INGENIERÍA MECATRÓNICA

DI_CERO

Diego Cervantes Rodríguez

Ingeniería Asistida por Computadora

COMSOL Multiphysics 5.6

0: Viga 2D

Contenido

DESCRIPCIÓN DEL PROBLEMA:

2.6 Analyzing a horizontal beam structure

A horizontal beam structure, shown in Figure 2.9, is made of two solid cylinders with different materials and radii. Determine the displacement and slope at the points where force or moment is applied. For Beam 1: E = 210 GPa, D = 5 cm; and for Beam 2: E = 180 GPa, D = 4 cm, where D is the diameter of the cylinder.

There are unlimited options for elements and nodes distribution, and some of these options are shown in Figure 2.10. Increasing the number of elements will definitely enhance the accuracy of the results, but only up to a certain number of elements. After this number, the results become independent of the number of elements. The first mesh contains just two elements, which is the minimum to solve this problem. The second mesh contains 3 elements, the third mesh contains 6 elements, the fourth mesh contains 12 elements, and the fifth mesh contains 24 elements. For an illustration purpose, the first mesh is selected because it has the minimum number of elements.

The first moments of inertia of the first and second beams are required to solve the problem, and they are

$$I_1 = \frac{\pi}{4}R^4 = \frac{\pi}{4}\left(2.5\times10^{-2}\right)^4 = 3.067\times10^{-7}\,m^4$$

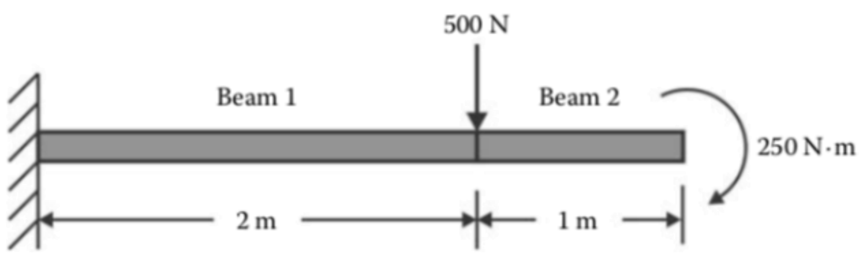


FIGURE 2.9 Beam structure.

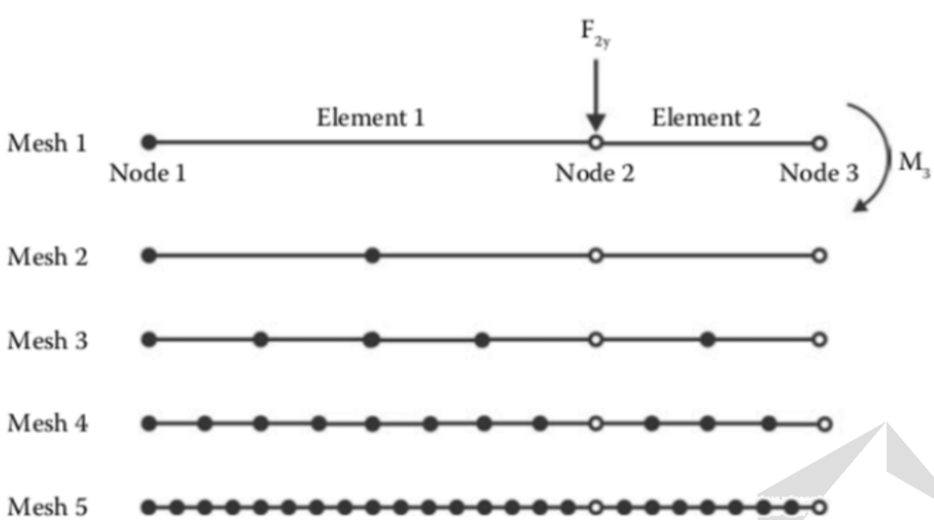


FIGURE 2.10 Meshes for the beam structure.

$$I_2 = \frac{\pi}{4}R^4 = \frac{\pi}{4}\left(2.0\times10^{-2}\right)^4 = 1.256\times10^{-7}\,m^4$$

First, the stiffness matrix for each element is obtained using Equation 2.85. For the first element, which has nodes 1 and 2, the stiffness matrix is

