Computational Engineering Structures

Marc Barcelo (s316955)
MSc Aerospace Computational Engineering

December 31, 2020

Finite Element Method Treatment - Comments prior to the solution

The exercises described here will follow the Finite Element Method approach to solve the continuous mechanics problem numerically through the computational discretized domain. This "translation" is not evident as several assumptions must be made and the stability of the solution can be compromised. As it can be seen, the dimension parameters ($\Delta x, \Delta t, \Delta y$, etc) must satisfy certain considerations (CFL, Fourier Series) to ensure the stability of the solution. The Finite Element Method works by translating the physical equations into the individual-cell of the variables in the particular grid (elements), assembling the "solution setups" of all those individuals to obtain the unified solution system for the whole domain. Therefore, it turns the physical problem into a matrix set of linear equations which can be solved through iterative means (see Gauss-Seidel, Jacobi, Thomas Algorithm, etc). Hence, its evident application and high performance in several ambits such as thermodynamics (specially the Fourier's conduction problem), electromagnetism and specially structural, where huge accuracy of the real problem is indeed obtained.
The FEM methodology for structures can rely on two direct approaches: the forces method or, the most widely used, the direct stiffness method. In these exercises, this last methodology will be the one applied, in which the displacements are considered to be the unknown and therefore the first variables to be solved. The relationship between forces and displacement will be accordingly done by using the Hooke's Law, as we will assume that the displacements are small and consequently we will be always in the elastic (more importantly, linear) regime. However, plasticity will be simulated in the impact problem of exercise 4.

Exercise 1

The beam shown in Figure 1 is clamped at the two ends and acted upon by the force P and moment M in the mid-span.

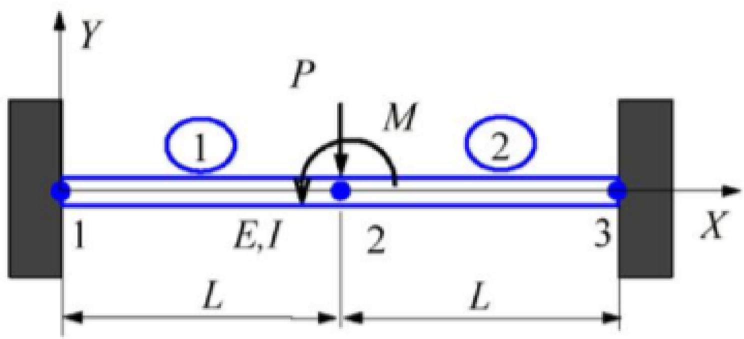


Figure 1: Fixed hinged beam subjected to a force and a moment

a) Find the deflection and rotation at the centre node (6 Marks)

Figure 1 presents 2 elements and 3 nodes in which nodes 1 and 3 are fixed and node 2 is set free. However, in this node, load force P and moment M are applied in the initial condition and hence the structure will react to it. As it is considered to be a beam, the deflections and rotations of the structure at this node will be the first variables to compute.

First, we construct the connectivity table in Table 1 in order to facilitate the formulation of local matrices, global matrices for each element and then the assembly of them into the global stiffness matrix.

Table 1: Connectivity Table for Exercise 1

Element	Length	Nodes	$\cos(\theta)$	$\sin(\theta)$
1	L	1,2	1	0
2	L	2,3	1	0

As it can be seen, both elements are distributed in the same orientation (the local axis are equal) and at the same time are in the same reference global axis. Therefore, no transformation is needed to be applied. According to the information provided in the slides by Dr Iman Dayyani and by the reference book [1], for the Beam Element, the local direct stifness matrix is given by

$$k^{Beam} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

so that $\{F\} = [k] \cdot \{d\}$ and hence

$$\begin{Bmatrix} F_{yi} \\ M_i \\ F_{yj} \\ M_j \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{Bmatrix} \nu_i \\ \theta_i \\ \nu_j \\ \theta_j \end{Bmatrix}$$

being E the Young modulus of the material, L its length, I its inertia moment, and consequently ν the deflection and θ the rotation.
The calculation of the stifness matrix of the two elements -beams- 1 and 2 which are geometrically and materially equal is here provided.

$$k_{[1]}^{Beam} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

$$k_{[2]}^{Beam} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

and the assembly of the global coordinates matrices into the global stiffness 6x6 matrix for the structure is computed.

$$k_{[global]}^{Beam} = k_{[1]}^{Beam} + k_{[2]}^{Beam} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 \\ & & 24 & 0 & -12 & 6L \\ & & & 8L^2 & -6L & 2L \\ & & & & 12 & -6L \\ Sym. & & & & & 4L^2 \end{bmatrix}$$

If the Hooke's Law for the elastic regime ($\{F\} = [k] \cdot \{d\}$) is here applied for the total structure, all the unknown variables will be presented in one system of algebraic equations such as it is shown below.

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \\ F_{3y} \\ M_3 \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 \\ & & 24 & 0 & -12 & 6L \\ & & & 8L^2 & -6L & 2L \\ & & & & 12 & -6L \\ Sym. & & & & & 4L^2 \end{bmatrix} \begin{Bmatrix} \nu_1 \\ \theta_1 \\ \nu_2 \\ \theta_2 \\ \nu_3 \\ \theta_3 \end{Bmatrix}$$

Now, in order to solve the particular problem we are facing, we must impose the boundary conditions or constraints given by the geometry and applied loads. If not, as it can be seen, apart from solving a general case, the global stiffness matrix is singular and the problem could not be solved. The constraints for the exercise 1 are definitely the following:

$$\begin{cases} F_2 = -P\overline{j} \\ M_2 = M \\ \nu_1 = \theta_1 = 0 \rightarrow (fixed) \\ \nu_3 = \theta_3 = 0 \rightarrow (fixed) \end{cases}$$

By applying them, specially the clamped constraints $\nu_1, \theta_1 = 0$ and $\nu_3, \theta_3 = 0$ lets us erase the rows and columns 1,2,5,6 of the global stiffness matrix. Therefore, once performed this operation

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ -P \\ M \\ F_{3y} \\ M_3 \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 \\ & & 24 & 0 & -12 & 6L \\ & & & 8L^2 & -6L & 2L \\ & & & & 12 & -6L \\ Sym. & & & & & 4L^2 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ \nu_2 \\ \theta_2 \\ 0 \\ 0 \end{Bmatrix}$$

the system gets reduced to the following 2x2 equivalent matrix.

$$\begin{Bmatrix} -P \\ M \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 24 & 0 \\ 0 & 8L^2 \end{bmatrix} \begin{Bmatrix} \nu_2 \\ \theta_2 \end{Bmatrix}$$

and its solution is straightly direct and here attached.

$$\begin{cases} \nu_2 = -\frac{PL^3}{24EI} \\ \\ \theta_2 = \frac{ML}{8EI} \end{cases} \tag{1}$$

b) Find the reaction forces and moments at the two ends (2 Marks).

