



**Author:**  
*Nikolay Prieto Ph.D(c)*

# Computational Laboratory

## ANSYS (Part II

### Elements and Meshes)

## Outline

|          |                               |          |
|----------|-------------------------------|----------|
| <b>1</b> | <b>Introduction</b>           | <b>1</b> |
| <b>2</b> | <b>Elements</b>               | <b>1</b> |
| 2.1      | Element shapes . . . . .      | 2        |
| 2.2      | Number of nodes . . . . .     | 2        |
| 2.3      | Degrees of freedom . . . . .  | 3        |
| <b>3</b> | <b>Practical Exercises</b>    | <b>4</b> |
| 3.1      | Material properties . . . . . | 4        |
| 3.2      | Boundary Conditions . . . . . | 5        |
| <b>4</b> | <b>Report content</b>         | <b>5</b> |

## 1 Introduction

In the previous laboratory, we discussed some of the assumptions that can be used to simplify and reduce the dimensionality of a Finite Element Model (FEM). For example, if a beam has a constant cross section, it can be modeled as a one-dimensional body. If a beam is symmetric with a uniform cross section, it can be modeled as a two-dimensional body. Or, it can be modeled as three dimensional body. If created correctly, all three types of models will produce the same results. However, they will have different solid model geometry and must be meshed using different element types (Fig. 1 )

## 2 Elements

The Ansys library element is available to be used dependent on the purpose. There are specific elements for heat transfer, static structural, fluid dynamics or transient (time domain) problems. Each ANSYS element has a number of properties including its name, characteristic and degenerate shapes, number of nodes, degrees of freedom, real constants, key options, material properties, permitted loads, and special features. An overview of these features can be found in the input summary of each element in the ANSYS Element Library. The input summary follows the element description and the element input data.

Each element in ANSYS has a name followed by a number. The combination of a name and a number cannot exceed 8 characters. The name indicates the element's family (FLUID, PLANE, SHELL, SOLID, etc.). The number is a unique identifier called the element routine number. For example, a PLANE182 element

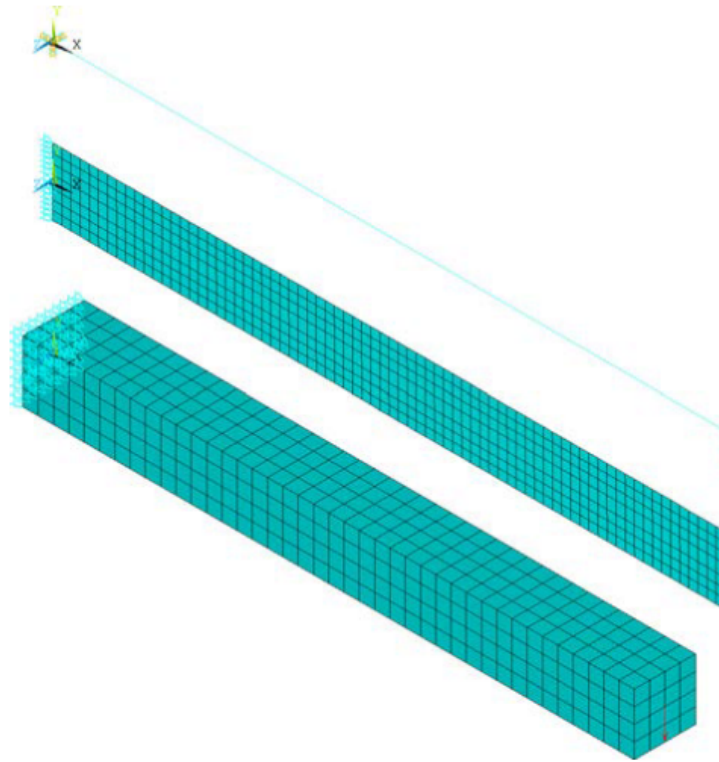


Figure 1: Element Plots of a Square Cantilever Beam Modeled using 1D Beam Elements (BEAM189, top), 2D Continuum Elements (PLANE183, center), and 3D Continuum Elements (SOLID185, bottom).

is a member of the PLANE element family and it is element routine number 182. When element types are defined using the command line or input files, only the routine number is required. The element family name is optional.

## 2.1 Element shapes

There are eight possible element shapes in ANSYS: points (for point elements); lines (for line elements); triangles or quadrilaterals (for area elements); and tetrahedrons, pyramids, prisms, or bricks (for volume elements). These shapes are shown in Fig. 2

## 2.2 Number of nodes

All elements have nodes that define their location in space. For example, quadrilateral elements like PLANE55 have four nodes (I, J, K, and L)—one for each corner (Fig. 3, left). For example, PLANE77 is the counterpart to PLANE55 with midside nodes. It has a total of eight nodes (I, J, K, L, M, N, O and P) - one for each corner and one in the middle of each side (Figure 3, right). The node letters refer to the position of the nodes on a generic element (i.e., on the element type). The node lettering convention starts with the letter I and increments one letter per node.



