University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-04-01

PRACTICAL GUIDELINES FOR FE APPLICATIONS

Lecture Outline

1. **Introduction**
2. **Data Input for FE software**
3. **Accuracy and Convergence of FE Solutions**
4. **General Guidelines for Using FE Software**
5. **Preventing Rigid Body Motion**
6. **Examples of Good And Bad Practice**
7. **Summary**

1 Introduction

Inexperienced users of FE often struggle with the following questions:

- Which element type should be used? i.e. linear, quadratic, plane stress/plane strain, three-dimensional, beam, shell, etc.?
- How many elements should be used?
- How can the real-life boundary conditions be translated into data input?
- In the absence of other (non-FE) solutions to compare with, how can the accuracy of the FE software be gauged?

2 Data Input for FE software

To model a given problem using FE software, the user must specify:

- Geometry
- Material properties
- Analysis type
- Displacement boundary conditions
- Applied loads
- Element type
- Other information, such as the objective of the analysis

Setting up an analysis problem

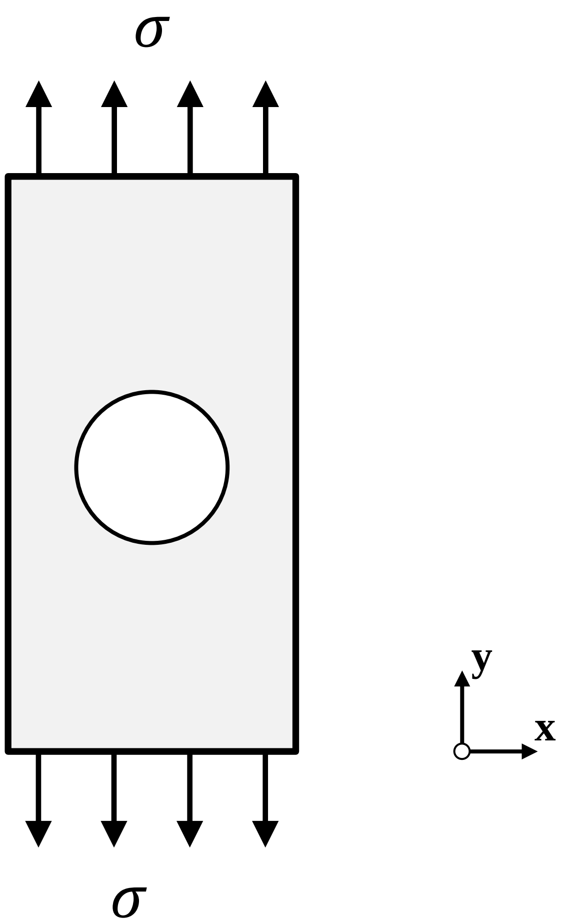


Figure 1: Perforated Plate Subject to Uniaxial Stress

Explanation

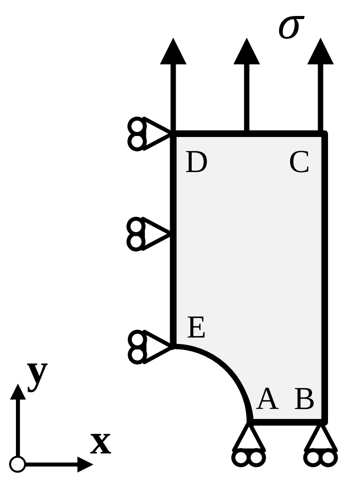
REF. NO.	Test001 (file name Test1.inp)	• Specify a unique reference number (and the file name containing the data input)
DESCRIPTIVE TITLE	Perforated plate analysis (symmetric quarter)	• Specify a unique descriptive title for the problem
GEOMETRY	 2D Plane strain Continuum elements Length of CD = 25 mm Hole radius = 20 mm Applied stress, σ = 100 N/mm^2	• Sketch the geometry • Show all displacement constraints • Show the applied loads • Specify the dimensionality (i.e. 2D plane strain/stress, 3D, axisymmetric, etc.) • Specify the configuration (e.g. continuum, beam , shell, plate) • Specify units (even if not used in the analysis)

