MAE 656 - Advanced Computer Aided Design

04. 2D and 3D Solids – Doc 01

Introduction

Introduction

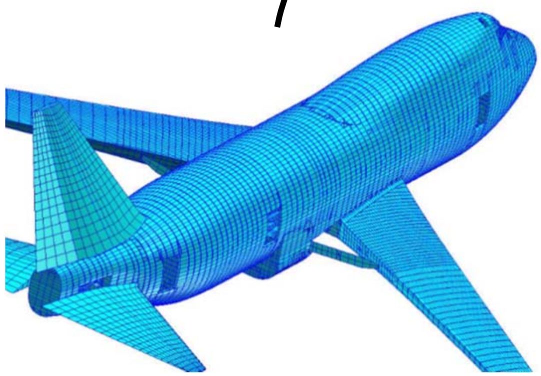
A FEM simulation of a solid element seeks to obtain the mechanical performance of the structural component.

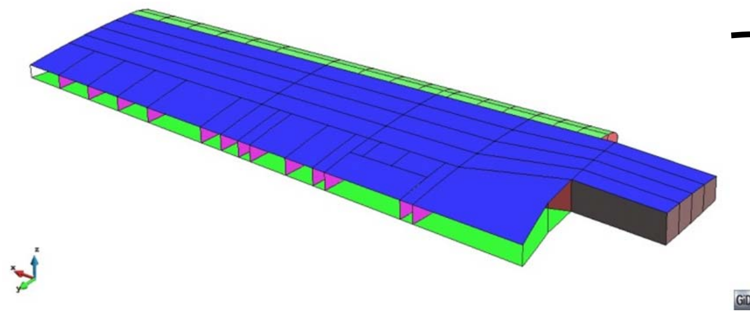
The accuracy of the solution provided by a FEM simulation depends on several parameters, such as mesh, material model

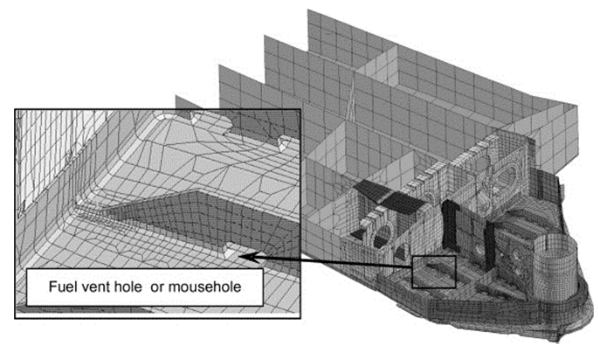
However, the main responsible of the solution accuracy is the simplifications (detail level) that we may decide to apply to our numerical model.

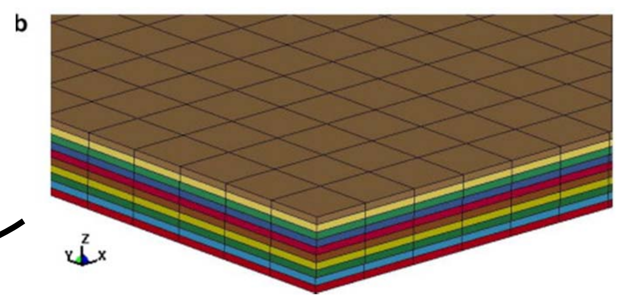
Introduction

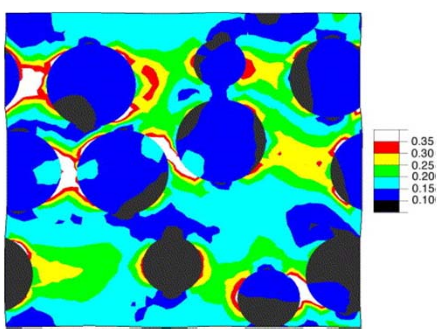
Example:











Introduction

We will learn:

- How mesh affects accuracy
- Simplifications that can be made to the model to reduce its computational cost:
  - 2D vs. 3D
  - Symmetries
- (and, of course, how to simulate solid structures)

Introduction

We cannot learn, in a FEM class, what model simplifications we have to assume/apply.

As this depends on many engineering factors such as,

- In what part of the structure we are interested:

  i.e.: if we want to know the global structural performance of an airplane, there is no need to simulate the laminate layup.

