University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

**Computer Modelling Techniques**

# FE-06

# PRACTICAL GUIDELINES FOR FE APPLICATIONS

## 6.1 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? What degree of mesh refinement is needed?
- 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?

# 6.2 Data Input for FE software

To model a given problem using FE software, the user must specify, without ambiguity, all of the data required to define a problem with a unique solution. These include the following:

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

It is essential to define all of the above data before attempting to use FE software. A typical "**FE data input sheet**" is shown in Figure 6.1. It is advisable to use a Data Input Sheet for each new FE run. This not only helps the analyst in record-keeping, but also helps to explain, in a concise manner, the data used in the FE software to other analysts.

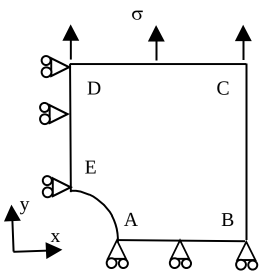
| | *Example* | *Explanation* |
|---|---|---|
| **REF. NO.** | Test001 (file name Test1.inp) | • Specify a unique reference number (and the file name containing the data input) |
| **DESCRIPTIVE TITLE** | Perforated plate analysis (symmetric quarter) | • Specify a unique descriptive title for the problem |
| **GEOMETRY** | <br>2D Plane strain<br>Continuum elements<br>Length of CD = 100 mm<br>Hole radius = 20 mm<br>Applied stress, $\sigma$ = 100 N/mm$^2$ | • Sketch the geometry<br>• Show all displacement constraints<br>• Show the applied loads<br>• Specify the dimensionality (i.e. 2D plane strain/stress, 3D, axisymmetric, etc.)<br>• Specify the configuration (e.g. continuum, beam , shell, plate)<br>• Specify units (even if not used in the analysis) |
| **MATERIAL PROPERTIES** | E = 250.0 x 10$^3$ N/mm$^2$<br>v = 0.25<br><br>(If plasticity is also considered , specify the yield stress and the uniaxial stress-strain curve) | • Specify all material properties relevant to the analysis<br>• Specify all units used (this is particularly important for non-linear problems)<br>• 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.<br>(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) |
| **ELEMENT TYPE** | 8-node 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 :<br>- Objective of the analysis<br>- Any special features, e.g. load applied in a number of load steps, initial conditions, etc. |

*Figure 6.1: A typical FE data input sheet*

# 6.3 Accuracy and Convergence of FE Solutions

## Sources of Error in FE Analysis

FE solutions are not exact solutions; they involve many approximations and assumptions. There are three main sources of error that may occur in FE solutions:

(a) Modelling Errors:
These errors occur if the geometry is not accurately modelled or the boundary conditions are not correctly 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:
These are usually due to round-off in the computations, where numbers are truncated due to insufficient digits being used in the calculations. In some problems, the solution matrix may become "ill-conditioned", i.e. very sensitive to small changes in the variables. This can occur when the stiffness matrix contains coefficients of varying orders of magnitudes in the same row, due to a large variation in element sizes or modulus of elasticity. Numerical errors can also occur in the numerical integration procedures.

## Convergence of FE Solutions

For a given problem, the FE solution should approach the exact solution, i.e. converge, as the mesh is refined. This should not be confused with convergence in non-linear problems where iterations are used.

The displacement-based FE formulation usually gives an over-estimate of the true stiffness of the element, i.e. elements are assumed 'over-stiff'. Therefore, 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 stresses are also usually under-estimated. In other words, the errors in the displacement values (and stresses) are always negative, as shown in Figure 6.2, which shows that as the mesh is refined, the computed displacement magnitudes slightly increase and approach the exact solution.

This assumes that a reasonably good mesh is used in the FE model. 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. This is an important design consideration when using FE solutions to arrive at stress values. It is interesting to note that, in contrast to FE, the Boundary Element (BE) method over-estimates the stress values.