Figure 1: A typical FE data input sheet

Explanation

MATERIAL PROPERTIES	$E = 250.0 \times 10^3$ N/mm^2 $\nu = 0.25$ (If plasticity is also considered , specify the yield stress and the uniaxial stress-strain curve)	• Specify all material properties relevant to the analysis • Specify all units used (this is particularly important for non-linear problems) • Specify relevant material law (e.g. plasticity or creep law)
ANALYSIS TYPE	Static Elastic-plastic analysis (non-linear material law)	• Specify relevant analysis required (e.g. elastic/plastic/creep, thermal, static/dynamic, linear/non-linear , etc.)
DISPLACEMENT BOUNDARY CONDITIONS	(a) Zero y-displacement (roller conditions) specified on line AB. (b) Zero x-displacement (roller conditions) specified on line DE.	• Write down the displacement constraints (referring to the sketch shown in the Geometry Section)
APPLIED LOADS	A uniform tensile stress (distributed load) specified at the top surface (line CD).	• Write down all the applied loads and their units (referring to the sketch shown in the Geometry Section)

Explanation

ELEMENT TYPE	8-node iso-parametric quadratic element with 2x2 Gauss integration points (CPE8R in ABAQUS)	• Write down the type of element used and the number of integration points (e.g. element code used in the FE software)
OTHER INFORMATION	Objective: to determine the stress concentration around the hole	• Write down any relevant data regarding the FE model, such as : - Objective of the analysis - Any special features, e.g. load applied in a number of load steps, initial conditions, etc.

3 Accuracy and Convergence of FE Solutions

Sources of Error in FE Analysis

(a) Modelling Errors:

These errors occur if the geometry is not exactly modelled or the boundary conditions are not accurately interpreted.

(b) Mesh Errors:

These errors occur if the mesh is not a "good" mesh, e.g. containing long thin elements, not refined in regions of sharp variation of variables, etc.

(c) Numerical Errors:

- Numerical round-off error in the computations
- The solution matrix may become "ill-conditioned"
- Numerical errors can also occur in the numerical integration procedures.

Convergence of FE Solutions

- The displacement-based FE formulation usually gives an **over-estimate of the true stiffness** of the element, i.e. elements are assumed **'over-stiff'.**

- Since stiffness multiplies the displacement to obtain the external force, the **displacements are under-estimated**.

- Since stresses are calculated from the displacement values, this means that usually **stresses are under-estimated**.

- If a very poor mesh is used, then it is possible that some nodal displacement values will be over-estimated while most others will be under-estimated.

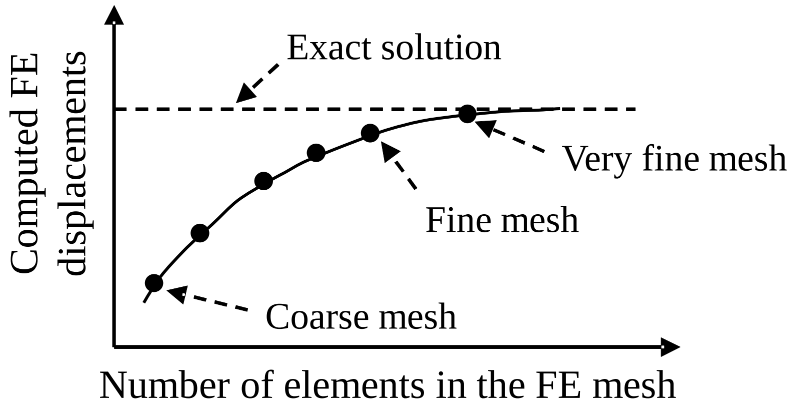
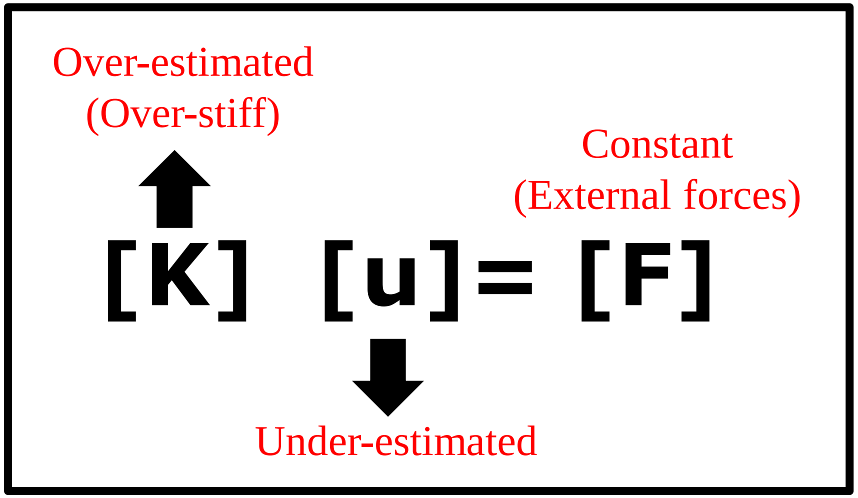