- Motivation of our simulation:

  i.e.: are we designing or we want to understand a failure mechanism? In the first case an elastic simulation is enough, in the second one not.

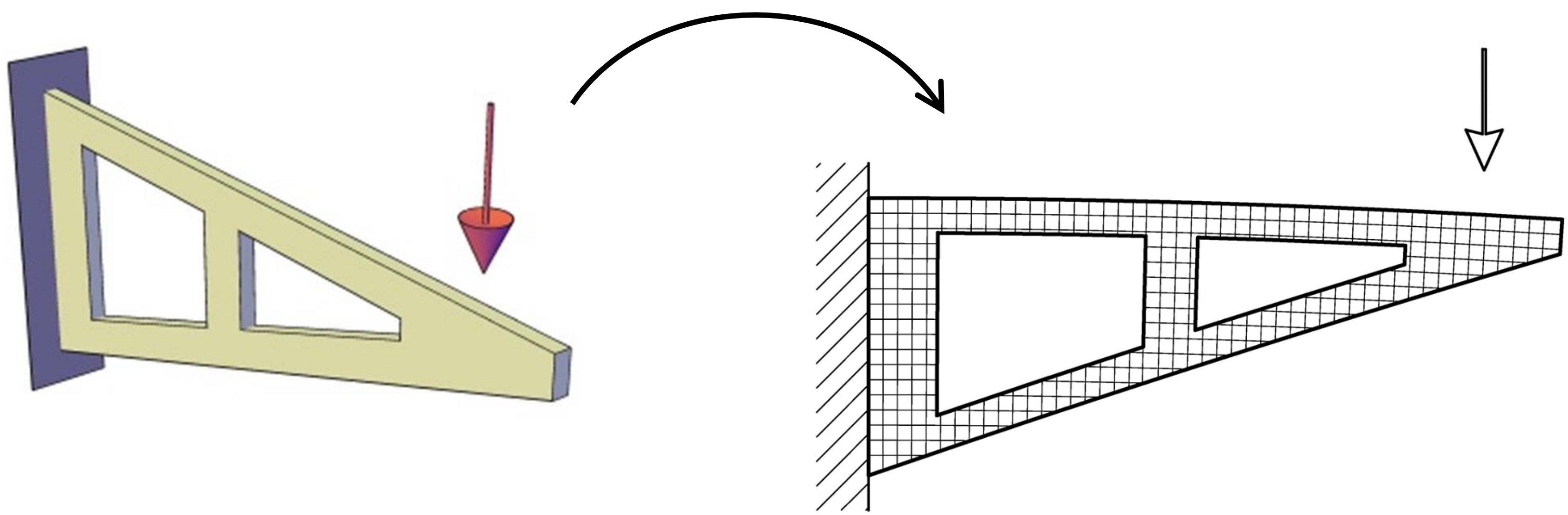
- Computational capabilities

  Nowadays it is "impossible" to solve a plane wing including in the model the detail of each single carbon thread.

- etc.

Formulation

The Finite Element Method applied to structural simulations of solid structures defines the correlation between strains and stresses in a Finite Number of Points of the structure, which is attached to the medium with several boundary conditions:



Formulation

The equation used to solve the structural problem is the

PRINCIPLE OF VIRTUAL WORK:

$$\int_V \delta\varepsilon^T \cdot \sigma \cdot dV = \int_V \delta u^T \cdot b \cdot dV + \int_A \delta u^T \cdot q \cdot dA + \sum \delta u^T{}_i \cdot P_i$$

Which applied to the discrete medium becomes, for each finite element:

$$\int_{V_e} B^T \cdot D \cdot B \cdot t \cdot dA \cdot a = \left(\int_{V_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right) + P_i$$

Formulation

Resulting in the solution of a linear system of equations:

$$K \cdot a = P_{ext}$$

With a the displacement of each node in which the structure is discretized, and P_{ext} the load applied to each node.