The clamping of the nodes 1 and 3 and hence its inability to bend or rotate entails reactive forces and moments that are contrary to the ones present in the structure so that $\sum F$ and $\sum M = 0$ in the whole global geometry. From the Stiffness Matrix, we know that:

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ -P \\ M \\ F_{3y} \\ M_3 \end{Bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 \\ & & 24 & 0 & -12 & 6L \\ & & & 8L^2 & -6L & 2L \\ & & & & 12 & -6L \\ Sym. & & & & & 4L^2 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ \nu_2 \\ \theta_2 \\ 0 \\ 0 \end{Bmatrix}$$

and consequently, if we aim for F_{1y}, M_1, F_{3y} and M_3 that are indeed the reactions of the forces by applying the internal force equilibrium, we will get the followin algebraic system that is easily solved if we attend that the values of ν_2, θ_2 are already calculated in the prior section.

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{3y} \\ M_3 \end{Bmatrix} = \begin{bmatrix} -12 & 6L \\ -6L & 2L^2 \\ -12 & -6L \\ 6L & 2L^2 \end{bmatrix} \begin{Bmatrix} \nu_2 \\ \theta_2 \end{Bmatrix}$$

In the end, the results of the reactions of forces and moments is properly calculated in the following lines:

$$
\begin{aligned}
F_{1y} &= -12\nu_2 + 6L\theta_2 = \frac{PL^3}{2EI} + \frac{3ML^2}{4EI} \\
M_1 &= -6L\nu_2 + 2L^2\theta_2 = \frac{PL^2}{4EI} + \frac{ML^3}{4EI} \\
F_{3y} &= -12\nu_2 - 6L\theta_2 = \frac{PL^3}{2EI} + -\frac{3ML^2}{4EI} \\
M_3 &= 6L\nu_2 + 2L^2\theta_2 = -\frac{PL^2}{4EI} + \frac{ML^3}{4EI}
\end{aligned} \tag{2}
$$

c) Explain how the solving procedure changes if we replace the concentrated load and moment with uniform distributed load along the beam? (2 Marks) Imagine that the problem from Figure 1 is now changed into Figure 2 by substituting the nodal force and the moment in node 2 for a distributed load q.

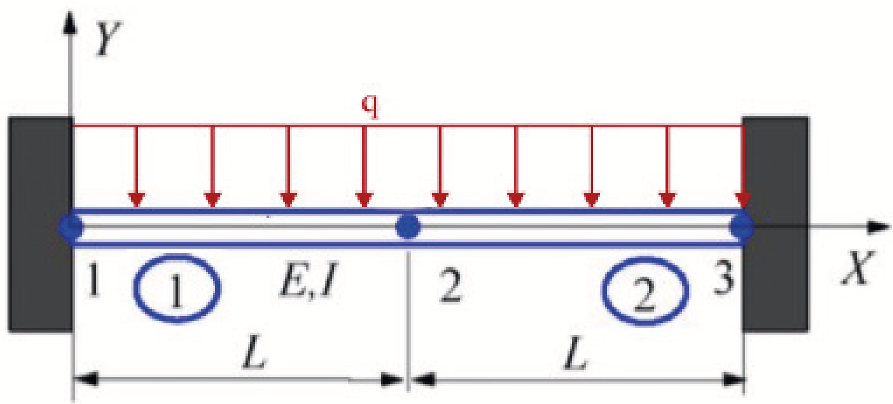
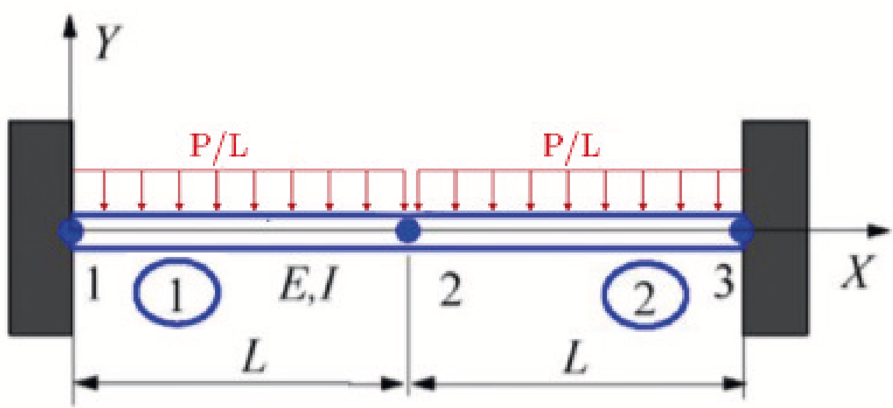
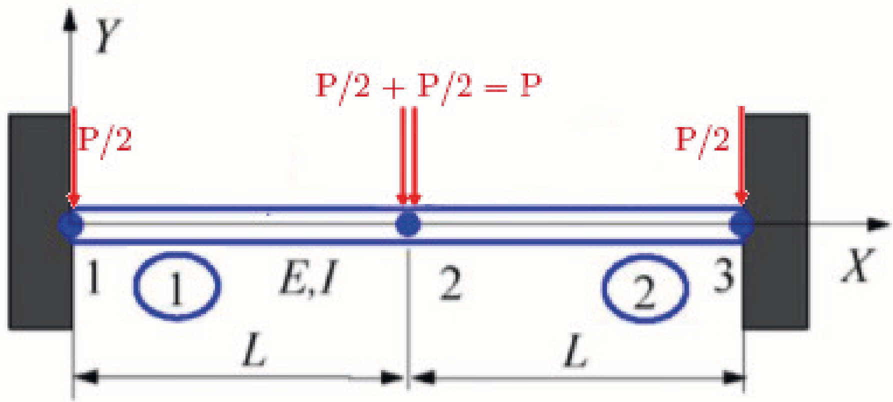


Figure 2: Fixed hinged beam subjected to a uniformly distributed load "q"

Now, this distributed load can be split into two distributed loads, one per element. Imagine, for simplicity and trying to establish some analogy with the previous exercise, that the distributed load is -for instance- $q = \frac{P}{L}$. Now we will have the following problem.