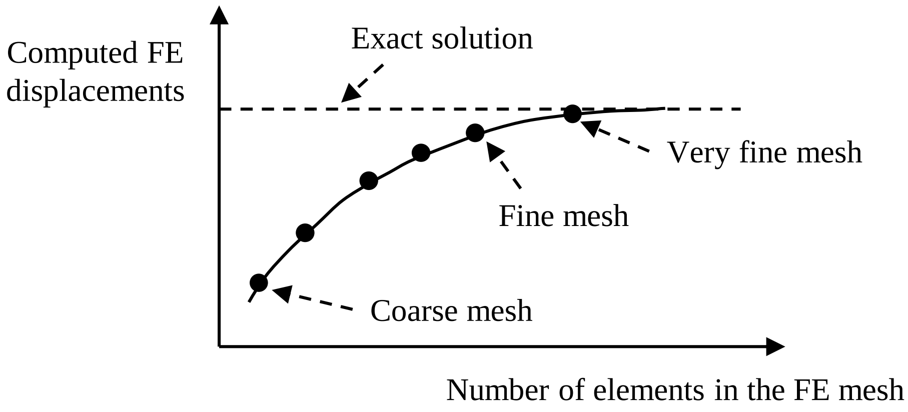



*Figure 6.2: Convergence of FE Solutions*

## 6.4 General Guidelines for Using FE Software

FE software should not be used as "black boxes" without a firm understanding of the underlying theory and principles behind the technique. The results should always be examined closely for discrepancies or checked against other solutions, whenever possible.

Some guidelines for improving the accuracy of FE solutions are given below:

- Choose the correct element type
  - The elements chosen for the problem must be of the correct geometrical type (configuration) with valid assumptions, e.g. 2D plane strain, 2D plane stress, axisymmetric, 3D, plate, beam, shell, etc.
  - Choose the most appropriate element order (e.g. linear, quadratic, etc.) suitable for the problem. In general, isoparametric quadratic elements are suitable for most problems. The elements must fit the surface or boundary of the problem.
  - If the geometry has awkward boundary shapes, higher-order elements, e.g. quadratic elements, should be used.

- Use a good mesh design
  - A "good" FE mesh may be defined as the mesh with the minimum number of elements required to arrive at an acceptable solution accuracy. Unnecessary use of very fine meshes is wasteful of computer resources, and may be seen as an indication of a poor understanding of the FE model.

- In regions of expected sharp stress gradients, e.g. at a notch or near a hole, the elements must be of small size, and gradually increase in size as the distance from this region increases. This is similar to curve fitting where more points are needed to fit a rapidly varying curve.

- Element sizes must not change abruptly from one element to another, i.e. element sizes must change gradually between adjacent elements. This is necessary to ensure that the degree of approximation is evenly spread across the FE mesh.

- <u>Avoid long thin elements</u>

  - The "aspect ratio" of the element, defined as the ratio of the largest to the smallest side, must be as close to 1 as possible, i.e. similar in shape to a square, an equilateral triangle or a cube.

  - Long thin elements always cause errors, but can be tolerated if there are only a few of them, and they are placed in regions of low stress gradients.

  - FE Programs usually perform a check on the aspect ratio of the elements to identify any 'bad' elements, i.e. elements with a high aspect ratio.

- <u>Perform a mesh convergence study</u>

  - It is often difficult to establish the optimum mesh refinement needed for an FE model. One way of checking that the FE mesh used is a reasonable one, is to start from a relatively coarse mesh, then refine it (e.g. by doubling the number of elements) to see the effect on the solutions. If the solutions exhibit large changes, this indicates that a further mesh refinement is necessary.

  - It is important to note that a mesh convergence study is not a 'fool-proof' approach, since a badly designed FE mesh will always give inaccurate solutions, regardless of the mesh refinement used.

- <u>Check stress accuracy</u>

  - Since displacements are the primary unknowns, displacement values are always more accurate than the stress values. If stress values are desired to a high degree of accuracy, a fine mesh should be used.

  - Stress values are always more accurate at the Gauss integration points (which are placed inside the element), rather than the averaged nodal stress values (which are usually obtained by extrapolation and interpolation from the Gauss points).

  - Discontinuities in stress values between adjacent elements (before nodal averaging) should be checked. These discontinuities can indicate the error in the approximations and should be as small as possible.

- <u>Avoid rigid body motion by restraining the displacement</u>

  - 'Rigid body motion' means the body may move as a whole without causing any strain. This violates the ‘small deformation’ assumptions used in the FE formulation. Therefore, rigid body motion must be prevented by specifying displacement constraints in all coordinate directions.

  - For example, in a 2D analysis, if there is a constraint in the x-direction but no constraint in the y-direction, the body will move as a “rigid body” in the y-direction.