Once known the displacement of each node, it is possible to calculate the strain in each element:

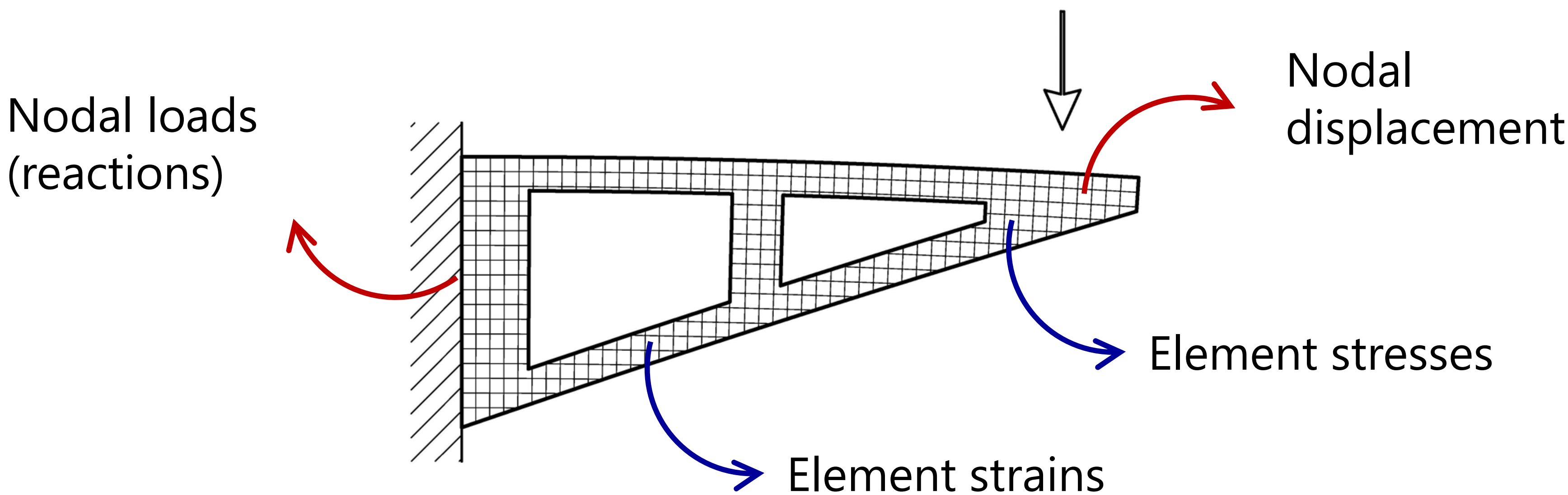
$$\varepsilon = B \cdot a$$

Formulation

And, form the strain values, we can obtain the stresses in each element as:

$$\sigma = D \cdot \varepsilon$$

Therefore, with a solid simulation we can obtain:



Problem definition

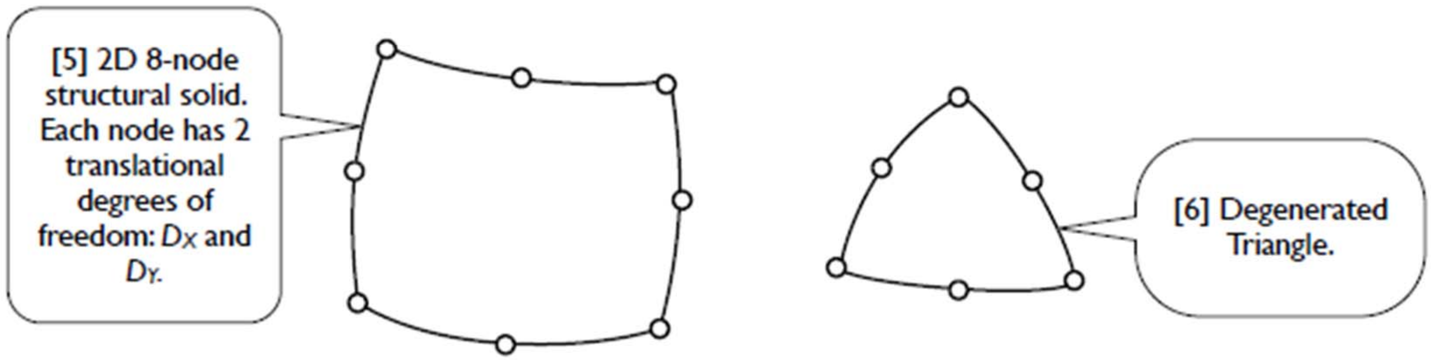
To do so we must define:

- Geometry
- Materials
- Boundary conditions
  - Loads
  - Supports
- Discretization (mesh)

