LAB REPORT: LAB 8
Simulation of stress and strain distribution using finite element method

MODELING OF PHYSICAL SYSTEMS

Patryk Gałczyński

Tuesday 8th May, 2018

1 Aim of laboratory

The aim of this laboratory was to calculate, visualize and analyze stress and strain distribution in various 2d shapes. What is more, one of the side goals was to get familiar with advanced matlab PDE Modeler tool and use it to represent previously stated problem.

2 Simulation description

2.1 Draw and mesh

First object of interest was cantilever beam with length of 1.5 meter, thickness of 0.2 meters supported on left side and with applied load on the other side. Such object has been developed in PDE Modeler with draw rectangle option (without grid snap, which is important later) with dimensions specified above.

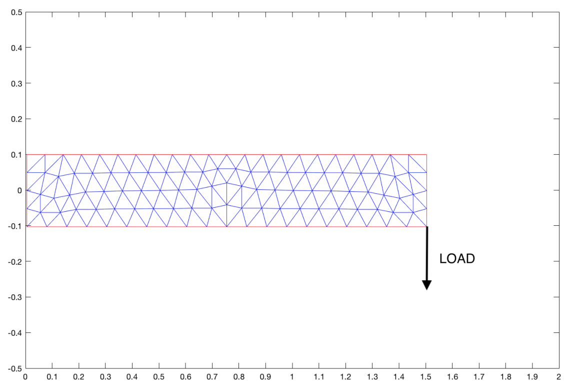


Figure 2.1: Cantilever beam in PDE Modeler in mesh mode

2.2 Boundary conditions and PDE settings

Next step was to set boundary conditions for every side, their settings are available in table below:

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

Table 1: Boundary conditions settings

Which stands for:

1. left side is fixed to wall
2. top and bottom side are free to move
3. force is applied to right side (negative force value means it is directed downwards)

Another important thing was to set PDE specification with following parameters (these parameters are appropriate for stainless steel beam):

- Young modulus - $E = 2.0E11[Pa]$
- Poisson ratio - $nu = 0.305[Num]$
- Density - $rho = 7480[kg/m^3]$

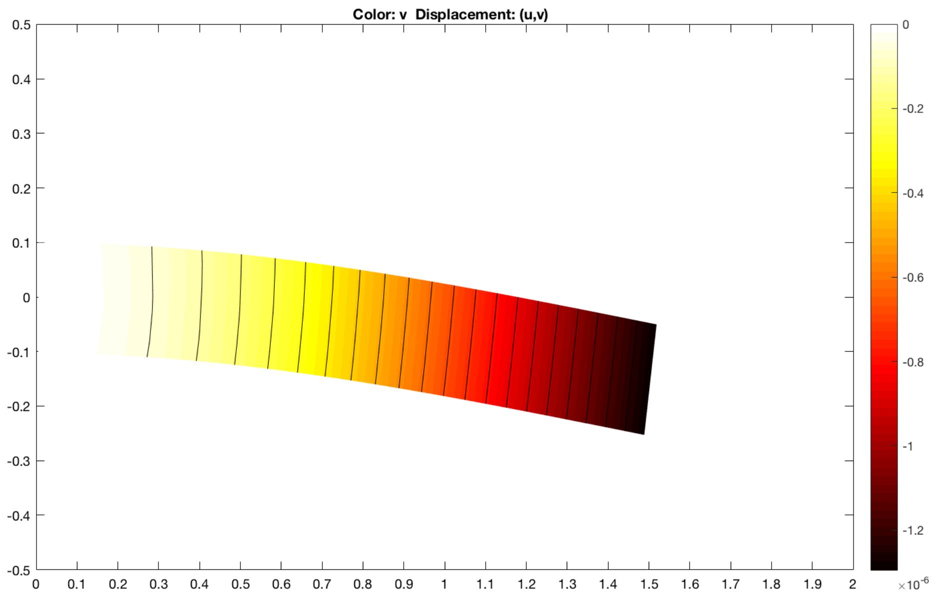
2.3 Plot settings selection

Last but not least, before obtaining results, it is necessary to set some parameters for plotter:

- set color to `y displacement`
- set contour checkbox
- set deformed mesh checkbox

2.4 PDE modeler results

Solving such model with PDE modeler gives the following chart:



It is clearly visible, that applying 100kg of weight on right end of beam results in high deformation. Beam end color when compared with scale, suggests that it deformed about between $< 1, 1.5 > \cdot 10^{-}6m$. Precise value can be obtained by exporting solution to matlab variable and running `min` function on it:

Listing 1: Obtaining minimal value of spatial deformation

```
>> min(v)
ans =
   -1.2943e-06
```

2.5 Calculate theoretical value

Same value can be calculated with following formula:

$$h = \frac{F \cdot L^3}{3E \cdot J} \quad , \quad J = \frac{g \cdot d^3}{12} \tag{1}$$

Where:

- $F = 1000N$ - loading force
- $L = 1.5m$ - beam length
- $E = 1.8 * 10^11Pa$ - Young's modulus
- $g = 1m$ - beam thickness
- $d = 0.2m$ - beam width

Calculation of such equation in matlab script is simple as:

Listing 2: Theoretical value calculation script

```
% parameters
g = 1;            % ?
d = 0.2;          % m
L = 1.5;          % m
F = -1000;        % N
E = 1.8 * 10^11;  % Pa

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

With result of:

Listing 3: calculation result