Figure 2: Possible Element Shapes in ANSYS.

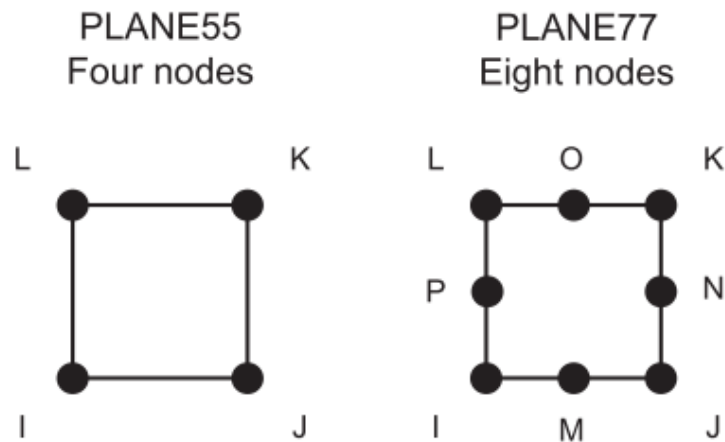


Figure 3: Quadrilateral Elements Without and With Mid-Side Nodes.

Quadrilateral and brick elements can be forced into degenerate shapes with one or more triangular faces in order to mesh geometries that could not be meshed otherwise. For example, a quadrilateral element may collapse into a triangle (Figure 4.7, 4), while a brick element may collapse into a prism, a pyramid, or a tetrahedron. This is achieved by defining the same node number for multiple nodes (Figure 4, right).

## 2.3 Degrees of freedom

The degrees of freedom (DOFs) are the primary unknowns in the equations that constitute a FEM. Solving the equations determines the values of the DOFs for each node in the model. These values are referred to as the "primary data" in the documentation. The derived data (stresses, strains, gradients, fluxes, etc.) are calculated from the DOF solution. ANSYS DOFs include displacements (UX, UY, UZ), rotations (ROTX, ROTY, ROTZ), temperature (TEMP), fluid pressure (PRES), fluid kinetic energy (ENKE), magnetic vector potential (AX, AY, AZ), voltage (VOLT), and current (CURR). As an example Figs. 5 and 6 show the DOF of some ANSYS elements for structural and thermal analysis, respectively.

The degrees of freedom included in each element type reflect the physics of the underlying problem. You should choose elements that offer only the degrees of freedom that you require since additional degrees of freedom increase computation time and provide no benefit. For example, structural analysis does not (usually) involve heat transfer or voltages. Therefore, the element(s) used to model structures should not have TEMP or VOLT DOFs.

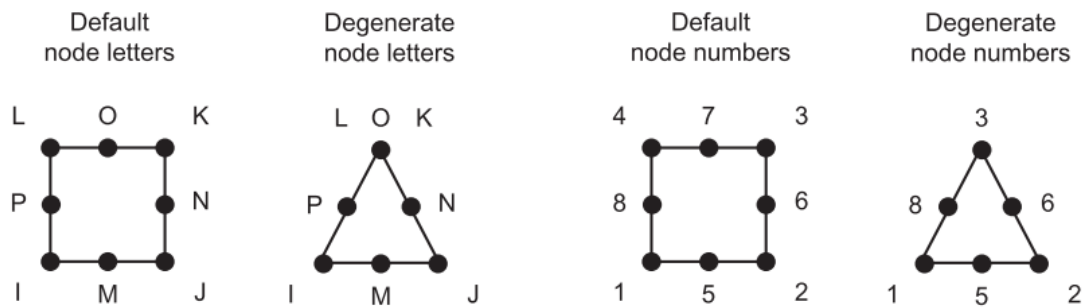


Figure 4: Quadrilateral Elements with Default and Degenerate Shapes: Node Letters for a Generic Element (left) and Node Numbers for a Specific Element in a FE Mesh (right)

| Element  | Order | Shape | Nodes | Physics    | DOFs       |
|----------|-------|-------|-------|------------|------------|
| PLANE182 | 2D    | Quad  | 4     | Structural | UX, UY     |
| PLANE183 | 2D    | Quad  | 8     | Structural | UX, UY     |
| SOLID185 | 3D    | Brick | 8     | Structural | UX, UY, UZ |
| SOLID186 | 3D    | Brick | 20    | Structural | UX, UY, UZ |
| SOLID187 | 3D    | Tet   | 10    | Structural | UX, UY, UZ |

Figure 5: Commonly Used Structural Continuum Elements

### 3 Practical Exercises

In this laboratory, you will perform a steady-state structural analysis of a cantilever beam using 1D, 2D, and 3D models meshed with beam, plane, and solid elements. The beam is made of aluminum with a Young's modulus of 73.1GPa and a Poisson's ratio of 0.33. It is 1.0m in length with a  $100 \times 100$ mm cross section. It has a load of 5000N applied to the unsupported end (Figure 7).

