Dimas Rizki Nugroho Adi/308816

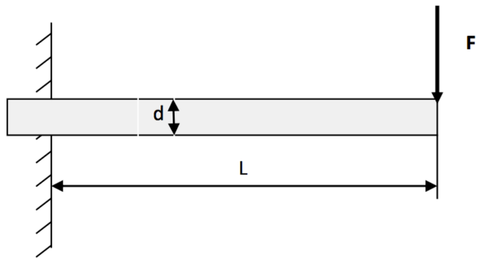
Report of Laboratory no. 6: Simulation of stress and strain distribution using finite element method

Aim of Exercise

The aim of the laboratory exercise is to calculate spatial distribution of stress and strain in various 2D shapes using the application of finite element. But in this laboratory exercise, we take advantage of MATLAB's Partial Differential Equation Toolbox and use it to solve the problem.

Description of Experiment

The simulation object is cantilevered beam supported from the left side with length of 1.5 meters, thickness of 0.2 meters and with applied load on the other side.



If the simulation object is cantilevered beam under a load then in will deformed by value **h** which can be calculated using formula below:

$$\boldsymbol{h = \frac{F.L^3}{3E.J}}$$

Where:

- F = 1000 N – loading force.
- L = 1.5 m – beam length.
- E = 1.8 * 10^8 Pa – Young's modulus
- J – moment of inertia of plane area.

For the rectangular cross section elastic modulus can be calculated from the equation :

$$\boldsymbol{J = \frac{g.d^3}{12}}$$

Where:

- g – beam thickness (for 2D problem we assume g equal to 1)
- d – beam width

Result Presentation and Analysis

➢ Drawing the Object.

Using PDE Modeler we draw a cantilevered beam with length 1.5 meter, thickness of 0.2 meters supported on the left side.

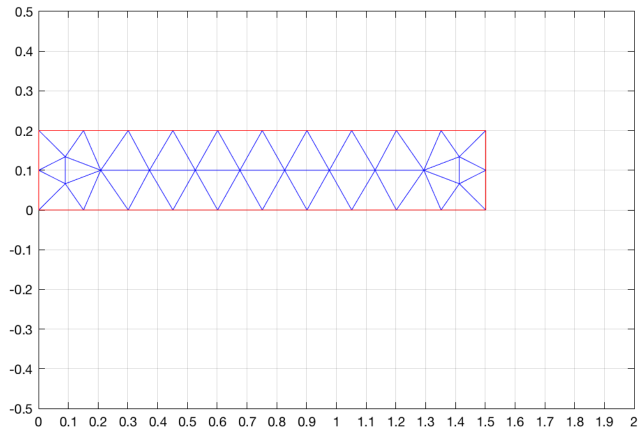


Figure 1. Cantilever Beam drew using MATLAB's PDE Toolbox in mesh mode

➢ Boundary conditions and PDE settings.

After we drew the object, to correctly model the object we need to define the boundary conditions, which are:

Side	Condition type	Surface tractions	Weights
left	Dirchlet	NA	h11, h22 = 1, h12, h21 = 0
top	Dirchlet	NA	h11, h12, h21, h22 = 0
right	Neumann	g1 = 0, g2 = -1000	NA
bottom	Dirchlet	NA	h11, h12, h21, h22 = 0

On the left side h11,h22 given value = 1, it is mean that this side will not move under pressure. g2 = -1000, it is mean that 1000 N applied downwards.

PDE Settings (these setting are appropriate for stainless steel beam) :

- Young modulus – E =2.0E11[Pa]
- Poisson ratio – nu = 0.305[Num]
- Density – rho = 7480 [$kg/m3$]

- PDE Modeler Result.

The last step are to calculate and visualize simulation. On the ***Mesh Menu*** choose ***Mesh Mode*** and then ***Refine Mesh***. Select ***Solve Menu*** and then ***Solve PDE*** . There are various options for plotting the results. We can select on ***Plot Menu***, set color to ***y displacement*** , set ***contour*** checkbox and set ***deformed mesh*** checkbox .

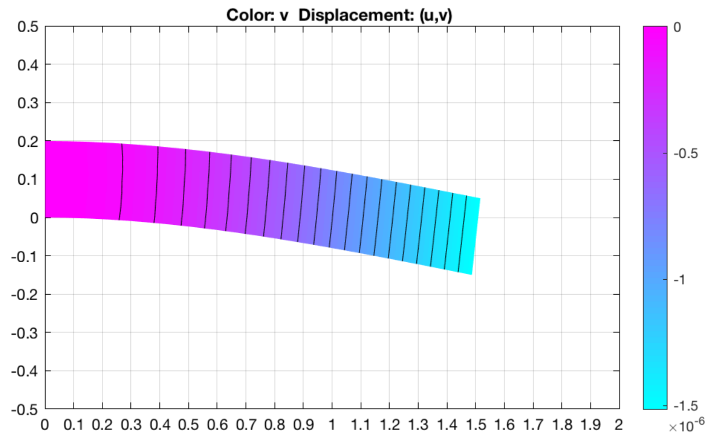


Figure 2. Cantilever Beam after 1000 N pressure from the right side

From the result we could see the high deformation on the right side of cantilever beam. When we compare the Beam end color with the scale, it refers that it deformed about $-1.5.10^{-6}$m. To obtain the precise value, we can export the solution from ***Solve Menu*** then ***Export Solution***, change the value to ***v.*** Then run the ***min*** function on it.

```
>> min(v)
ans =
  -1.5133e-06
```

- Calculation of Theoretical value.