```
>> h
h =
   -9.3750e-06
```

3 Results comparison

Both generated results are presented in a table below:

Method	result
PDE Modeler	$-1.2943 \cdot 10^{-6}$
Theoretical value	$-9.3750 \cdot 10^{-6}$

Table 2: Calculation results

First of all, it is worth noticing, that both values are within same order of magnitude, which is always a good sign. However, they are a bit different. This is probably due to two main reasons:

1. theoretical model took exact provided dimensions, PDE modeler used dimensions based on authors ”draw”, which was a bit off (grid snap was turned off);

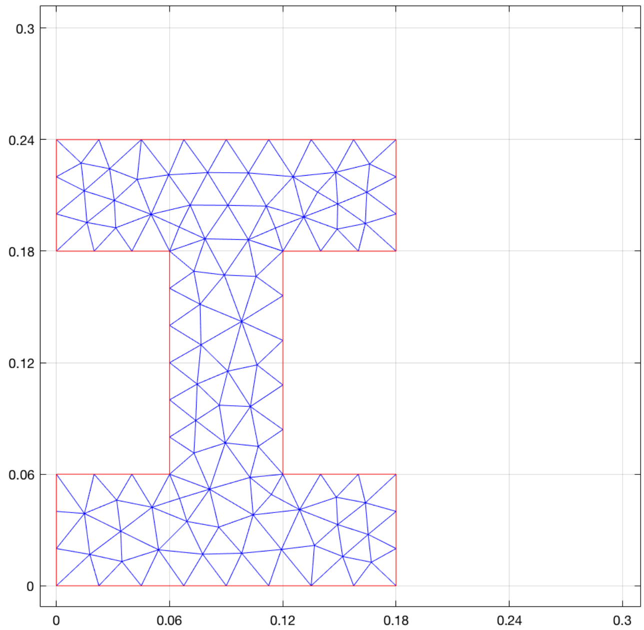
2. theoretical model uses simple calculations to obtain result, PDE modeler uses numerical approach, which is limited to its numerical accuracy and causes this difference.

4 One, more complex shape simulation

One of many reasons to use PDE modeler is that it can calculate complex shapes fairly quickly and, what is more important, it gives possibility to calculate stress and strain distribution without analytical model (most of the time, it is very hard to obtain one)

4.1 Fallen H shape

More complex shape chosen by author was so called fallen H shape, visible at picture below (mesh view):



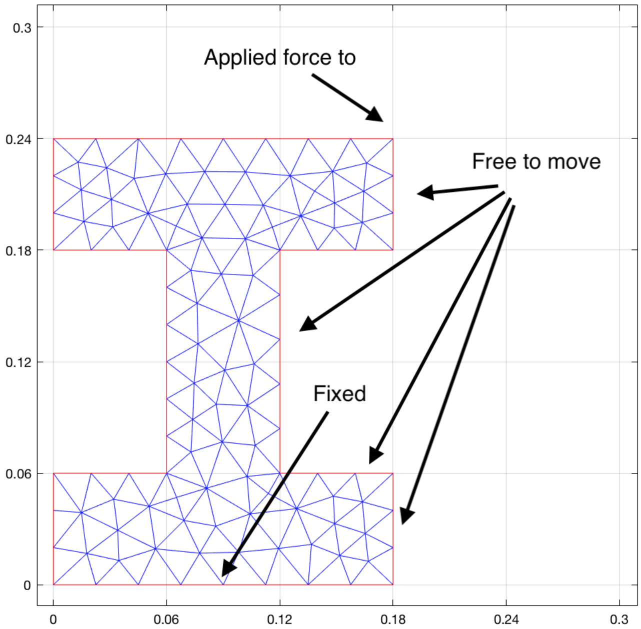
with the following settings:

- Young modulus - $E = 2.0E11[Pa]$
- Poisson ratio - $nu = 0.305[Num]$
- Density - $rho = 7480[kg/m^3]$

Side	Condition type	surface tractions	weights
bottom	Dirchlet	NA	h11, h22 = 1, h12, h21 = 0
left	Dirchlet	NA	h11, h12, h21, h22 = 0
top	Neumann	g1 = 0, g2 = -1e6*x-4e5	NA
right	Dirchlet	NA	h11, h12, h21, h22 = 0

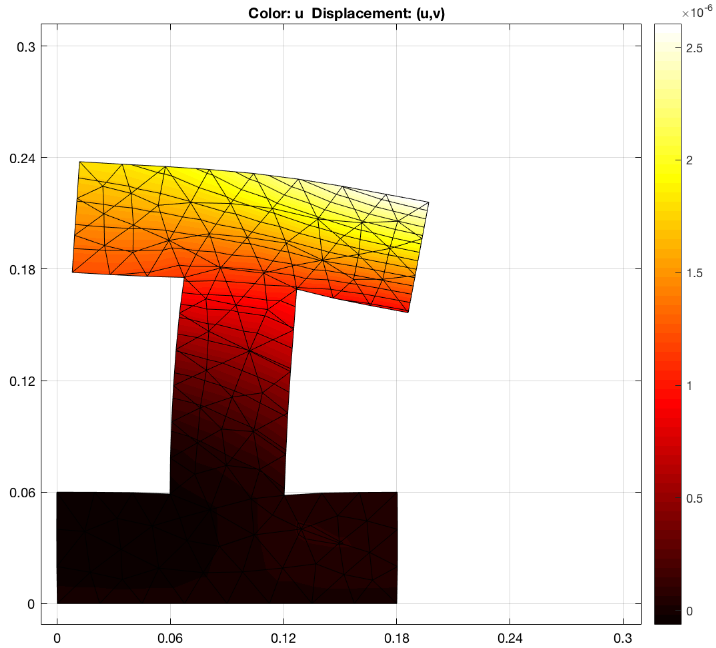
Table 3: Boundary conditions settings

Which stands for:



The force is being applied to to every part of top side, however, because it is linear function, its value gets bigger along with x axis.

Running such simulation gave the following results:



5 Conclusion

This report covered stress and strain distribution using finite element method and PDE modeler tool. One simple model was tested against theoretical and tool based calculations. Results obtained were comparable, which proves usability of PDE modeler tool. Also, another advantage of this approach came clear, it is not always possible or too time-consuming to get analytical model of some problems (like complex shape used as second example), hence numerical approach is appreciated.