  - In some problems, it may become necessary to add additional “artificial” displacement constraints to prevent rigid body motion. Such artificial constraints should be placed away from the region of interest or stress concentration areas, such that the overall deformations and stresses in the model are not adversely affected.

- <u>Check Reaction forces at the restraints</u>

  - Check that the summation of the externally applied loads is equal, or approximately equal, to the summation of the reaction forces at the constrained nodes.

- <u>Use a Benchmark with a known solution</u>

  - Before attempting a new analysis, use a "benchmark", i.e. a problem for which a reliable solution exists. A benchmark can be used to test the accuracy of the FE software, and to learn how to correctly use the FE software.

  - Benchmarks are particularly important in non-linear problems.

- <u>Ensure inter-element connectivity (if different types of elements are used)</u>

  - If elements types are mixed within the same FE mesh, e.g. triangular elements adjoin quadrilateral elements, then it is important to ensure that adjoining elements are correctly inter-connected such that there are no holes or gaps. This ensures that the compatibility relationships between adjacent elements are not violated.

  - The best solutions are obtained when all the elements in the FE mesh have the same number of nodes and the same order of variation (e.g. all linear or all quadratic). As a general rule, adjacent element sides should have the same number of nodes and the same order of variation.

# 6.5 Preventing Rigid Body Motion

## Why does rigid body motion occur?

Linear FE formulations are based on the assumption that element displacements and strains are small. 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.

Consider, as an example, a 2D problem involving a notched steel plate, under a stress of say 100 MPa, as shown in Figure 6.3(a). Since the forces are in equilibrium, the summation of all forces in the x and y-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 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

One way of preventing rigid body motion from occurring in this example is to fix the bottom surface, i.e. cement it to a rigid surface, as shown in Figure 6.3(b). This would make the FE solution valid since the small negligible "residual" force error, say $10^{-10}$ N, would only cause a negligible displacement in the constrained body. However, the resulting deformed shape of the FE model would be different from that of the real-life problem. Despite this, the stress concentration at the notch would be the same, or almost the same, provided that the notch is sufficiently far away from the bottom surface. Another possibility of imposing sufficient displacement constraints is shown in Figure 6.3(c).

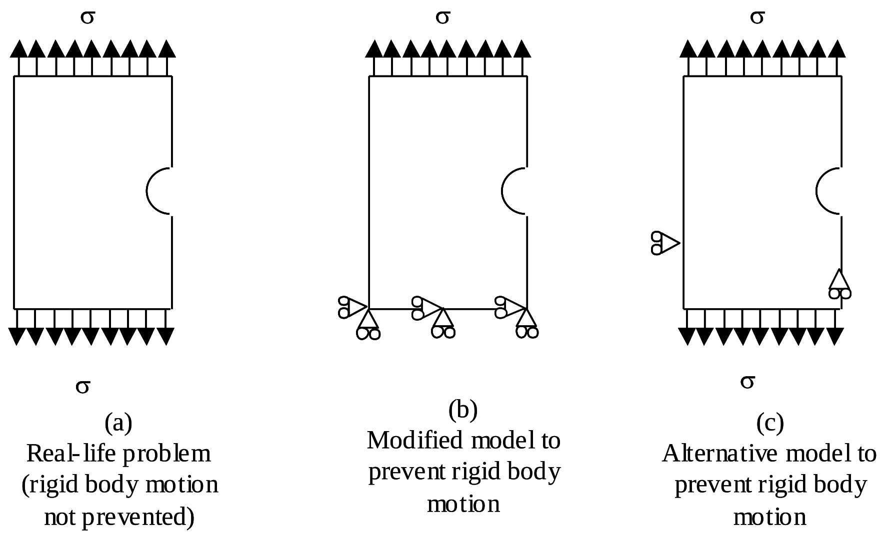
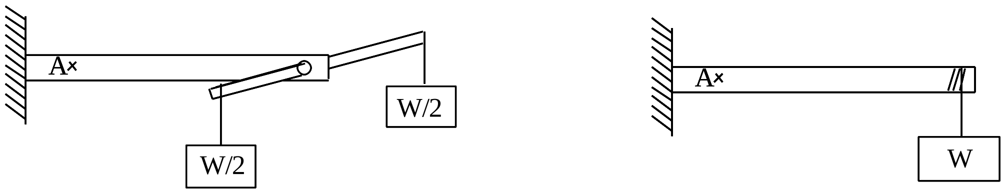



*Figure 6.3: Preventing rigid body motion*

## The St Venant Principle

In many FE problems, it becomes necessary for the user to introduce additional displacement constraints 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). This is effectively an application of the "*St. Venant Principle*" which 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.