The relevant equations to calculate the deformation on the end of beam were introduced before. We use MATLAB to calculate the ***h*** value at the very end on the right side. Here is the calculation:

```
F = -1000; % Force
L = 1.5; % Beam Length
E = 2.0 * 10^11; % Young's modulus for stainless steel
g = 1; % Beam thickness
d = 0.2; % Beam width

J = (g * d^3) / 12;
h = (F * L^3) / (3 * E * J)
```

With the result of :

```
>> m1
h =
 -8.4375e-06
```

Result Comparison

Comparison between theoretical value and calculated by tool is presented in the tabular below.

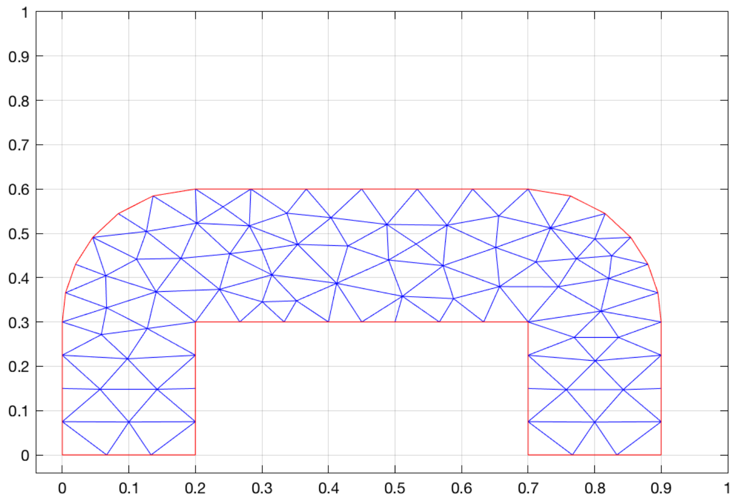
Method	Result
PDE Tool	$-1.5133 \cdot 10^{-6}$
Theoretical value	$-8.4375 \cdot 10^{-6}$

From both result we could analyze that both values are within the same order of magnitude but it is noticeable difference nonetheless. This is probably due to:

1. Parameter density was not presented in used equation while PDE Modeler takes this parameter into account.
2. There is human error on the drawing process of the object, because PDE Modeler used dimension based on authors "draw", which was bit off (grid snap was turned off).
3. Theoretical model took exact provided dimension.

More Complicated Geometry

For simulation with complicated Geometry, I try to model Simple Bridge on the PDE Modeler, as we can see below:

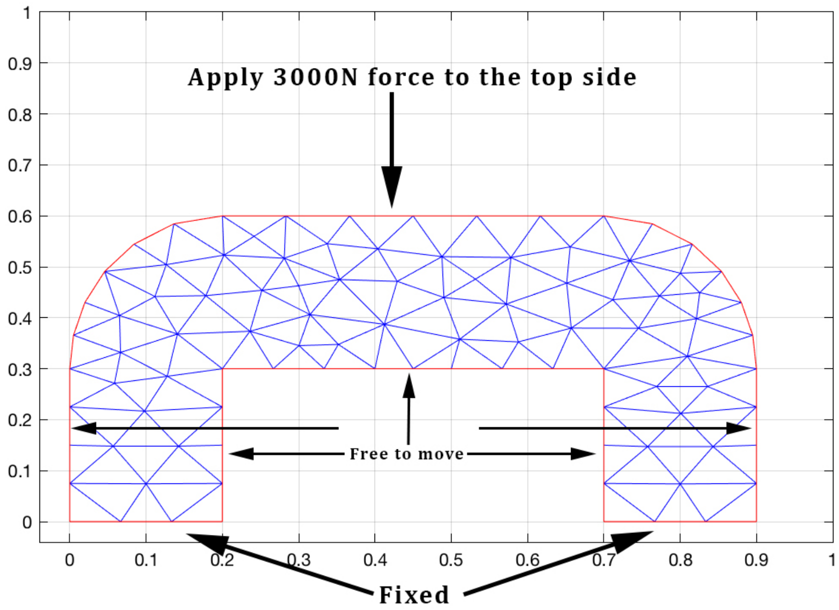


We are going to apply these boundaries and parameters settings to the model:

Side	**Condition type**	**Surface Tractions**	**Weights**
Two bottom side	Dirchlet	NA	h11, h22 = 1, h12, h21 = 0
Left side pier	Dirchlet	NA	h11, h12, h21, h22 = 0
Top side	Neumann	g1 = 0, g2 = -3000	NA
Right side pier	Dirchlet	NA	h11, h12, h21, h22 = 0
Bottom beam	Dirchlet	NA	h11, h12, h21, h22 = 0

PDE Settings :

- Young modulus – $E = 2.0E11[Pa]$
- Poisson ratio – nu = 0.305[Num]
- Density – rho = 7480 [$kg/m3$]



The Result for Complicated Geometry :

Conclusion

1. From the Simulation of stress and strain distribution using finite element in PDE Toolbox. We can learn that modeling such a complicated shape using Theoretical Calculation would be very challenging and time-consuming for us. However, with the help of PDE Toolbox, we are allowed to simulate different kind of force that affect on mesh in fast prototyping with easy to understand visualization.
2. On the first simulation, one simple model of the cantilevered beam was tested to compare the Theoretical Calculation against PDE Tool based calculation. Despite the obtained result were not the same from both approaches; the obtained results are within the same order of magnitude. The small difference on the result is probably because of human error on drawing the shape or some parameters need to be added on the theoretical calculation. It is proved that the PDE Tool is usable.