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ M_2 \end{Bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} d_{1y} \\ \theta_1 \\ d_{2y} \\ \theta_2 \end{Bmatrix}$$

$$[K^{(1)}] = \frac{210\times10^9\left(3.067\times10^{-7}\right)}{2^3}\begin{bmatrix} 12 & 12 & -12 & 12 \\ 12 & 16 & -12 & 8 \\ -12 & -12 & 12 & -12 \\ 12 & 8 & -12 & 16 \end{bmatrix}$$

$$[K^{(1)}] = 10^4 \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{1}} & \underline{\mathbf{1}} & \underline{\mathbf{2}} & \underline{\mathbf{2}} \end{array} \\ \begin{bmatrix} 9.66 & 9.66 & -9.66 & 9.66 \\ 9.66 & 12.88 & -9.66 & 6.44 \\ -9.66 & -9.66 & 9.66 & -9.66 \\ 9.66 & 6.44 & -9.66 & 12.88 \end{bmatrix} \end{array} \begin{array}{c} \underline{\mathbf{1}} \\ \underline{\mathbf{1}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \end{array}$$

For the second element, which has nodes 2 and 3, the stiffness matrix is

$$[K^{(2)}] = \frac{180\times10^9\left(1.256\times10^{-7}\right)}{1^3}\begin{bmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 2 & -6 & 4 \end{bmatrix}$$

$$[K^{(2)}] = 10^4 \begin{array}{c} \begin{array}{cccc} \underline{\mathbf{2}} & \underline{\mathbf{2}} & \underline{\mathbf{3}} & \underline{\mathbf{3}} \end{array} \\ \begin{bmatrix} 27.13 & 13.56 & -27.13 & 13.56 \\ 13.56 & 9.04 & -13.56 & 4.52 \\ -27.13 & -13.56 & 27.13 & -13.56 \\ 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix} \end{array} \begin{array}{c} \underline{\mathbf{2}} \\ \underline{\mathbf{2}} \\ \underline{\mathbf{3}} \\ \underline{\mathbf{3}} \end{array}$$

Assembling $[K^{(1)}]$ and $[K^{(2)}]$ using Equation 2.86 yields:

$$\begin{Bmatrix} F_{1y} \\ M_1 \\ F_{2y} = -500 \\ M_2 = 0 \\ F_{3y} = 0 \\ M_3 = -250 \end{Bmatrix} = 10^4 \begin{bmatrix} 9.66 & 9.66 & -9.66 & 9.66 & 0 & 0 \\ 9.66 & 12.88 & -9.66 & 6.44 & 0 & 0 \\ -9.66 & -9.66 & 36.79 & 3.9 & -27.13 & 13.56 \\ 9.66 & 6.44 & 3.9 & 21.88 & -13.56 & 4.56 \\ 0 & 0 & -27.13 & -13.56 & 27.13 & -13.56 \\ 0 & 0 & 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix}\begin{Bmatrix} d_{1y} = 0 \\ \theta_1 = 0 \\ d_{2y} \\ \theta_2 \\ d_{3y} \\ \theta_3 \end{Bmatrix}$$

The first and second columns and rows are deleted to remove the singularity from the stiffness matrix, and it becomes

$$\left\{\begin{array}{c} F_{2y} = -500 \\ M_2 = 0 \\ F_{3y} = 0 \\ M_3 = -250 \end{array}\right\} = 10^4 \begin{bmatrix} 36.79 & 3.9 & -27.13 & 13.56 \\ 3.9 & 21.92 & -13.56 & 4.56 \\ -27.13 & -13.56 & 27.13 & -13.56 \\ 13.56 & 4.52 & -13.56 & 9.04 \end{bmatrix} \left\{\begin{array}{c} d_{2y} \\ \theta_2 \\ d_{3y} \\ \theta_3 \end{array}\right\}$$