The 1D model will be meshed with BEAM189 elements. The 2D model will be meshed with PLANE182 elements. And, the 3D model will be meshed with SOLID186 elements. The goal of each analysis is to determine the deflection at the end of the beam and the stresses throughout the beam. Once all three models have been solved, the differences and similarities among the results of the three models will be examined.

#### 3.1 Material properties

- Young's Modulus  $7.310 \times 10^{10}$ Pa
- Poisson's ratio 0.33



| Element | Order | Shape | Nodes | Physics | DOFs |
|---------|-------|-------|-------|---------|------|
| PLANE55 | 2D    | Quad  | 4     | Thermal | TEMP |
| PLANE77 | 2D    | Quad  | 8     | Thermal | TEMP |
| SOLID70 | 3D    | Brick | 8     | Thermal | TEMP |
| SOLID90 | 3D    | Brick | 20    | Thermal | TEMP |
| SOLID87 | 3D    | Tet   | 10    | Thermal | TEMP |

Figure 6: Commonly Used Thermal Continuum Elements

### 3.2 Boundary Conditions

- 6000 N downward load applied to the center of the free end of the beam
- The fixed end of the beam is fully constrained in x, y, and z.

## 4 Report content

- The report should be formatted as a IEEE template style.
- The report may content the following sections: Abstract, Introduction, Methods, Results, Discussion and Conclusions.
- Do not forget to perform the analysis in 1D, 2D and 3D.
- Apply the element types mentioned.
- Implement a convergence analysis for each dimension, you can plot it as shown in Fig. 8. For each dimension problem implement the following ranges in quantity of elements:  $elem = [100, 1000, 5000, 10000, 50000]$
- Perform the analytical method (pure bending)
- Make a table with the main results on each and compare with the analytical answer, as shown in Fig. 9
- Do not forget to establish strong conclusions.

This laboratory is inspired from the book of [Thompson and Thompson \(2017\)](#). Enjoy this lab, regards!

## References

Thompson, M. K. and Thompson, J. M. (2017). *ANSYS Mechanical APDL for Finite Element Analysis*.

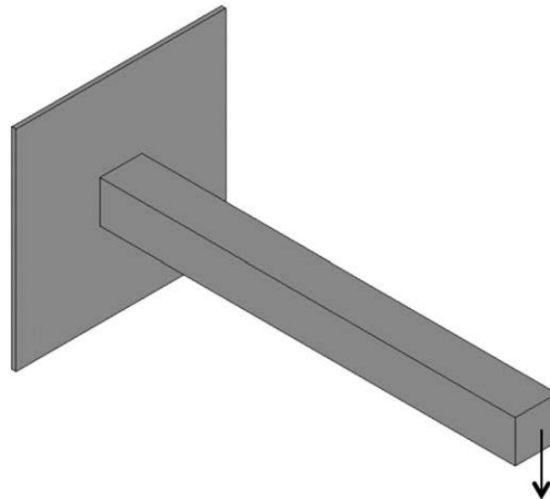


Figure 7: Schematic of Cantilever Beam with End Load.

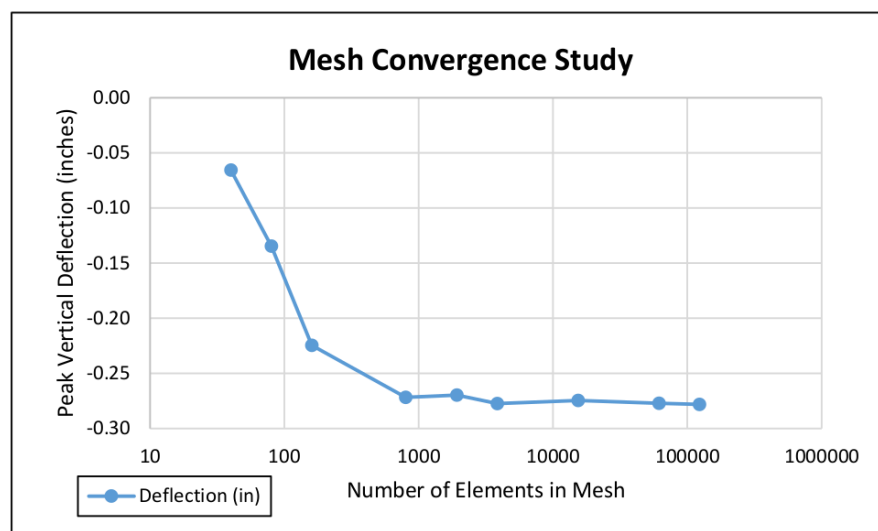


Figure 8: Example of a mesh convergence plot.



|          | Maximum Stress | Maximum Displacement |
|----------|----------------|----------------------|
| Theory   | 3.00e7 Pa      | 2.736E-3             |
| Beam189  | 3.00e7 Pa      | 2.758E-3             |
| Plane182 | 3.03e7 Pa      | 2.739E-3             |
| Solid186 | 3.01e7 Pa      | 2.766E-3             |

Figure 9: Example of general results plot.