As an example, consider the Cantilever problem shown in Figure 6.4 where two load cases are considered. In load case (a), two loads each of magnitude $W/2$ are applied through a rigid pin inserted in the beam. In load case (b), a hanging weight $W$ is attached to the free end of the beam. Since both load cases are statically equivalent, the stress and strains at point A, assuming it is far away from the load, are the same regardless of which load case is used. However, at or near the loaded end of the beam, the stresses and strains are very different in load cases (a) and (b).



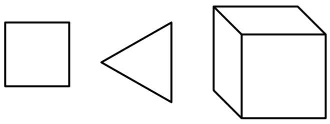
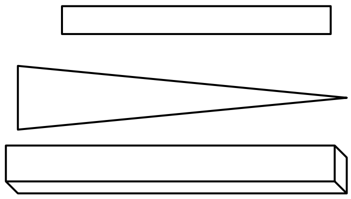
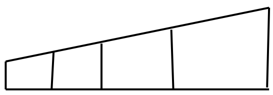
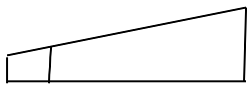
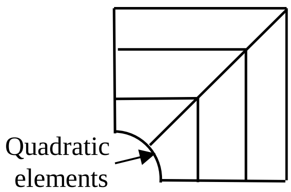
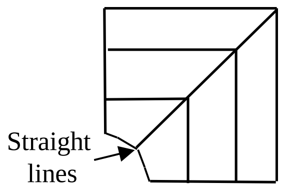

Load case (a) Load case (b)

*Figure 6.4: Example of the application of St. Venant's Principle*

Therefore, it is very important to prevent rigid body motion by imposing sufficient displacement constraints on the structure. The displacement constraints must be placed in all Cartesian directions. For example, in a 2D analysis, if there is a displacement constraint in the x-direction but no displacement constraint in the y-direction, the body will move as a "rigid body" in the y-direction. If it becomes necessary to add additional "artificial" displacement constraints to prevent rigid body motion, then they should be placed away from the region of interest so that the overall deformations and stresses in the model are not adversely affected.

# 6.6 Examples of Good And Bad Practice

Figure 6.5 shows a summary of some good and bad practices in applying the FE method.

| Good Practice | Bad Practice |
|---|---|
| Using good element shapes (close in shape to an equilateral triangular, a square or a cube)<br> | Using long thin elements (with large aspect ratios)<br> |
| Using a well-graded mesh (with a gradual, not abrupt, change in size of adjacent elements)<br> | Using a mesh with abrupt changes in the size of adjacent elements<br> |
| Using quadratic (or higher-order) elements to fit a circular or curved boundary.<br><br> | Using straight-sided (linear) elements to fit a circular or curved boundary<br><br> |