There are four equations and four unknowns, and solving for displacements and rotations, the results are

$$d_{2y} = -0.0279 \text{ m}$$

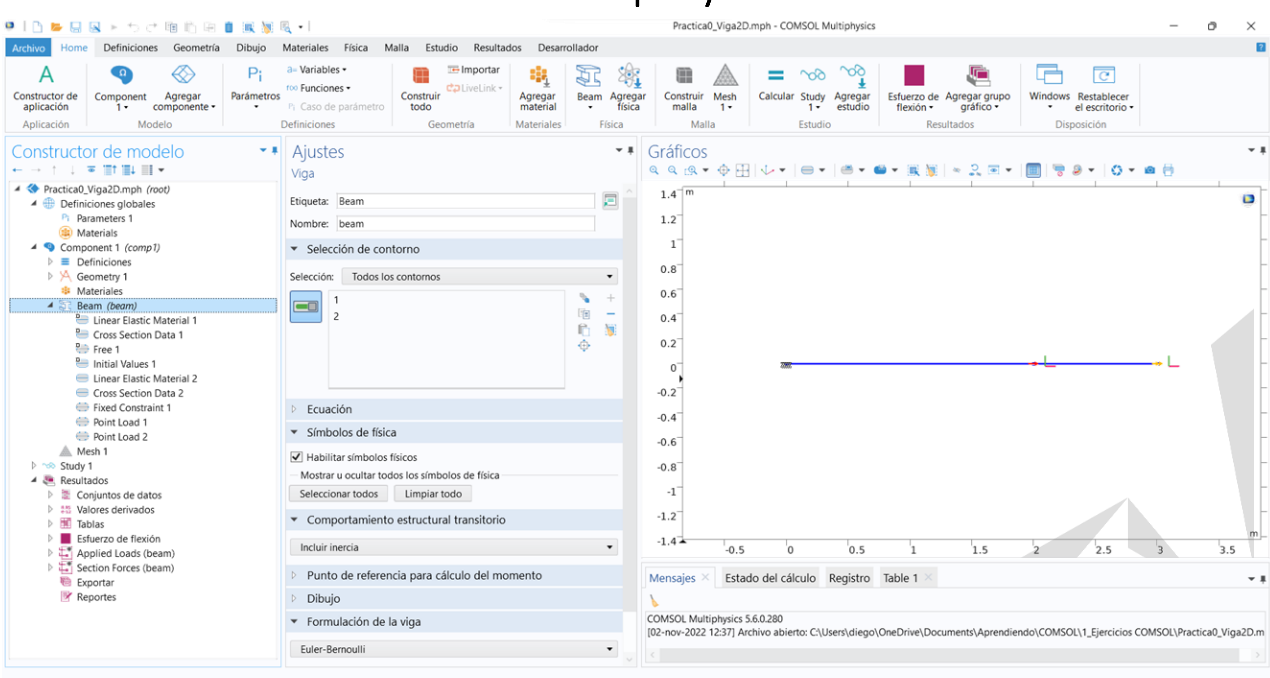
$$d_{3y} = -0.0562 \text{ m}$$

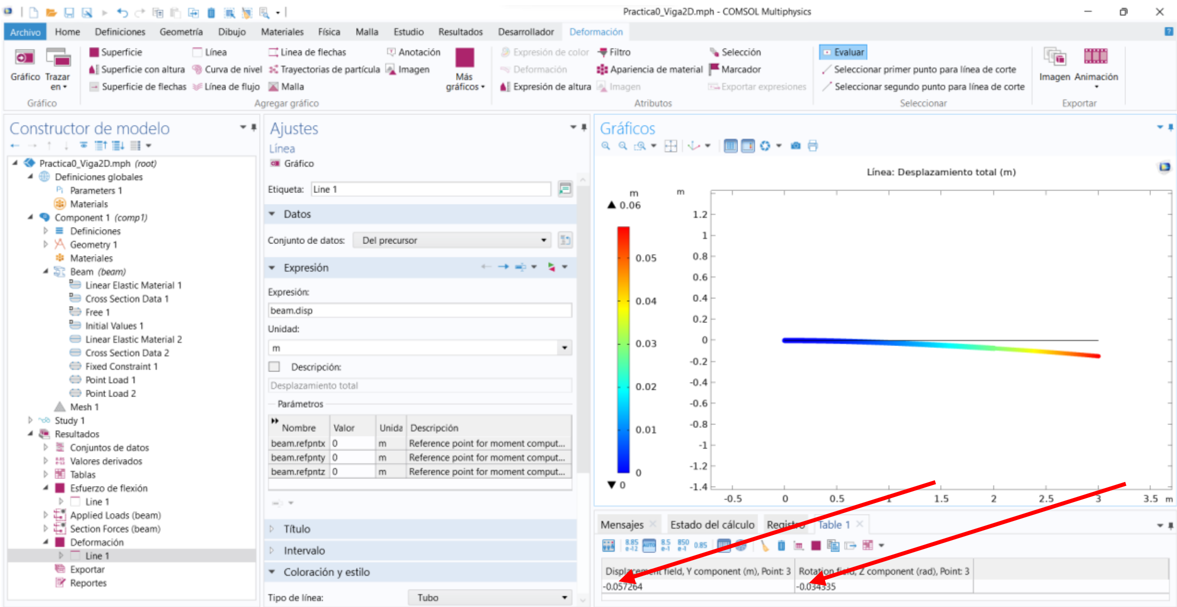
$$\theta_2 = -0.0227 \text{ rad}$$