Or, if this consideration was not done, one "q" per element. Now, the distributed load, by applying the equivalence from distributed to nodal so that the conservation principles are satisfied, will result in the following problem.



which is not evidently the same problem we already solved in this exercise. If the theoretical assumption that $q = \frac{P}{L}$ was not made up, we would have qL/2 in each side and qL applied in node 2.Even though when we pass from distributed loads to puntual, the force must be applied in the centroid, however in FEM no intermediate loads (not nodals) can be considered, hence the force in the centroid is splitted in two at the nodes. However, when we calculate the moments at the nodes, the force must be considered as a single applied in the centroid. This is why, as the mesh is finer and more nodes are considered, more approach to the real problem is logically obtained and hence better results. The solution of this problem should be now done in the same manner that we have performed in sections a) and b)

Exercise 2

For the truss shown below, use symmetry to determine the displacements of the nodes and the stresses in each element. All elements have E = 30*10^6 psi. Elements 1, 2, 4, and 5 have A =10 in^2 and element 3 has A=20 in^2 . (No need to change units)

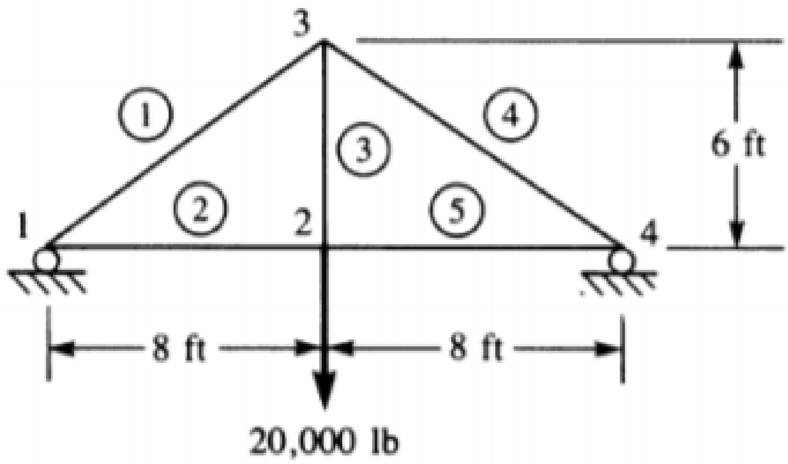


Figure 3: Symmetric truss subjected to a vertical force

In order to proceed with the resolution of the problem with the FEM Truss element (equally known as bars), several assumptions must be made to justify the mathematical model. These are the following:

i. The bar cannot hold shear forces or bending moments. In the same manner, the effects of transverse displacement are conveniently neglected.

ii. The Hooke Law is applied through the elastic expression $\sigma_x = E \cdot \varepsilon_x$

iii. For each element, there will not be any point applied load in the medium: they will also be considered at the nodes (ends).

Here, the reflective symmetry using the symmetry axis from nodes 2-3 allows us to simplify and significantly reduce the calculations for the problem by one half. This can be applied as the shape, load positions, material properties, size and boundaries are symmetric in that plane.
However, when a problem is reduced by reflective symmetry, some considerations must be carefully and properly applied.

i. For loads that are in the plane of symmetry, the reduced problem must have half of the load. The elements, in the same manner, must decrease its cross-sectional area a by a 50%.

ii. Finally, in all the nodes contained in the plane of symmetry - this is, nodes 2 and 3 in this case- the displacement components that are perpendicular to the plane of symmetry are locked and hence have 0 value. Therefore, for this particular problem, u_2 and $u_3 = 0$ being u the horizontal displacement and v the vertical.

The reduction of the problem by applying symmetry, attending to the considerations explained above which have been also detailed in [1], results in Figure 4. In this process, the cross-sectional area of the element 3 will be reduced to half and hence it will be the same value than the others, and they will all be noted with "A".

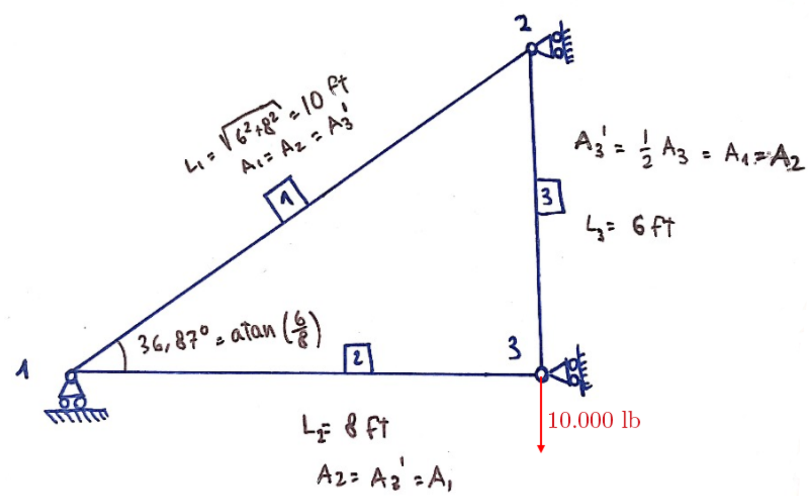


Figure 4: Reflective Reduction applied to Figure 3

As it can be seen, the boundaries are shown in a different notation -but equally valid- in which the author of this document is more used and prefers. In this way, the v_1, u_2 and u_3 are more visually perceived to be 0. Now, the connectivity table is constructed for the simplified system:

Table 2: Connectivity Table for Exercise 2

Element	Length	Nodes	$\cos(\theta)$	$\sin(\theta)$	θ
1	10 ft	1,3	0.8	0.6	36.87º
2	8 ft	1,2	1	0	0º
3	6 ft	2,3	0	1	90º

It is evident that, for the global matrix, we will need to use the transformation from local to global matrix expression:

$$k^{truss} = \frac{EA}{L}\begin{bmatrix} cos^2 & cos\cdot sin & -cos^2 & -cos\cdot sin \\ & sin^2 & cos\cdot sin & -sin^2 \\ & & cos^2 & cos\cdot sin \\ Sym. & & & sin^2 \end{bmatrix}$$