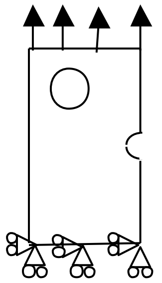
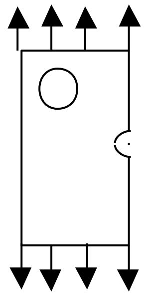
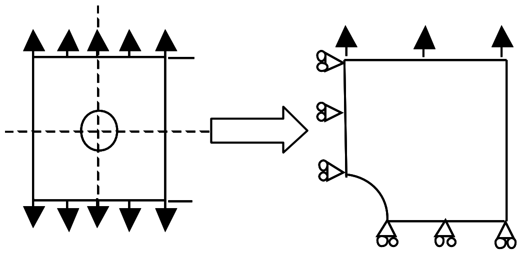
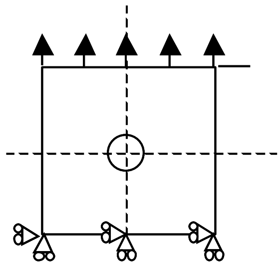
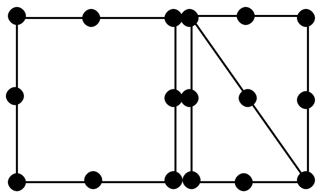
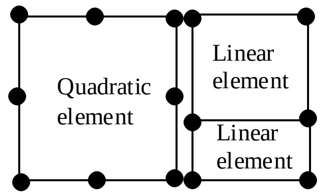
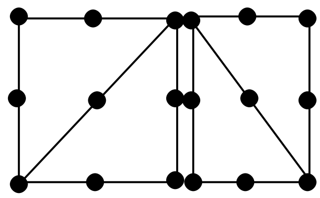
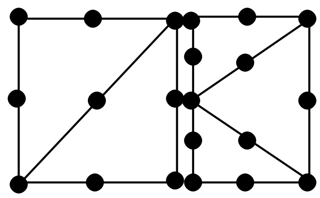
| Good Practice | Bad Practice |
|---|---|
| Preventing rigid body motion (in both x and y directions)  | Rigid body motion not prevented (Here additional displacement constraints are needed)  |
| Taking advantage of symmetry to reduce problem size (here a quarter of the geometry is modelled)  | Symmetry not used  |
| Using adjacent element sides with the same number of nodes and the same order of variation  Quadratic elements | Mixing linear and quadratic elements where adjacent element sides have different order of variation  |
| Corner nodes connected to other corner nodes  | Corner nodes connected to mid-side nodes  |

*Figure 6.5: Examples of good and bad practice*

# 6.7 Pre and Post Processing

## FE Software Capabilities

A large number of commercial FE software packages are available for engineering analysis. FE codes can be used on a wide range of computer hardware systems ranging from personal computers to large main-frame computers. The capabilities of the FE software vary widely, but most general-purpose FE commercial codes offer the following features.

- Analysis capabilities
  Stress analysis, heat conduction, dynamic behaviour, etc.

- Element library
  2D, 3D, axisymmetric, beam, plate, shell, quadratic and higher-order elements.

- Material behaviour models
  Elastic, plastic, creep, visco-elastic, anisotropic, etc.

- A range of boundary conditions and loads
  Point constraints, sliding, applied loads, pressures, heat flux, contact with friction, etc.

- Automatic mesh generation
  Graphical tools for generating 3D geometries.

- Pre and post-processing facilities
  Mesh plots, deformed shapes, stress contour plots, graphs of variables, etc.

Finite element analysis can be broken down into three distinct stages (see Figure 6.6):

Stage 1: The Pre-processor, in which the mesh is generated and the data input file is constructed

Stage 2: The Solver, in which the matrices are assembled and solved ("number crunching" stage)

Stage 3: The Post-processor, which is used to view the results of the solver.