Figure 2: Convergence of FE Solutions

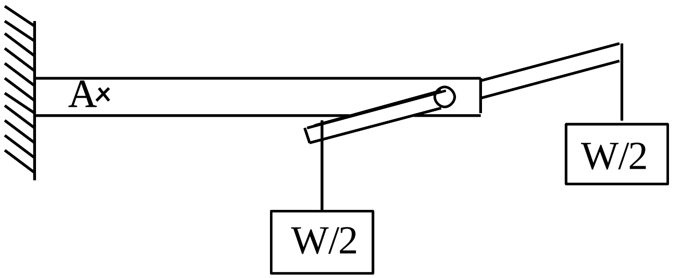


4 General Guidelines for Using FE Software

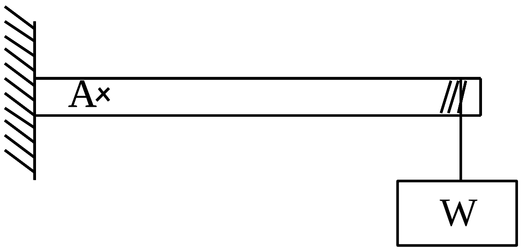
- Choose the correct element type
- Use a high quality mesh
- Avoid long thin elements
- Perform a mesh convergence study
- Check stress accuracy
- Avoid rigid body motion
- Check reaction forces
- Use a benchmark
- Ensure inter-element connectivity (if different types of elements are used)

The St Venant Principle

The "*St. Venant Principle*" states that if a structure is subjected to two statically equivalent load cases, then the stresses and displacements "remote" from the point of application of the load are unaffected by the details of the load application.



Load case (a)



Load case (b)