Hence, the global matrices for the elements 1, 2 and 3 will be given by:

$$k_{[1]}^{truss} = \frac{EA}{L_1}\begin{bmatrix} 0.64 & 0.48 & -0.64 & -0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix} = 2.5\cdot 10^6 \begin{bmatrix} 0.64 & 0.48 & -0.64 & -0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix}$$

$$k_{[2]}^{truss} = \frac{EA}{L_2}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} = 3.125\cdot 10^6 \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$k_{[3]}^{truss} = \frac{EA}{L_3}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} = 4.167\cdot 10^6 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

For the structure global stiffness matrix of the problem K (6x6), the vectorial sum of those individual element global stiffness matrices is required.

$$K_{[global]}^{truss} = k_{[1]}^{truss} + k_{[2]}^{truss} + k_{[3]}^{truss} = 10^6 \begin{bmatrix} 4.73 & 1.2 & -3.13 & 0 & -1.6 & -1.2 \\ & 0.9 & 0 & 0 & -1.2 & -0.9 \\ & & 3.13 & 0 & 0 & 0 \\ & & & 4.17 & 0 & -4.17 \\ & & & & 1.6 & 1.2 \\ Sym. & & & & & 5.07 \end{bmatrix}$$

Now, similarly to Exercise 1, the formula that relates the nodal displacements with the axial forces in truss $\{F\} = [k]\cdot\{d\}$ is here applied and we will have an algebraic set of equations to solve.

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ F_{2x} \\ F_{2y} \\ F_{3x} \\ F_{3y} \end{Bmatrix} = 10^6 \begin{bmatrix} 4.73 & 1.2 & -3.13 & 0 & -1.6 & -1.2 \\ & 0.9 & 0 & 0 & -1.2 & -0.9 \\ & & 3.13 & 0 & 0 & 0 \\ & & & 4.17 & 0 & -4.17 \\ & & & & 1.6 & 1.2 \\ Sym. & & & & & 5.07 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

Now, by imposing the boundary conditions or constraints of the problem:

Boundary Conditions$\begin{cases} F_{2y} = -10^4 psi \\ v_1, u_2, u_3 = 0 \end{cases}$
The solution of the problem is straightforward. We can erase the columns and rows 2, 3, 5 and get to the reduced matrix

$$\begin{Bmatrix} 0 \\ -10^4 \\ 0 \end{Bmatrix} = 10^6 \begin{bmatrix} 4.73 & 0 & -1.2 \\ 0 & 4.17 & -4.17 \\ -1.2 & -4.17 & 5.07 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_2 \\ v_3 \end{Bmatrix}$$

and therefore we must solve the next system of linear equations. This process can be done iteratively, with numerical solvers, Cramer's rule or other matrix computations. Here, the system of equations will be solved through operation and substitution procedure as it is short and easy to compute.

$$\begin{cases} 4.73u_1 - 1.2v_3 = 0 \to u_1 = 0.254v_3 \\ 4.17v_2 - 4.17v_3 = -0.01 \\ -1.2u_1 - 4.17v_2 + 5 - 07v_3 = 0 \to -0.3044v_3 - 4.17v_2 + 5.07v_3 = 0 \end{cases}$$

$$\begin{cases} 4.17v_2 - 4.17v_3 = -0.01 \\ -4.17v_2 + 4.7656v_3 = 0 \to v_2 = 1.143v_3 \end{cases}$$

And finally, substituting in this last set of 2 equations with 2 unknowns and substituting again in the prior 3 equations system we get to the final result of the displacements:

$$v_3 = -0.0168in \to v_2 = -0.01919in \to u_1 = -4.27 \cdot 10^{-3} in \tag{3}$$

Now, in order to obtain the stresses, we will need to go back with these values to the local axis system for each element (1,2,3). In order to do this, the expression of transformation of coordinates is required. However the expression $\sigma = c' \cdot d$ is ideal to be applied, being c'= $\frac{E}{L} \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{bmatrix} cos & sin & 0 & 0 \\ 0 & 0 & cos & sin \end{bmatrix}$. Therefore, the stresses are easily calculated.

$$\begin{aligned} \sigma_1 &= 2.5 \cdot 10^5 \begin{bmatrix} -0.8 & -0.6 & 0.8 & 0.6 \end{bmatrix} \begin{bmatrix} -4.27 \cdot 10^{-3} \\ 0 \\ 0 \\ -0.0168 \end{bmatrix} = -1666 psi \\ \sigma_2 &= 3.125 \cdot 10^5 \begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} -4.27 \cdot 10^{-3} \\ 0 \\ 0 \\ -0.01919 \end{bmatrix} = 1334.375 psi \\ \sigma_2 &= 4.167 \cdot 10^5 \begin{bmatrix} 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ -0.01919\ 0 \\ -0.0168 \end{bmatrix} = 995.913 psi \end{aligned} \tag{4}$$

Due to the symmetry, σ_4 will be equal to σ_1 and similarly, σ_5 to σ_2. In contrast, σ_3 will not double its value as it will also double its area, then the value calculated is the correct and final.

Exercise 3

For the following thin plate with a uniform shear load acting on its right edge.

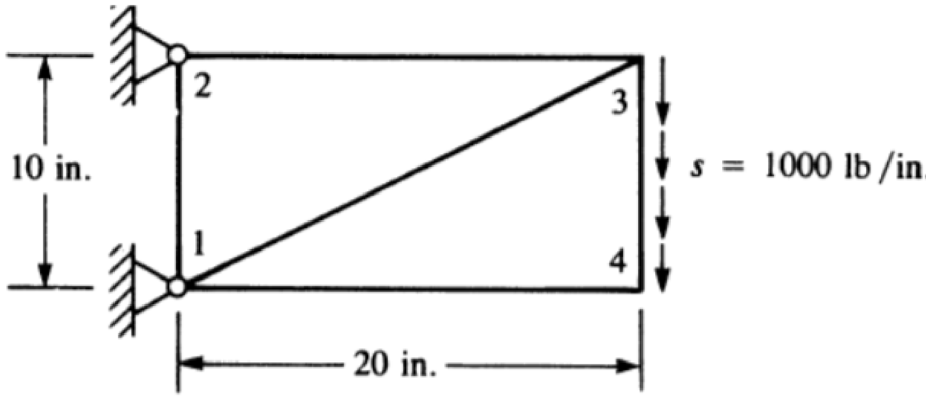


Figure 5: Thin plate with a uniform shear load acting on the right edge

a) Determine the nodal displacements and the element stresses (14 Marks). Use E= 30 x 10^6 psi, ν= 0.3 and t= 1 in. Similarly to exercise 1 subsection c), the first step to perform is to translate the uniform shear load to nodal forces. The total force applied in the centroid is sx10 in= 10.000 lb in -y direction, which is, by the FEM approach, splitted into -5000 lb in y direction at node 3 and the same value and direction for node 4 ad Figure 6 shows.

