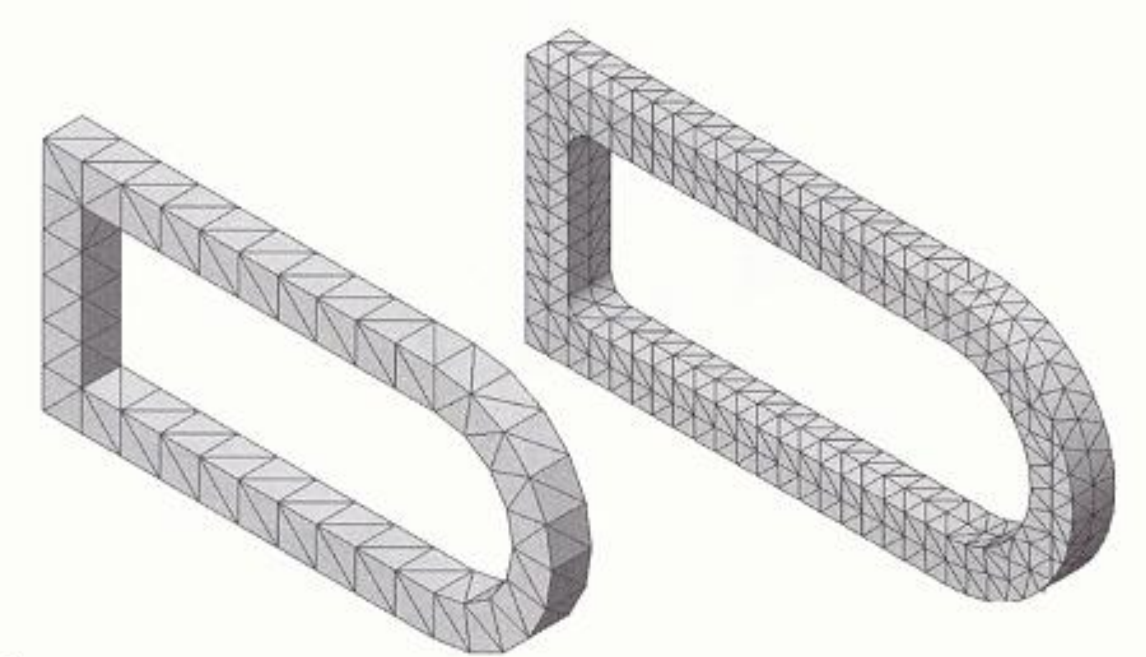


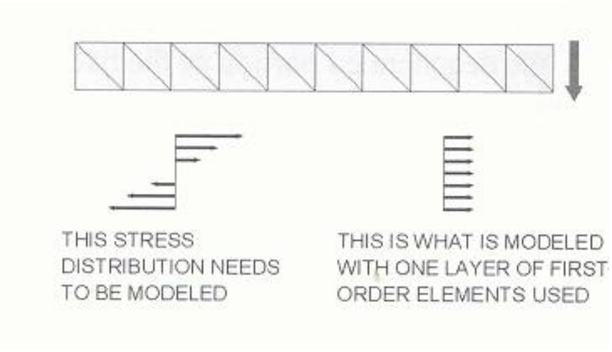


von Mises stress (p=1)

appropriate meshing

- solids for bending



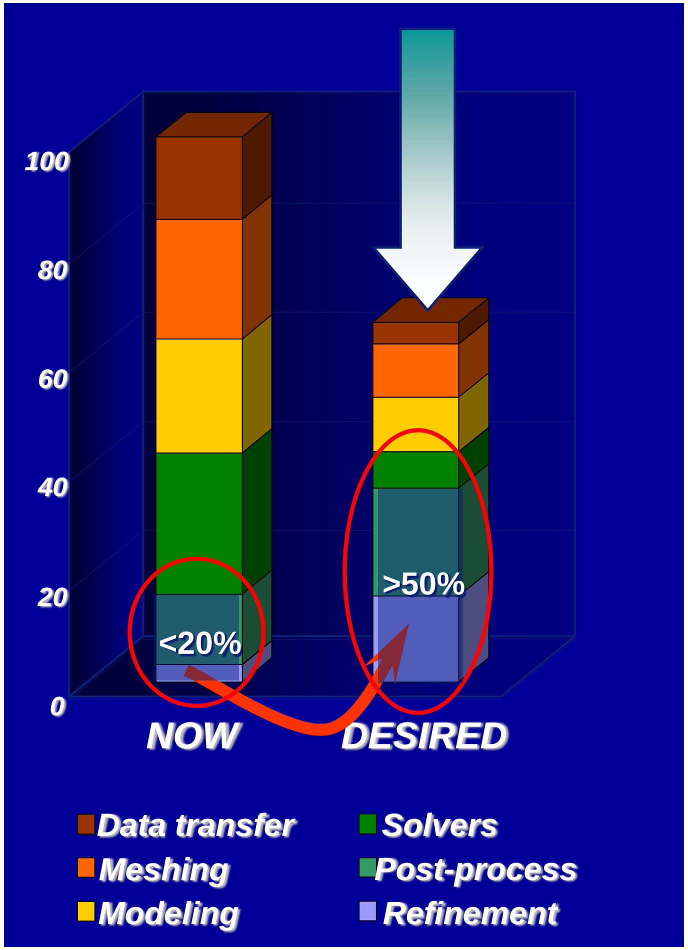


not OK

OK

Challenges

- **numerical modelling techniques**
  - enhanced computational efficiency
  - account for variability
  - advanced material models
  - ...
- **mesh generation process**
  - automation
  - (multi-physics) compatibility
  - re-use of models
  - morphing
  - ...
- **computer resources**
  - parallel computing
  - ...
- **data management**
  - exchange, sharing
  - interpreting, mining
  - ...



increasing "value-added engineering time"

Some reference books

- Paul M. Kurowski:
  'Finite Element Analysis for Design Engineers'
  (ISBN 0-7600-1140-X - SAE International - 2004)

- O.C. Zienkiewicz and R.L. Taylor: 'Finite Element Method Set'
  (ISBN 0470395036 - Butterworth-Heinemann - 2000)
  - volume 1 : 'The Basis'
  - volume 2 : 'Solid Mechanics'
  - volume 3 : 'Fluid Dynamics'