Figure 3: Example of the application of St. Venant's Principle

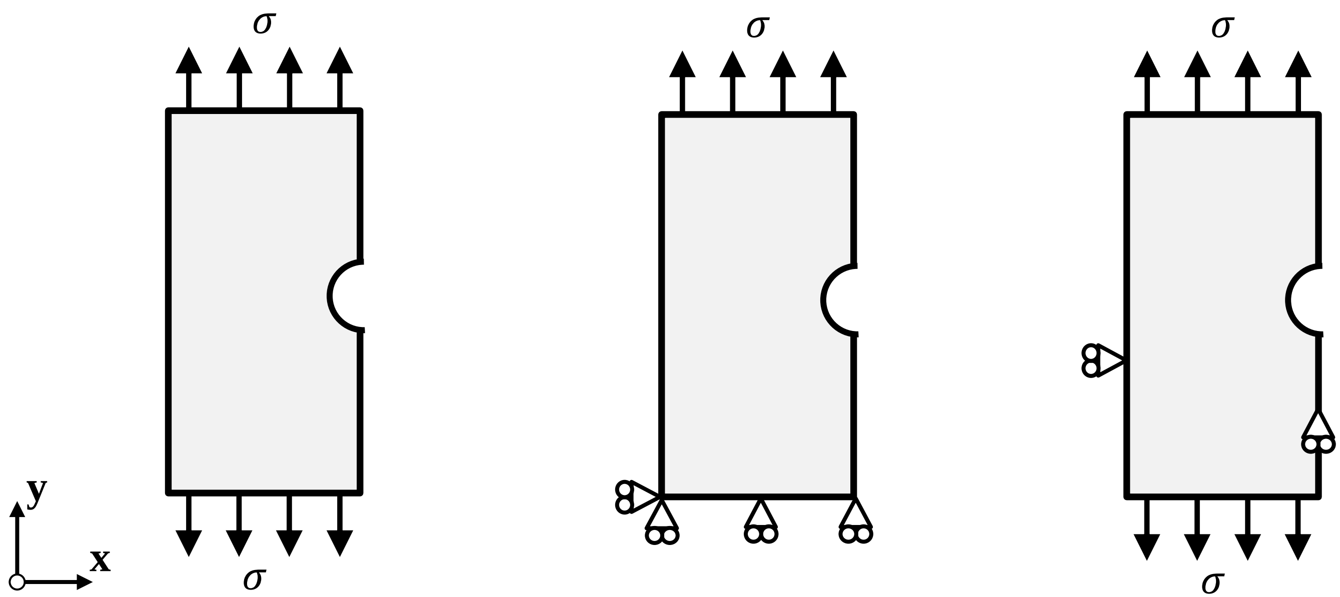
5 Preventing Rigid Body Motion

Why does rigid body motion occur?

- If a structure is insufficiently restrained, it may move as a whole, i.e. undergo rigid body motion, which would invalidate the FE solutions.
- The body should also be prevented from spinning freely about a pivotal point.
- Since the forces are in equilibrium, the summation of all forces in all directions must be zero. This would be true if an exact analytical solution is derived.
- However, since FE solutions are approximate due to the round-off error in the computational operations, the summation of all the nodal forces will never be exactly equal to zero, but equal to a very small negligible number, say 10^{-10} N in the y-direction.
- Since this is a non-zero net force, it will be sufficient to cause the body to move as a whole in the y-direction, thus invalidating the small deformation assumption.

How to prevent rigid body motion

- “Artificial” constraints may be imposed to prevent rigid body motion.
- The location of the constraint must be chosen to be as far away as possible from the region of interest (usually the region of stress concentration) - St Venant’s principle



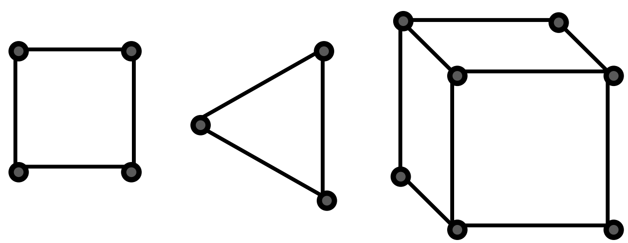
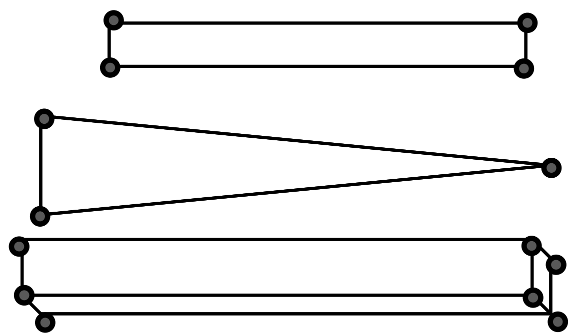
(a) Real-life Problem
(Rigid Body Motion Not Prevented)

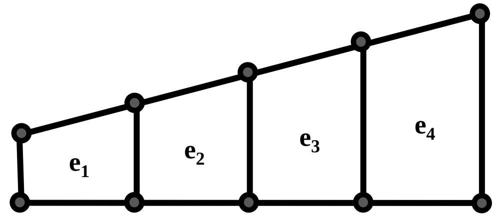
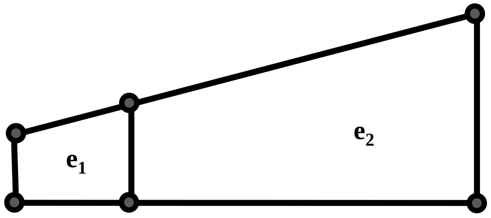
(b) Modified Model
(Prevents Rigid Body Motion)