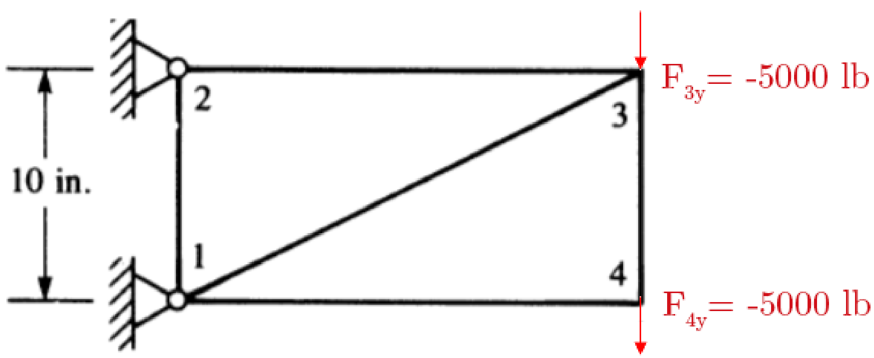


Figure 6: Thin plate with nodal force equivalence from Fig 5

Even though in the plate element the exactly same procedure applied in previous exercises cannot be applied as the elements must be studied , the connectivity table will be conveniently provided as follows:

Table 3: Connectivity Table for Exercise 3

Element	Length	Nodes	$\cos(\theta)$	$\sin(\theta)$	θ
1	10 in	1,2	0	1	90º
2	20 in	2,3	1	0	0º
3	10 in	3,4	0	-1	-90º
4	20 in	1,4	1	0	0º
5	22.36 in	1,3	0.895	0.447	26.56º

The resolution of plate elements is trickier than bars or beams consequently an scheme of the notation and elements which will be considered in the solution methodology will be referred to nodes, elements, and connections from Figure 7.

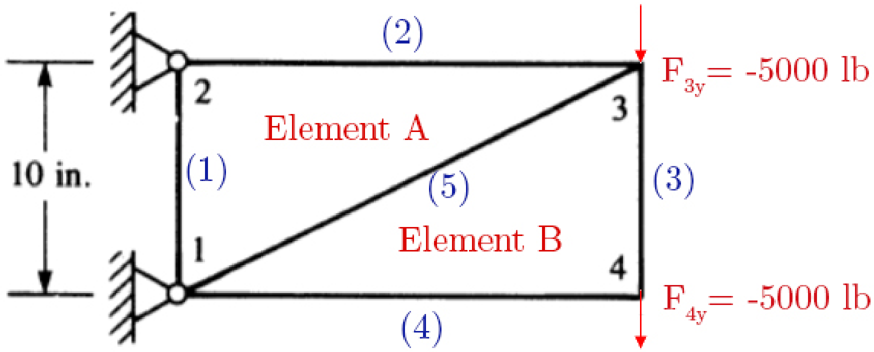


Figure 7: Scheme of the problem to solve splitting Figure 6 into parts and elements

The procedure of assembling the total stiffness matrix varies as it has been said. Now, the triangular elements considered in Figure 7 will contribute with an individual stiffness matrix given by the equations provided in the theory of plate elements, which can be withdrawn with the module's reference book from Daryl Logan in [1]. The stiffness matrices for each of the triangular elements A, B will be calculated with the expression:

$$k = t \cdot A \cdot [B]^T \cdot [D] \cdot [B]$$

It is important to remark that in the whole procedure, the isotropic material will be also assumed. Now, let's focus in the calculation of the Element A.

Triangular element A

The Area of the triangle is straightforward, the base is 10 in and the height 20 so that A=$\frac{1}{2}bh^2 = 100in^2$. For the calculus of the parameters in the B matrix, the nodal coordinates of the triangle must be provided. By quick geometry visualization, they stand for the values:

$$\left.\begin{array}{r} (x,y)_1 = (0,0) \\ (x,y)_2 = (0,10) \\ (x,y)_3 = (20,10) \end{array}\right\}$$

Being the strain-displacement [B] matrix given by the expression:

$$[B] = \frac{1}{2A}\begin{bmatrix} \beta_1 & 0 & \beta_3 & 0 & \beta_2 & 0 \\ 0 & \gamma_1 & 0 & \gamma_3 & 0 & \gamma_2 \\ \gamma_1 & \beta_1 & \gamma_3 & \beta_3 & \gamma_2 & \beta_2 \end{bmatrix}$$

The parameters β and γ, according to [1], computed through the following parametric formulas:

$$\begin{cases} \beta_1 = y_3 - y_2 & \beta_3 = y_1 - y_2 & \beta_2 = y_1 - y_3 \\ \gamma_1 = x_2 - x_3 & \gamma_3 = x_2 - x_1 & \gamma_2 = x_3 - x_1 \end{cases}$$

$$\begin{cases} \beta_1 = 10 - 10 = 0 & \beta_3 = 10 - 0 = 10 & \beta_2 = 0 - 10 = -10 \\ \gamma_1 = 0 - 20 = -20 & \gamma_3 = 0 - 0 = 0 & \gamma_2 = 20 - 0 = 20 \end{cases}$$

And consequently the matrix B of the Constant Strain Triangle (CST) Element is now easily calculated

$$[B] = \frac{1}{2A}\begin{bmatrix} 0 & 0 & 10 & 0 & -10 & 0 \\ 0 & -20 & 0 & 0 & 0 & 20 \\ -20 & 0 & 0 & 10 & 20 & -10 \end{bmatrix}$$

At this point, the plane stress is computed by the constitutive matrix [D]

$$[D] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} = \frac{3\cdot 10^6}{1-0.09}\begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & \frac{0.7}{2} \end{bmatrix}$$

Now, we know, for element A, all the values of the variables and matrices to obtain the local stiffness matrix. We will use both matrices B and D to relate stresses with displacements. The remaining procedure is just matrix operation.