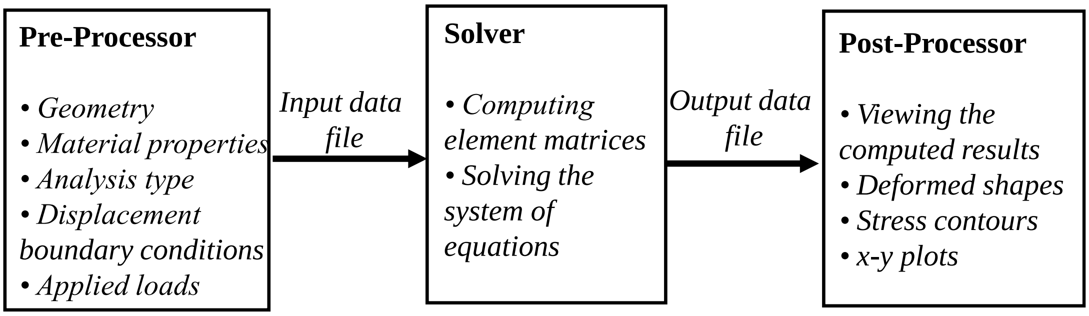


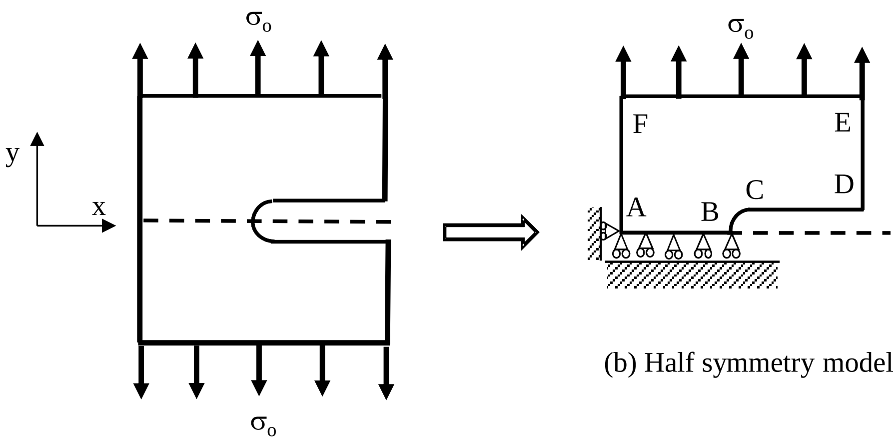

*Figure 6.6: The three stages of FE analysis*

## Stage 1 : The Pre-processor

The pre-processing stage produces an input data file which contains all the data needed to define the problem .The most time consuming part of generating an input file for an FE analysis is the mesh generation, i.e. the specification of the coordinates of each node and each element in the mesh. Commercial FE packages usually offer automatic or semi-automatic mesh generation, e.g. generating the mesh by simply specifying few points on the boundary and the number of elements. This also helps to avoid mistakes in the mesh description. FE meshes can also be generated from CAD drawings or 3D solid models.

A good pre-processor should be able to detect obvious errors in the data input, e.g. identifying long thin elements, incorrectly specified elements, missing information, etc. Pre-processors are usually interactive and use colour graphics to aid the visualisation of the mesh. They also exhibit good visibility features such as zooming, rotating, changing the angle of view, removing hidden lines, etc., which are particularly useful in generating three-dimensional meshes.

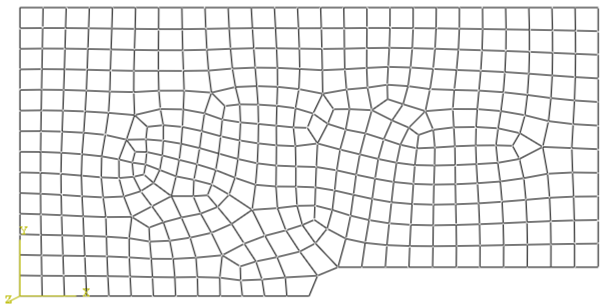
As an example, consider the notched plate shown in Figure 6.7. Due to symmetry, only half the plate is modelled with sliding (roller) conditions placed on the symmetry line (line AB). An additional displacement restraint is placed at point A to prevent rigid body motion in the x-direction. Examples of automatically generated meshes are shown in Figure 6.8.



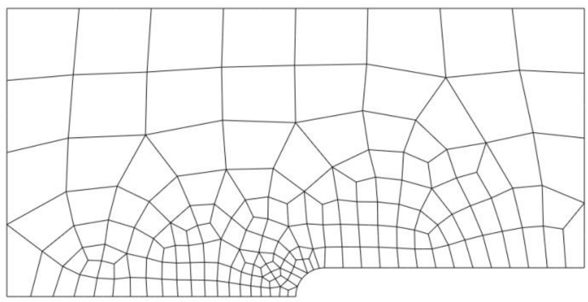

(a) Full geometry

(b) Half symmetry model

*Figure 6.7: Notched plate example*



(a) Free mesh without mesh refinement



(b) Free mesh with mesh refinement around the notch

*Figure 6.8: Examples of free meshing with and without mesh refinement at the notch area*

## Stage 2 : The Solver

The input file generated by the pre-processor is used to construct the element stiffness matrices of each element, and then assemble the overall system of simultaneous equations. The nodal displacement vectors are first calculated, and then used to obtain the strains, stresses and other variables.

The solver stage is often referred to as "number-crunching", because of the very large number of mathematical operations involved in constructing and solving the overall system of equations. After a successful run, the solver should produce an "output file", which contains all relevant information, e.g. displacements, stresses, strains, temperatures, etc. at all nodes and elements. The output file can be a "text" file which can be directly viewed by the user, or a "binary" file which is intended for the post-processor.