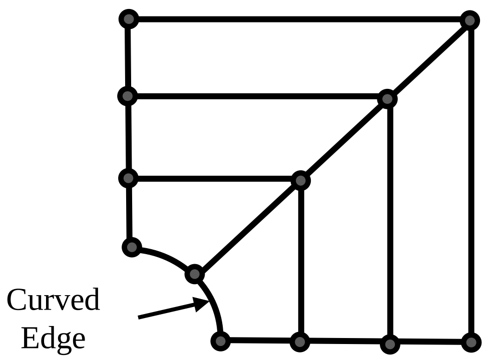
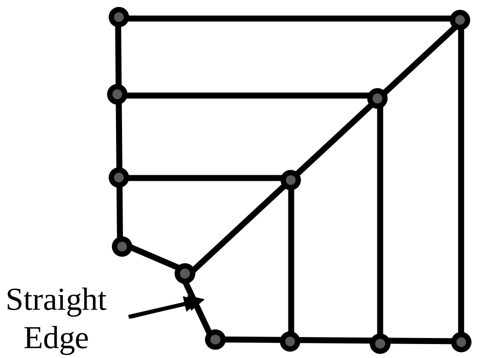
(c) Alternative Model
(Prevents Rigid Body Motion)

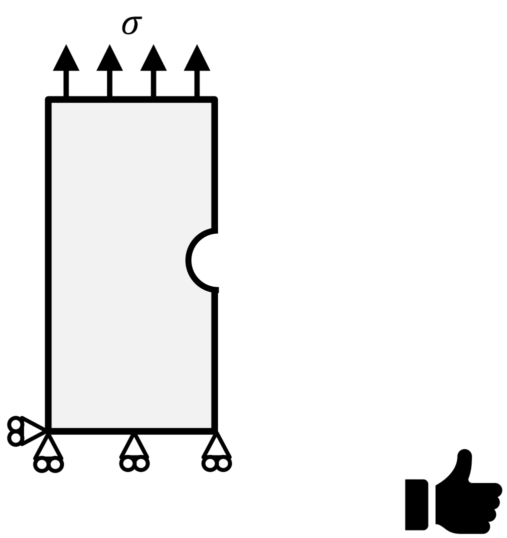
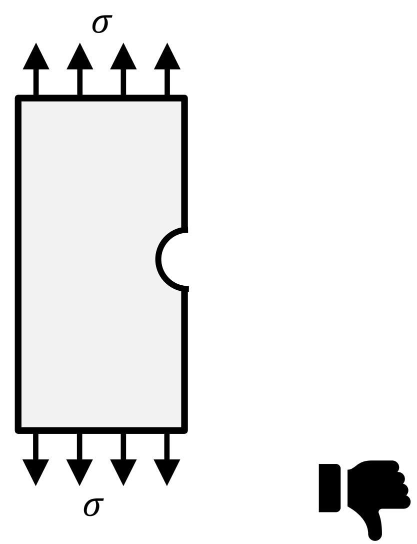
Figure 4: Preventing rigid body motion

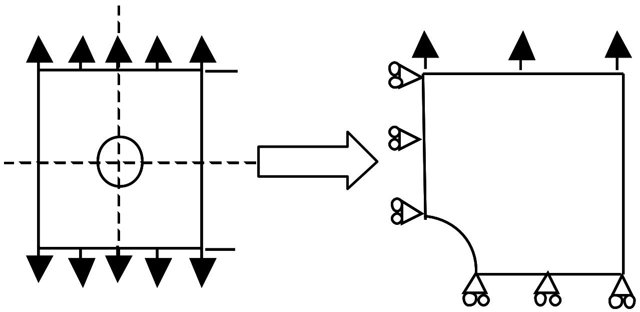
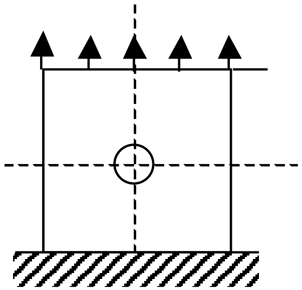
6 Examples of Good And Bad Practice

Good Practice	**Bad Practice**
Use undistorted element shapes: (Equilateral trianges, quads, tets, hex) 	Long/thin elements (high aspect ratio) - ‘Slivers’, 

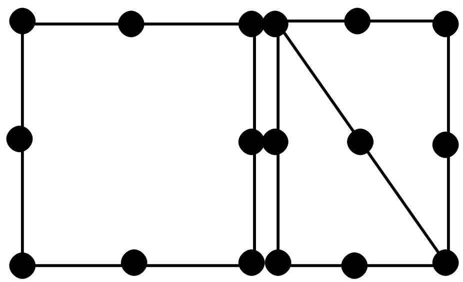
<table>
<tr><th>Good Practice</th><th>Bad Practice</th></tr>
<tr><td>Use a well-graded mesh (with a gradual, not abrupt, change in size of adjacent elements)