$$[B]^T \cdot [D] \cdot [B] = \frac{E}{4A^2(1-\nu^2)}\begin{bmatrix} 0 & 0 & 10 & 0 & -10 & 0 \\ 0 & -20 & 0 & 0 & 0 & 20 \\ -20 & 0 & 0 & 10 & 20 & -10 \end{bmatrix}^T \cdot \begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix} \cdot$$
$$\cdot \begin{bmatrix} 0 & 0 & 10 & 0 & -10 & 0 \\ 0 & -20 & 0 & 0 & 0 & 20 \\ -20 & 0 & 0 & 10 & 20 & -10 \end{bmatrix}$$

This last product multiplied by the thickness t=1 and Area A=100 in^2 will provide the stiffness matrix of the element A such that

$$k_A = 82417.6\begin{bmatrix} 140 & 0 & 0 & -70 & -140 & 70 \\ & 400 & -60 & 0 & 60 & -400 \\ & & 100 & 0 & -100 & 60 \\ & & & 35 & 70 & -35 \\ & & & & 240 & -130 \\ Sym. & & & & & 435 \end{bmatrix}$$

We must attend the order of the calculation follow, which is evidently established in the moment of calculating matrix B. Hence, the first two columns and rows account for (u_1, v_1),

the following 2 by (u_3, v_3) and finally the last two by (u_2, v_2). This insight will be important when the superposition or vectorial sum of the stiffness matrices of elements A and B are done to acquire the global stiffness matrix.
Now, this same procedure must be carried out for the triangular element B (consider its representation at Figure 7).

Triangular element B

The analogy or symmetry logically reveals that the triangular elements do possess the same area and hence the area of B is equal to the already calculated in A, that is, A=100 in^2. Node 1 coordinates are again (0,0), same for node 3 (20,10). The coordinates of the node 4 are, indeed, (20,0). With this magnitudes, we will compute the matrix B starting by its parameters β and γ, then we will obtain the matrix [D], and finally calculate the local stiffness matrix.

$$\begin{cases} \beta_1 = y_4 - y_3 = -10 & \beta_4 = y_3 - y_1 = 10 & \beta_3 = y_1 - y_4 = 0 \\ \gamma_1 = x_3 - x_4 = 0 & \gamma_4 = x_1 - x_3 = -20 & \gamma_3 = x_4 - x_1 = 20 \end{cases}$$

And the calculation of the [B] matrix, as it has been already done, gives the values of:

$$[B] = \frac{1}{200}\begin{bmatrix} -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & -20 & 0 & 20 \\ 0 & -10 & -20 & 10 & 20 & -0 \end{bmatrix}$$

and equally [D] is calculated, which, as it is evident, does not depend on the nodes coordinates and entails the same matrix obtained for element A.

$$[D] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} = \frac{3\cdot 10^6}{0.91}\begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix}$$

and now we are able to construct the stiffness matrix of element B such that:

$$k_B = t \cdot A \cdot [B]^T \cdot [D] \cdot [B] = 82417.6 \begin{bmatrix} 100 & 0 & -100 & 60 & 0 & -60 \\ & 35 & 70 & -35 & -70 & 0 \\ & & 240 & -130 & -140 & 60 \\ & & & 435 & 70 & -400 \\ & & & & 140 & 0 \\ Sym. & & & & & 400 \end{bmatrix}$$

As it has been already reported in matrix A, the displacements for this element matrix account first for (x,y) displacements of node 1, then the corresponding to node 4 and finally the ones relative to node 3.

Assembly of Global Matrix and solution of the problem

The global matrix K can now be assembled by the vectorial sum of both element stiffness matrices giving special attention and caution of operating and ordering the nodes properly. Once this tedious process is conducted, the global matrix K (8x8) is deducted and provided here:

$$K = k_A + k_B = 82417.6 \cdot \begin{bmatrix} 240 & 0 & 140 & 70 & 0 & -130 & -100 & 60 \\ & 435 & 60 & -400 & -130 & 0 & 70 & -35 \\ & & 240 & -130 & -100 & 70 & 0 & 0 \\ & & & 435 & 60 & -35 & 0 & 0 \\ & & & & 270 & 0 & -140 & 70 \\ & & & & & 435 & 60 & -400 \\ & & & & & & 240 & -130 \\ Sym. & & & & & & & 435 \end{bmatrix}$$

Afterwards, the boundary conditions of displacements and the applied loads at the nodes are implemented to solve this -still singular and thus unsolvable- algebraic set of equations.

$$\begin{cases} u_1, v_1 = u_2, v_2 = 0 \\ F_{3y} = -5000lb \\ F_{4y} = -5000lb \end{cases}$$

The fact that the displacements in (x,y) directions of nodes 1 and 2 allows us to reduce the global stiffness matrix once we apply the relation between forces and displacements -also known as Hooke's Law- so that $\{F\} = [K] \cdot \{d\}$. Hence, we will be capable to erase rows and columns 1,2,3 and 4. Due to this, the initial system of 8x8 gets simplified to 4x4 given by the algebraic set of equations:

$$\begin{Bmatrix} 0 \\ -5000 \\ 0 \\ -5000 \end{Bmatrix} = 82417.6 \cdot \begin{bmatrix} 240 & 0 & -140 & 70 \\ 0 & 435 & 60 & -400 \\ -140 & 60 & 240 & -130 \\ 70 & -400 & -130 & 435 \end{bmatrix} \begin{Bmatrix} u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix}$$

The solution of this system, since it is more time-consuming than the easier systems from the previous exercises, has been performed through the MATLAB software, as it has been properly learnt in the module -specifically, the last theoretical session of the first week. We will use the notation A·x=b, being A the reduced stiffness matrix, x the unknowns (displacements) and b the applied nodal forces. The process is here provided with screenshots in Figure 8.

```
>> A= 82417.6*[ 240 0 -140 70; 0 435 60 -400;-140 60 240 -130;70 -400 -130 435];
>> b=[0;-5000;0;-5000];
>> x=A\b

x =

    0.0005
   -0.0028
   -0.0006
   -0.0029
```