Elements in Ansys Workbench

2D Solid Bodies

Element PLANE 183



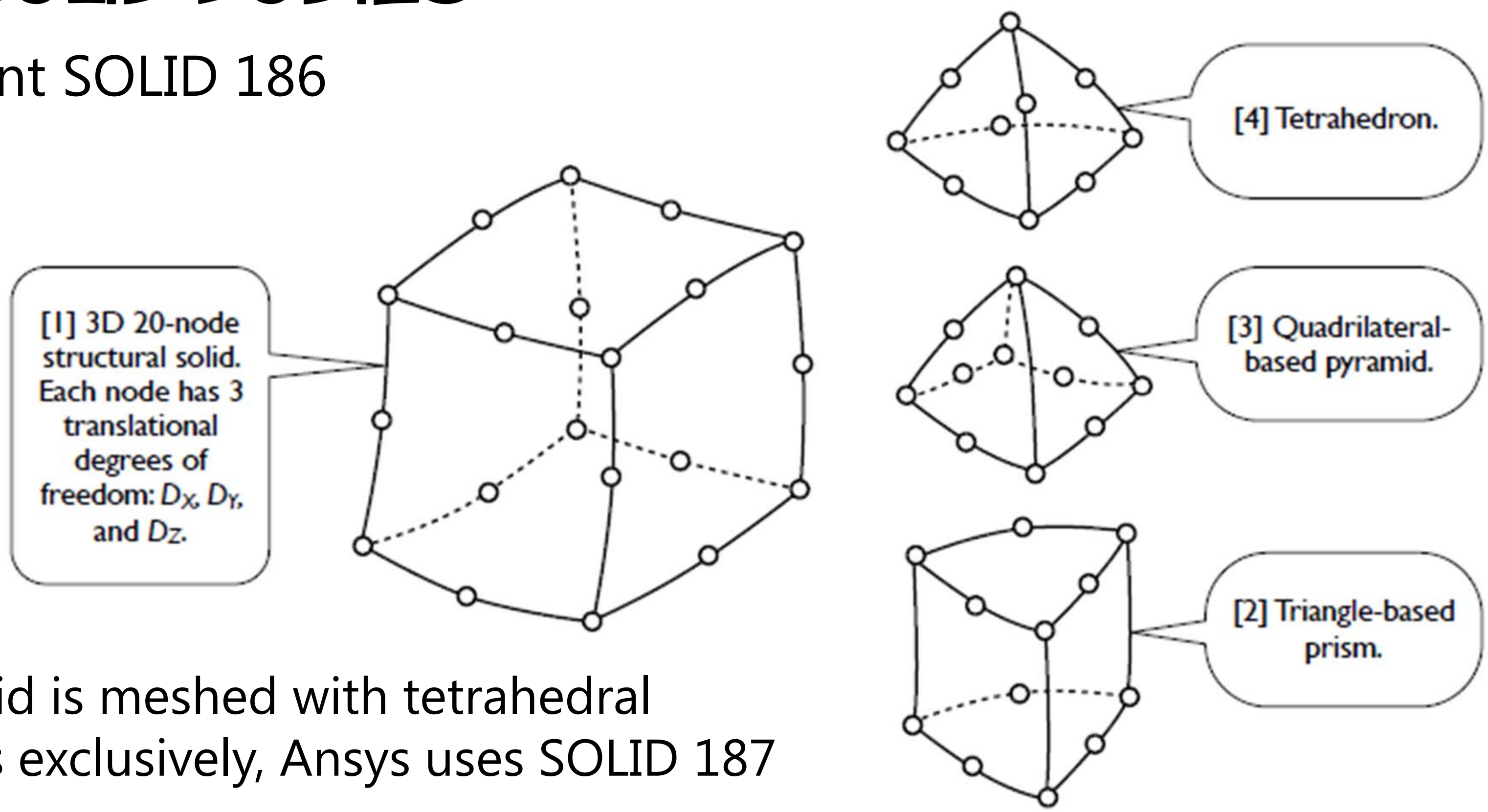
It is possible to drop the midside nodes. In this case, the element used is PLANE 182

The quadrilateral element has 2x2 integration points and the triangular element has 3 integration points.

Elements in Ansys Workbench

3D Solid Bodies

Element SOLID 186



If the solid is meshed with tetrahedral elements exclusively, Ansys uses SOLID 187

The brick element has 2x2x2 or 14 gauss points. Tetrahedron has 4 gp., the pyramid has 2x2x2 gp.