## Stage 3 : The Post-processor

Post-processing is used for examining the computed values, e.g. displacements or stresses at specific locations. Commercial FE software packages usually offer the user an interactive coloured graphical display of the results as well as printed listings of the variables. Features of post-processors include the following:

- Deformed shapes:
These are produced by multiplying the deformations of all the nodes by a suitable magnification factor and adding them to the nodal coordinates, as shown in Figure 6.9. This produces an "exaggerated" deformed shape which may be used as a qualitative observation of the general trend of the displacements, as well as checking that the displacement constraints have been correctly described. However, it is important to remember that exaggerated deformed shapes may be misleading.

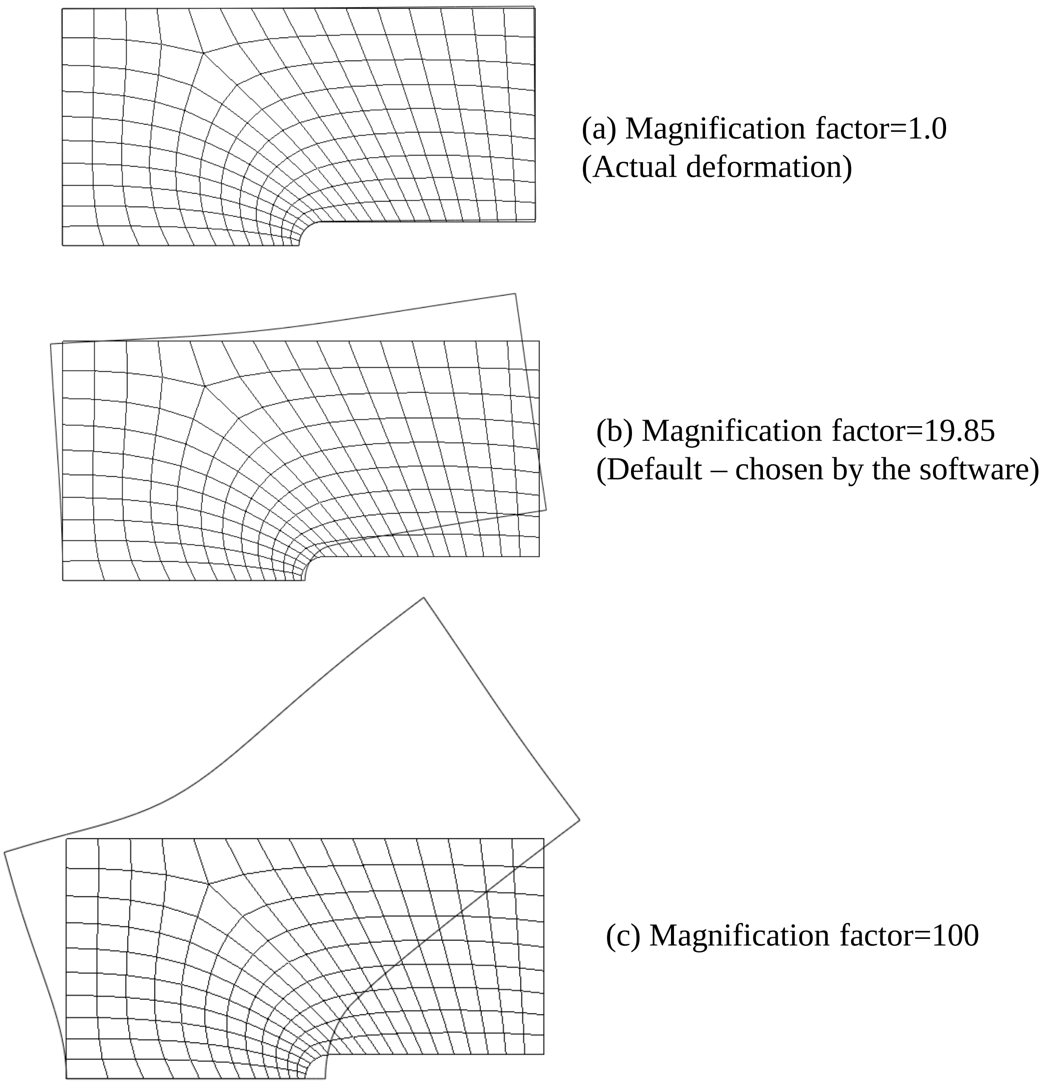



*Figure 7 9: Examples of deformed shapes with different magnification factors*

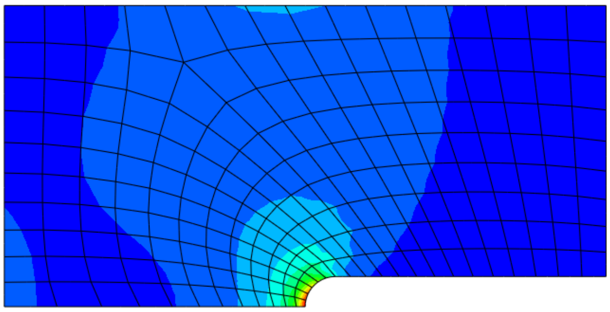
- Stress contours:
These are produced by extrapolation and interpolation of the stress values to generate "colour contours" of constant stress (similar to those used in geographical maps). They provide a useful visual interpretation of the results, e.g. highlighting stress concentration areas. Most FE software codes allow the user to manipulate the contours, e.g. to show only certain levels of stress values. Although FE solutions produce some degree of stress discontinuity between elements, stress contours are often "smoothed", i.e. presented as a gradual blend of colours, as shown in Figure 6.10.

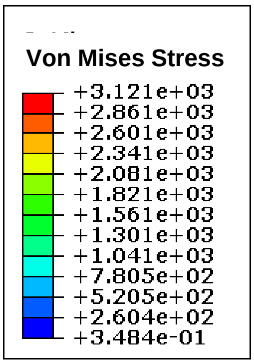
It is often convenient to plot colour contours showing the "Von Mises Equivalent stress" (also called the effective stress). This is based on the Von Mises yield criterion which stipulates that yielding occurs in any multi-axial stress situation when the value of the Von Mises equivalent stress reaches or exceeds the value of the material's yield stress. The Von Mises equivalent stress, $\sigma_e$, incorporates all stress components in one expression, as follows:

$$\sigma_e = \sqrt{\frac{1}{2}\left[\left(\sigma_{xx} - \sigma_{yy}\right)^2 + \left(\sigma_{yy} - \sigma_{zz}\right)^2 + \left(\sigma_{xx} - \sigma_{zz}\right)^2 + 6\left(\sigma_{xy}{}^2 + \sigma_{yz}{}^2 + \sigma_{xz}{}^2\right)\right]} \quad (6.1)$$

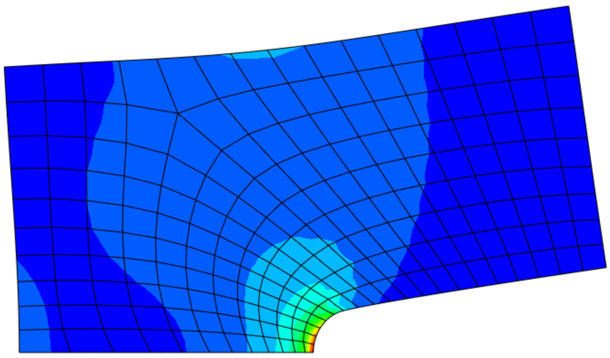
- X-Y charts:
  These are used to plot the variation of any variable along a given path, e.g. the variation of stress on a specific line or the variation of stress with time in time-dependent problems. These plots are very useful for inclusion in reports.

- Vectors plots:
  Vectors can be plotted to represent the magnitude and direction of some variables, e.g. principal stresses.






(a) Stress contours plotted on the undeformed shape



(b) Stress contours on plotted on the deformed shape

*Figure 6.10: Examples of stress contours for the notched plate problem*

# 6.8 Summary of Key Points

- 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.

- Quadratic elements are better at representing curved geometries and rapidly varying stresses.

- In beam, plate or shell elements, the boundary conditions must include the rotations (gradients of displacement) as well as the displacements.

- Pre-processing facilities allow the user to fully specify the data input requirements for a given problem.

- The solver stage, which follows the pre-processing stage, involves computing the element stiffness matrices and solving the equations. Solving the equations is the most computationally intensive stage in FE analysis.

- Post-processing facilities enable the user to examine the solution variables and check for any inconsistency in the computed results.

- Deformed shapes are useful in providing a qualitative assessment of the FE solutions, and may be used as a check that the displacement boundary conditions have been correctly applied.

- Coloured stress contour plots provide a useful visual check of the stress distribution in the solution domain, particularly around stress concentration areas.