$$\theta_3 = -0.0337 \text{ rad}$$

d_{2y} = desplazamiento vertical del nodo 2.
d_{3y} = desplazamiento vertical del nodo 3.
θ_2 = rotación del nodo 2.
θ_3 = rotación del nodo 3.

MODELO COMSOL Multiphysics:



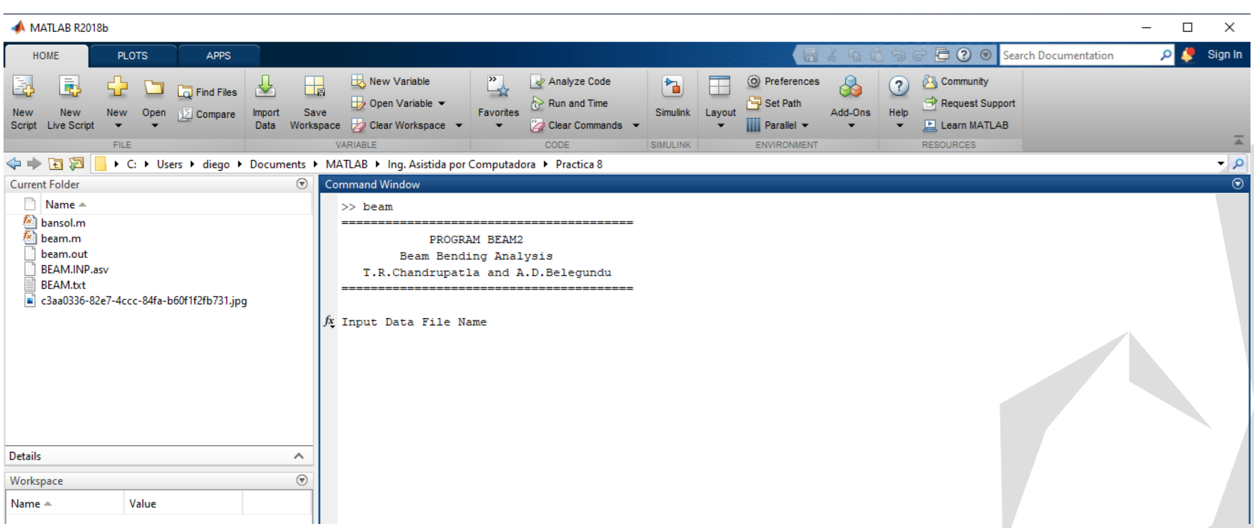


CÓDIGO MATLAB:

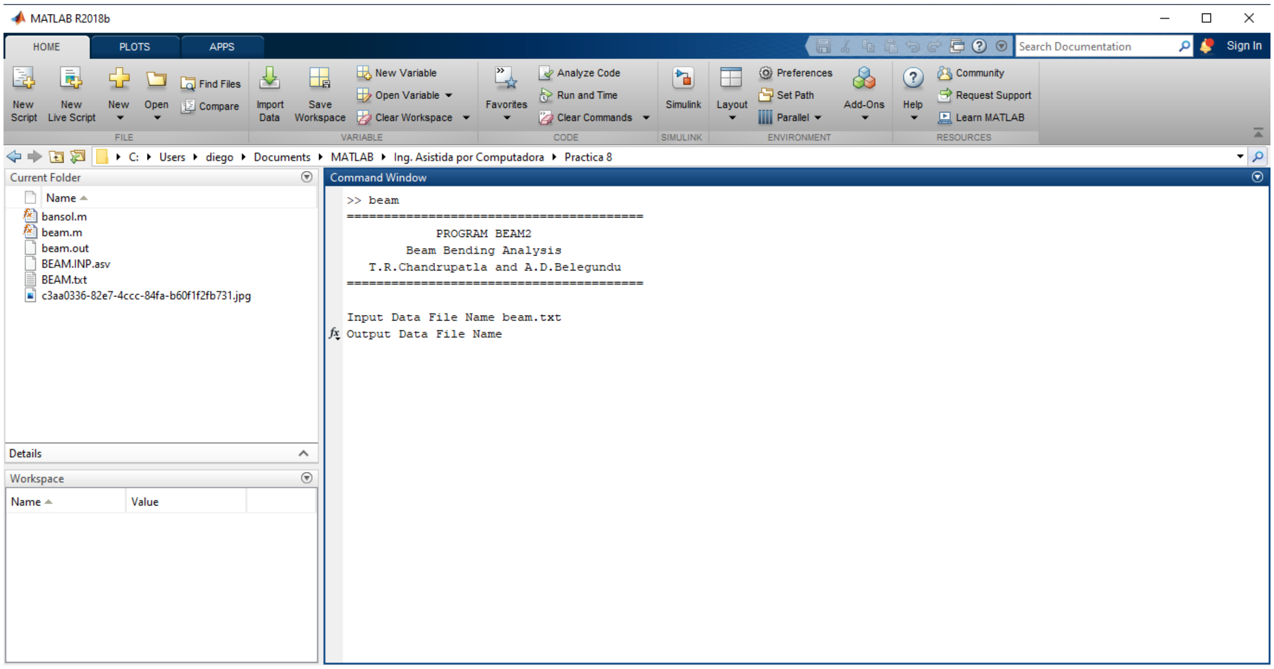
```
<< Beam Analysis using Program BEAM >>
Ejemplo de viga 2D con carga repartida triangular, constante, puntual y
momento
Con este software se calcula la deflexión e inclinaciones en una viga

NN(Numero de nodos)
NE(Numero de elementos o tramos de viga)
NM(Numero de materiales)
NDIM(Numero de dimensión, si se mueve hacia abajo, hacia arriba o rota)
NEN(Numero de nodos por elemento)
NDN(Numero de grados de libertad por nodo, osea las Q)

NN NE NM NDIM NEN NDN

6  5  1  1   2   2

ND(Numero de restricciones verticales o de rotación totales en la viga,
creados por los apoyos, pero las restricciones en x no se toman en cuenta)
NL(Numero de cargas totales, solo cargas y momentos puntuales)
Las cargas distribuidas que tenga las puedo sustituir por cargas puntuales
como se ve en la imagen que esta guardada en esta misma carpeta
NMPC(Vale 1 cuando tengamos un apoyo móvil o llamada multipunto)
 (Este tipo de restricciones no lo vamos a usar porque los apoyos no se
mueven, por eso se queda como cero)

ND NL NMPC
 2 11  0

Node#  Coordinates (Coordenadas) [metros]
 1(A)    0
 2(B)    3
```

```
 3(C)    5
 4(D)    7.5
 5(E)    10.5
 6(F)    13

Mat# se refiere al material de cada elemento
Elem#  N1  N2  Mat#  Mom_Inertia [metros^4]
 1      1   2   1       2e-4
 2      2   3   1       2e-4
 3      3   4   1       2e-4
 4      4   5   1       2e-4
 5      5   6   1       2e-4

DOF# se refiere a degree of freedom y representa a cada Q1, Q2, Q3, etc.
En este caso las unicas Q que no se mueven verticalmente son Q3 y Q9
DOF#  Displacement
 3      0
 9      0

DOF#  Load
 1     -6.75e3
 2     -4.5e3
 3     -15.75e3
 4     6.75e3
 5     -85e3
 7     -50e3
 8     -45e3
 9     -27.5e3
 10    -11.458333e3
 11    -27.5e3
 12    -11.458333e3

MAT#    E
 1     200e9
 Multi-point Constraints  B1*Qi+B2*Qj=B3
```

Ya que esté hecho este programa sin ningún comentario, nos vamos a Matlab a dónde está mi archivo y pongo beam.



Después pongo el nombre del programa



Y pongo el archivo donde quiero que se guarde mi resultado