Figure 8: Matlab Displacement Calculation. Solution of the reduced matrix system

And consequently, in inches, the displacement solution is finally obtained

$$u_3 = 0.0005, v_3 = -0.0028, u_4 = -0.0006, v_4 = -0.0029 \tag{5}$$

b) Determine the principal stresses for each element (11 Marks) Use E= 30 x 10^6 psi, ν= 0.3 and t= 1 in. The stress must be, as it has been already detailes in each of the previous exercises, calculated internally in each element. The stress is defined as the force per surface in each element, so if F admits the elastic calculation with Hooke's Law, so will the stress. Hence, for the plate element, the stresses in the x-axis (σ_x), y-axis (σ_y) and the shear stress in the plane formed by both directions (τ_{xy}) will be computed with the formula $\{\sigma\} = [D] \cdot [B] \cdot \{d\}$ where all the vectors are indeed known. The stresses for each element will result in:

Stresses for Triangular Element A

$$\{\sigma\}_A = \frac{E}{A(1-\nu^2)} \begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix} \begin{bmatrix} 0 & 0 & 10 & 0 & -10 & 0 \\ 0 & -20 & 0 & 0 & 0 & 20 \\ -20 & 0 & 0 & 10 & 20 & -10 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0.0005 \\ -0.0028 \\ 0 \\ 0 \end{Bmatrix} =$$

$$= \begin{Bmatrix} 826 \\ 247 \\ -1584 \end{Bmatrix} psi = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{x,y} \end{Bmatrix}$$

Now, it is important to notice that we have been asked for the principal stresses. The principal stresses are calculates thorugh a transformation in which the stress environment is analyzed under the angle theta θ so that the shear stress is 0. According to [2], the principal stresses can be calculated with the formula:

$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{(\frac{\sigma_x - \sigma_y}{2})^2 + (\tau_{xy})^2}$$

So that

$$\sigma_{1,2} = 536.5 \pm \sqrt{(289.5)^2 + (-1584)^2} = \begin{cases} \sigma_1 = 2146.7psi \\ \sigma_2 = -1073.74psi \end{cases} \tag{6}$$

The angle for the principal stresses is not asked but can be calculated through the expression:

$$2tan(\theta_p) = \frac{2\tau_{x,y}}{\sigma_x - \sigma_y}$$

Identically, this same methodology will be applied to obtain the principal stresses of the element B

Stresses for Triangular Element B

$$\{\sigma\}_B = \frac{E}{A(1-\nu^2)} \begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix} \begin{bmatrix} -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & -20 & 0 & 20 \\ 0 & -10 & 20 & 10 & 20 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ -0.0006 \\ -0.0029 \\ 0.0005 \\ -0.0028 \end{Bmatrix} =$$

$$= \begin{Bmatrix} -827 \\ 294 \\ -413 \end{Bmatrix} psi = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{x,y} \end{Bmatrix}$$

And now, to transform the vectorial stresses into the principal stresses for the element B:

$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{(\frac{\sigma_x - \sigma_y}{2})^2 + (\tau_{xy})^2} = -264.5 \pm \sqrt{(558.6)^2 + (-413)^2}$$

$$\begin{cases} \sigma_1 = 430.1 psi \\ \sigma_2 = -959.3 psi \end{cases} \tag{7}$$

Exercise 4

Metamaterials refer to materials designed in micro/nanoscale to expose exotic behaviour in the macroscale. A novel cellular structure design called Fish Cell that exhibits Zero Poisson's Ratio (ZPR) proposed by our team and is shown in Fig. 4(a). The 2D geometry of Fish Cell can be defined in general by ten parameters, however, considering geometric constraints this reduces to six parameters corresponding to eight nodes. The out of plane thickness of the Fish Cell defines the extrusion depth. The original 3D printing material (Nylon PA2200) is isotropic. Experimental investigations are performed to measure crashworthiness of the ZPR feature of the manufactured metamaterial. Force-displacement response was recorded by the machine to compare with FE results. List and describe with details the procedure you will perform for the finite element simulations of this impact test, for example in Abaqus. (Use 500 Words)

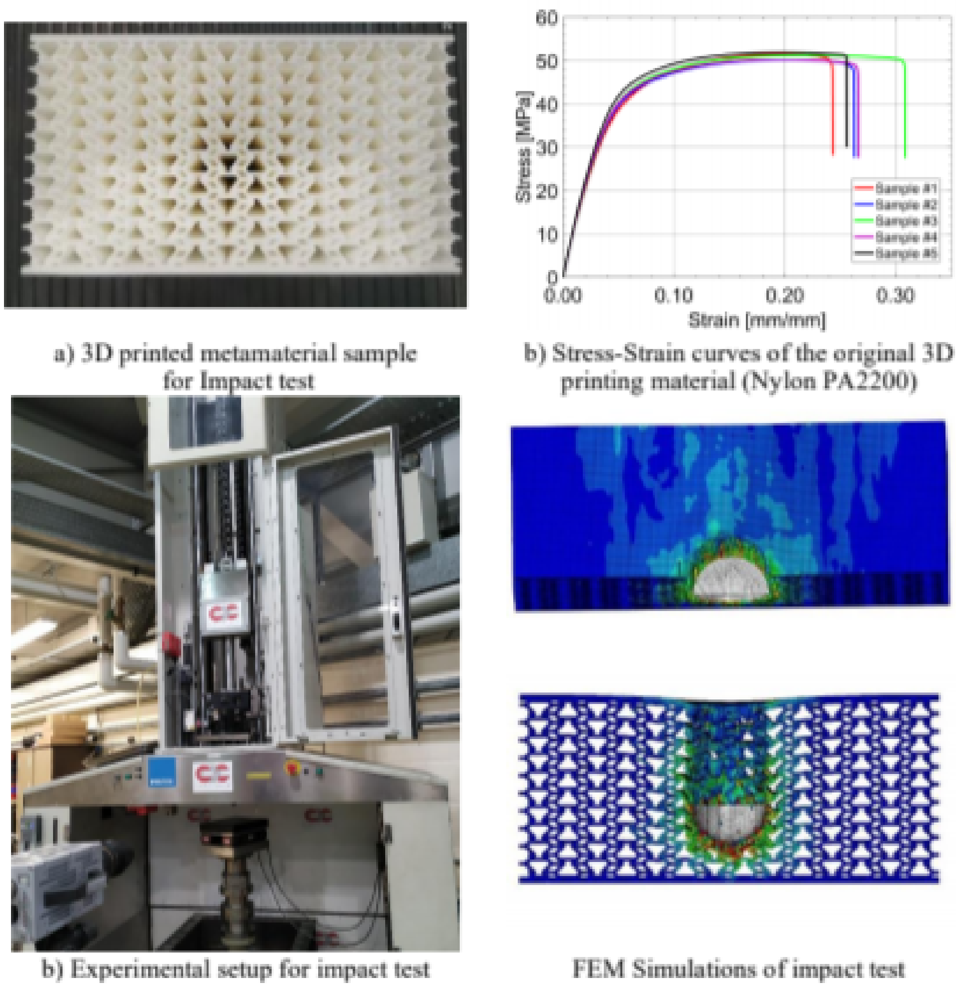


Figure 9: Impact test and simulation of 3D printed Zero Poisson Ratio metamaterials

First a brief introduction and literature review on ZPR meta-materials will be developed. Then, the procedure of the simulation, giving special attention to the plasticity introduced by the impact test will be explained. This methodology will be generally described, although will certainly follow the instructions of how should it be performed in the ABAQUS software.

