

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

### 4TME701U: Practical test of finite element

Y. Ledoux, duration : 1h20, année 2020-2021

1 personal A4 page is allowed

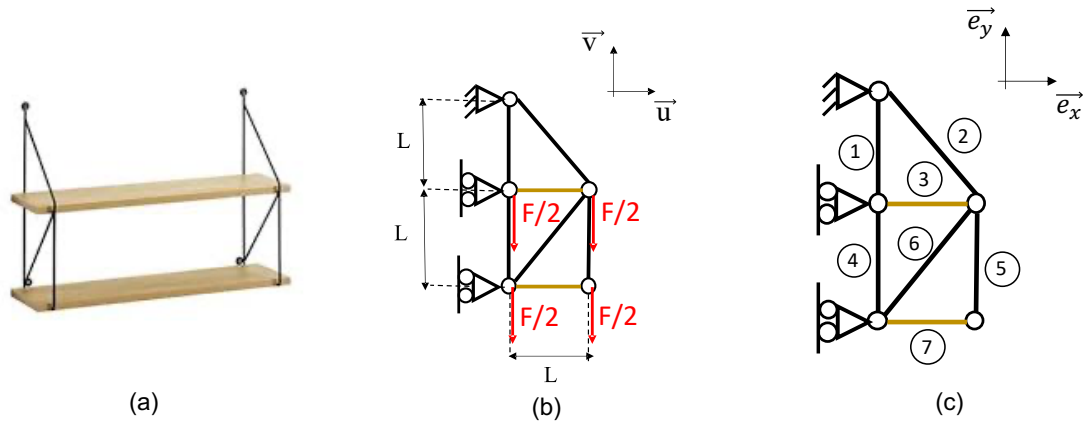


Figure 1 : Studied system (a) ; (b) equivalent modeling ; (c) numbering of the different frame. The load  $F/2$  is applied into the mechanical joint between the wood table and the metallic elements. The mechanical joints are modeled by pivot joints.

The study system is a wall-mounted shelf shown in Figure 1, consisting of a tubular metal structure (parts 1, 2, 4, 5, 6) and two wooden shelves (parts 3 and 7). The dimensions and characteristics are given in Table 1. The aim is to ensure that:

1. the stresses in the metal parts do not exceed the elastic limit of the material (100 MPa for the selected aluminum allows)
2. the shelves are as horizontal as possible.

Material	Young modulus [GPa]	Poisson's ratio	Shape of the cross section	Dimensions
Aluminum allows	70	0,3	Circular	R=5mm
Wood	12	0,33	Rectangular	15mm x 100mm

With  $F=200\text{N}$ ,  $L=100\text{mm}$ .

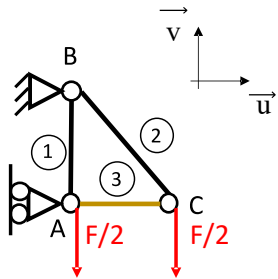
We give the stiffness matrix of the inclined truss:

$$[K_e] = \frac{ES}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \quad \text{such as} \quad \begin{cases} c = \cos \theta \\ s = \sin \theta \end{cases}$$

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

### 1.1 Theoretical computation (9 pt)

In this section, only a part of the system is examined composed of the parts 1, 2 and 3.



**Q 1.** Give the stiffness matrix of the part AC. Use  $k_{AC}$  parameter as the elementary component of stiffness. Give the value of  $k_{AC}$ . (1.5pts)

Figure 2 : Modèle pour la partie théorique

**Q 2.** Give the stiffness matrix of the part AC. Use  $k_{AB}$  parameter as the elementary component of stiffness. Give the value of  $k_{AB}$ . (1.5pts)

**Q 3.** Give the stiffness matrix of the part BC. Use  $k_{BC}$  parameter as the elementary component of stiffness. Give the value of  $k_{BC}$ . (1.5pts).

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

**Q 4.** Make the assembly of all elementary matrix. (3pts)

**Q 5.** Express the limit conditions to use and solve the system. Compute the displacements of the nodes A and C. (1.5pt).

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

## 1.2 Set up of the first model in Abaqus (5pt)

**Q 6.** Specify what type of elements you can use to model the problem in Figure 2. Explain the main differences according to the choice of elements (mechanical link, number of elements to use...) (1,5 pt)

**Q 7.** According you, what is the simplest model to implement? Justify in few words (1pts).

**Q 8.** Build the model in ABAQUS. Record the displacement values in A and C. Do you think that these results are satisfactory with respect to the function of this system? (eg. i. the stresses in the metal parts under 100 MPa; ii. the shelves are as horizontal as possible). (1,5pts).

*Valid the model by the teacher when, in your opinion, is correct.*

Record displacement value:  $u_A =$   $v_A =$

Record displacement value:  $u_C =$   $v_C =$

**Q 9.** In your opinion, is it necessary to use several elements for the different parts. Argue your answer in a few lines. (1 pt).

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

## 2 Modeling of the entire system

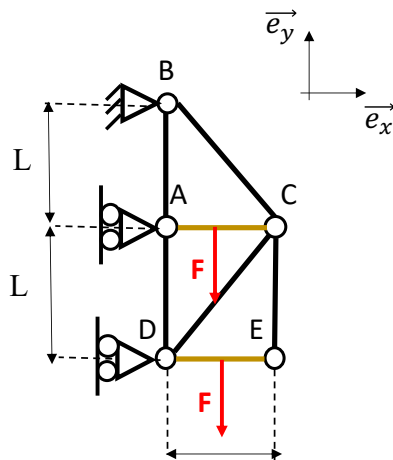


Figure 3 : System to study

**Q 10.** Set up the model in Abaqus. Read the displacement values in A, C, D and E.

Do you think these results are satisfactory with respect to the functional requirements of this system? (eg. i. the stresses in the metal parts under 100 MPa; ii. the shelves are as horizontal as possible) (2,5pts).

*Valid the model by the teacher when, in your opinion, is correct.*

Record displacement values:

$u_A =$   $v_A =$

$u_C =$   $v_C =$

$u_D =$   $v_D =$

$u_E =$   $v_E =$

**Q 11.** Record the maximum stress state in the metal parts and identify the area of maximum stress. What are your conclusions regarding the mechanical strength of the system? (1pt)

Recorded value:      Max stress value =      part n°

Prénom :	Nom :	Groupe	Date :
----------	-------	--------	--------

**Q 12.** The designer of these shelves wants to make a welded version at points A, C, D and E. Propose a modeling strategy in Abaqus of this new system (type of elements, number of elements, connections ...). Justify your choices. (1,5pt)

**Q 13. (bonus)** Make the model in Abaqus. And record the displacements.

*Valid the model by the teacher when, in your opinion, is correct.*

Record displacement values:

$u_A =$	$u_A$
$u_C =$	$v_C =$
$u_D =$	$v_D =$
$u_E =$	$v_E =$