</td><td>Abrupt change in size of adjacent elements

</td></tr>
</table>

<table>
<tr><th>Good Practice</th><th>Bad Practice</th></tr>
<tr><td>Use quadratic (or higher-order) elements to fit a circular or curved boundary.</td><td>Straight-sided (linear) elements used to fit a circular boundary (high-curvature)</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

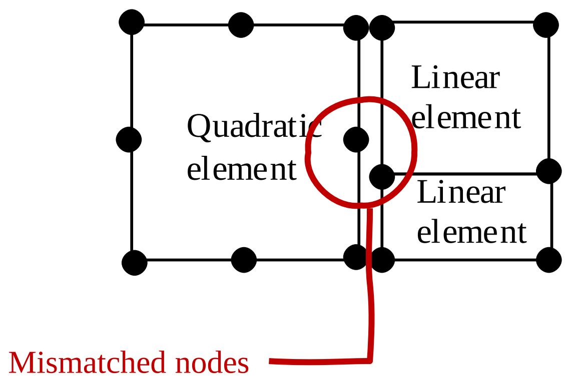
Good Practice	Bad Practice
Prevent rigid body motion	Rigid body motion not prevented
	

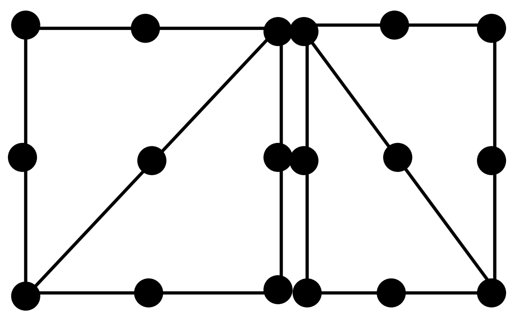
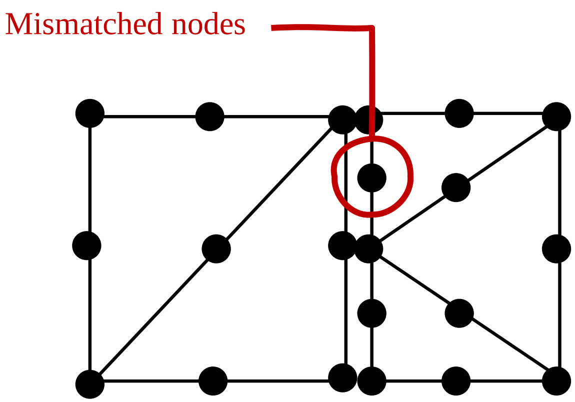
Good Practice	Bad Practice
Take advantage of symmetry to reduce problem size (e.g. quarter of the geometry is modelled) 	Symmetry not used 

Good Practice	Bad Practice
Use adjacent element sides with the same number of nodes and the same element order	Adjacent element sides have either different number of nodes or different element orders



Quadratic Elements



Good Practice	Bad Practice
Corner nodes connected to other corner nodes	Corner nodes connected to mid-side nodes
 Quadratic Elements	 Quadratic Elements

7 Summary

- FE solutions should always be carefully checked, and not taken for granted to be accurate.
- The accuracy of the FE solutions is strongly influenced by the degree of mesh refinement used. For best accuracy, the FE mesh should be refined in regions of rapidly changing stresses and should not contain long and thin elements.
- For every problem to be solved by FE analysis, the user must specify the geometry, material properties, analysis type, displacement boundary conditions and applied loads.
- Rigid body motion, where the structure may move without causing any strain, must be prevented by ensuring that there are sufficient displacement constraints in all Cartesian directions.
- Displacements and stresses are usually under-estimated in FE analysis, even if a very fine mesh is used. However, this underestimate is generally very small.
- FE results for stresses are generally less accurate than the displacement results.
- Symmetry should be used to reduce the size of the problem, with appropriate roller displacement constraints placed on the axes of symmetry.