Literature Review on Metamaterials and cellular solids.

Metamaterials have kept the attention of latest materials research due to their extraordinary characteristics -whilst maintaining the same composition- in many physical environment. According to (Lakes,1987) [3] they do not only affect the bending shape, as they also influence the wave-speed, strain and toughness of elements, which are indeed variables to consider.
Attending to their geometry, it is considered to be a pattern, as it is a 3-D repetition of nano-scale unitary cells over the whole domain. Depending on the shape and features of this cell, dependencies in the overall mechanic properties are noted as it is stated in (Gibson, Ashby, 1997) [4]. In this section, the Zero Poisson-Ratio metamaterials will be studied, that is, no cross sectional deformations will take place under the action of perpendicular forces. When metamaterials are studied through computational means (simulations), they are usually evaluated under linear regime asumption hence not much proves of the maintenance of their properties have been solidly obtained for large deformations (when non-linear plasticity increases its predominance). Some studies, such (Delissen et al, 2018) [5] try to enforce linearity by using geometrical optimization when they make use of spring in the geometry lattices. Other approaches to the non-linear problem have been done in recent investigations such as in (Wang et al, 2014) [6] where materials in the non-linear regime are treated with topology optimization, although the isotrophy is hard to prove and entails a significant computational expense. The topology optimization might also complicate the manufacture process. In this simulation, the non-linear plastic regime will be conveniently treated with the ABAQUS FEM solver, by defining the plasticity behaviour that is provided in the stress-strain curves of Nylon PA 2200.
The monolithic pieces printed in 3-D are made of cellular solid, which, according to (Gibson, Ashby, 1997) [4]can be defined as "interconnected network of solid trusses or plates which form the edges or faces of cells" or "collections of cells with solid edges which interconnect by packing together". Celullar solids, according to (Almandoz,2017) cellular solids are conveniently defined and characterized attending to the topology, shape, and size; which will definitely have a huge influence on the general properties.

Simulation Setup

The overall process of the simulation -for any FEM solver- will mainly consist of the following steps: geometry creation or import, material definition, mesh generation, implementation of boundary conditions (BCs) and applied loads, specification of time steps and evaluation of the results in case mesh needs to be refined. Here, the process will be further detailed giving special attention to the impact test in which the plastic effect (non-linear) will be considered.

The first step to conduct is to create the 3-D geometry (in the “parts” section in ABAQUS), model or sketch or import it from an appropriate file (CAD generated by SolidWorks or CATIA). If this last is the case, it is important to run a geometry diagnosis to identify and solve any error or mismatch of how the FEM software interprets the 3-D geometry. For this geometry, we should then create a material by introducing its mechanic properties (elastic and plastic) properties, in which the mass, density, young modulus, Poisson Ratio (0 as it is ZPR meta-material) and for the plasticity, the yield stress and plastic strain are also attached. For the yield and plastic strain, the Stress-strain plot from Figure 9 will be considered, obtaining the values of approximately:

Table 4: Plastic Regime Average for Nylon PA2200, experimental test provided in Figure 9

σ (MPa)	Plastic Strain
40	0.00
50	0.20

5 experiments on tensile tests on the Nylon PA2200 materials have been run in order to have a solid result coming from the average, even though if the most conservative approach was aimed, the maximum strain (test 3) should be considered, Here, we have used the average of them to represent the plastic behaviour.
Then, in the section material, the material we have just fully specified must be assigned to the geometry. Now, it is time to determine the FEM element to study (bar, beam, shell, plate, **solid**, ...) and equally the mesh resolution to be generated. In order to optimize it, it is necessary to further refine the zones in which the stress is supposed to be bigger (in the impact, in the central section) so as to achieve accuracy in the solution. Once done, it is now time to implement the boundaries and loads, and therefore the material probe will be encastred.
The applied loads account for the ball that is going to impact the material. Consequently, we must also model it. In order to do so, we will create a specified-radius sphere geometry (revolution) and assign the materials (for example, steel) with the mass and mechanical propertied (density, E) that it in reality owns. Then, we will generate the mesh for it (it is not necessary to be really fine). The procedure of simulating the impact will be modelled by assigning this spheric element an initial step (mechanic) with displacement (u_x, u_y, u_z) that will advance towards the metamaterial.
Now, everything has been setup except for the shear failure that will take place in the catastrophic fracture of the material. Some ABAQUS versions do not incorporate this function but it can be enabled by going to "Model" -> "Edit Keywords" and below the Plastic behaviour, write the command "Shear Failure" and the value, according to Figure 9, it must be approximately 0.24.

Now, all the whole simulation is ready to be ran and their results must be carefully evaluated, as it might present some errors or need further refinement in the metamaterial mesh.

References

[1] Daryl L Logan. *A first course in the finite element method*. Cengage Learning, 2011.

[2] Lev Davidovič Landau and Evgenii Mikhailovich Lifshitz. *Mechanics and electrodynamics*. Elsevier, 2013.

[3] Roderic Lakes. Foam structures with a negative poisson's ratio. *Science*, 235:1038–1041, 1987.

[4] Michael F Ashby and Lorna J Gibson. Cellular solids: structure and properties. *Press Syndicate of the University of Cambridge, Cambridge, UK*, pages 175–231, 1997.

[5] A Delissen, G Radaelli, LA Shaw, JB Hopkins, and JL Herder. Design of an isotropic metamaterial with constant stiffness and zero poisson's ratio over large deformations. *Journal of Mechanical Design*, 140(11), 2018.

[6] Fengwen Wang, Ole Sigmund, and Jakob Søndergaard Jensen. Design of materials with prescribed nonlinear properties. *Journal of the Mechanics and Physics of Solids*, 69:156–174, 2014.