C3: Approximation & Gauss Quadrature

Exercice 1

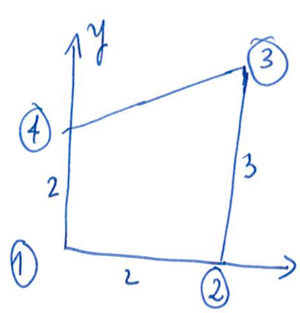
Integrate numerically $f(x) = e^x$ on $[0:6]$ using Gauss quadrature. compare with 'trapz', 'quad' Matlab's functions

$$\int_0^6 e^x \, dx = e^x \Big|_0^6 = e^6 - e^0 = e^6 - 1 \simeq 402{,}43$$

exact solution (see WOLFRAM α)

Compute the relative errors with exact solution

Exercice 2



Evaluate the integral $I = \iint_A (x^2 + y) \, dx \, dy$

over the quadrilateral shown.

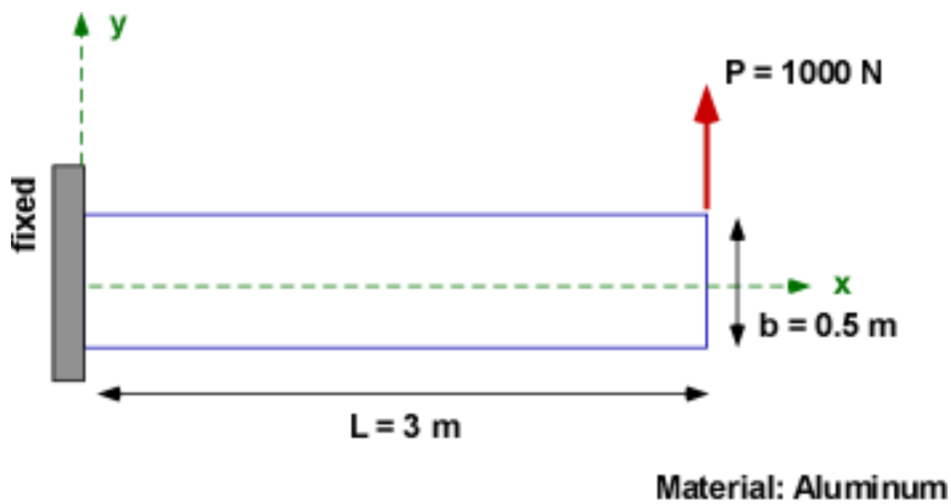
Corners: $x^T = [0 \ 2 \ 2 \ 0]$; $y^T = [0 \ 0 \ 3 \ 2]$

Using the function gaussQuad.m, find the optimal gauss points to reach tolerance 1E-4 for abs(I_exact-I_gauss)

C4: Play with a more “complex” FEA code* publish needed on LMS by pair

Using MATLAB PDE Toolbox

For **elliptical PDEs**, MATLAB provides you with a graphical tool that uses finite elements to solve the problem. However it is available only through the **Partial Differential Equation Toolbox**. The tool is started from the command line with the command ***pdetool***. This is graphical user interface (GUI) tool and requires you to follow a sequence of steps. Knowledge of finite elements is useful. We will use it as a tool to solve the 2D Elasticity problem shown below:



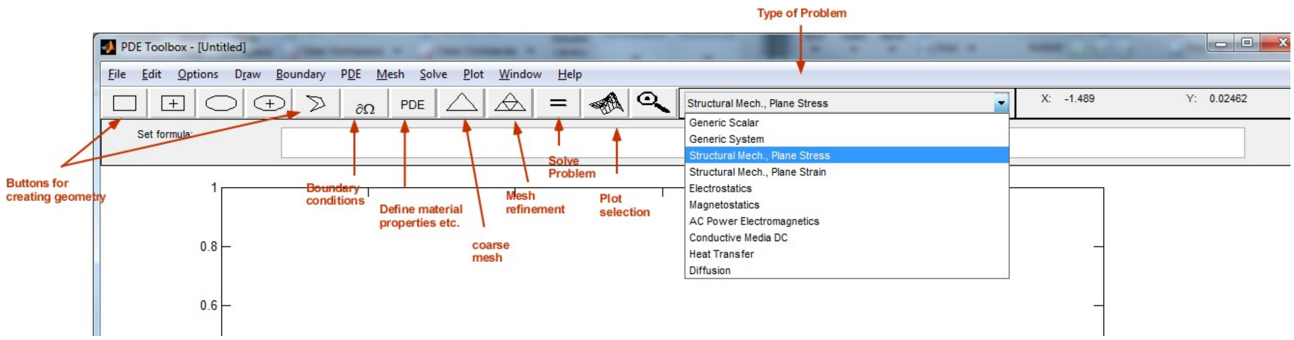
Thickness a is 1m.

The various steps in sequence of applying FEM is detailed below. It should be sufficient to get you started and solve similar problems as long as you remember to do it in sequence. I think the GUI makes it quite painless and interesting and that should be motivation enough.

We will be using the *pdetool* and therefore there is no code for solving this problem. Navigating the tool is through icons or the menu. Our problem is simple and we will use mainly the icons in the following. At the command prompt launch the tool

>> pdetool

Use the drop down menu and choose the Type of Problem : Plane Stress



Like many applications you should see a window with a menu on top, a list of icons below it, a line for formula entry, and a rectangular region for the problem.

Let us setup the problem. Remember you are solving for the displacements. You can get strain and stress from that solution.

Step 1 (Use menu to define problem region)

Options → Axes Limits (slightly more than our rectangle) Set X-axis range [-0.5 3.5]

Set Y-axis range [-0.5 1.0]

Options → Grid (Draw grid) Options → Snap

Click on Rectangle icon

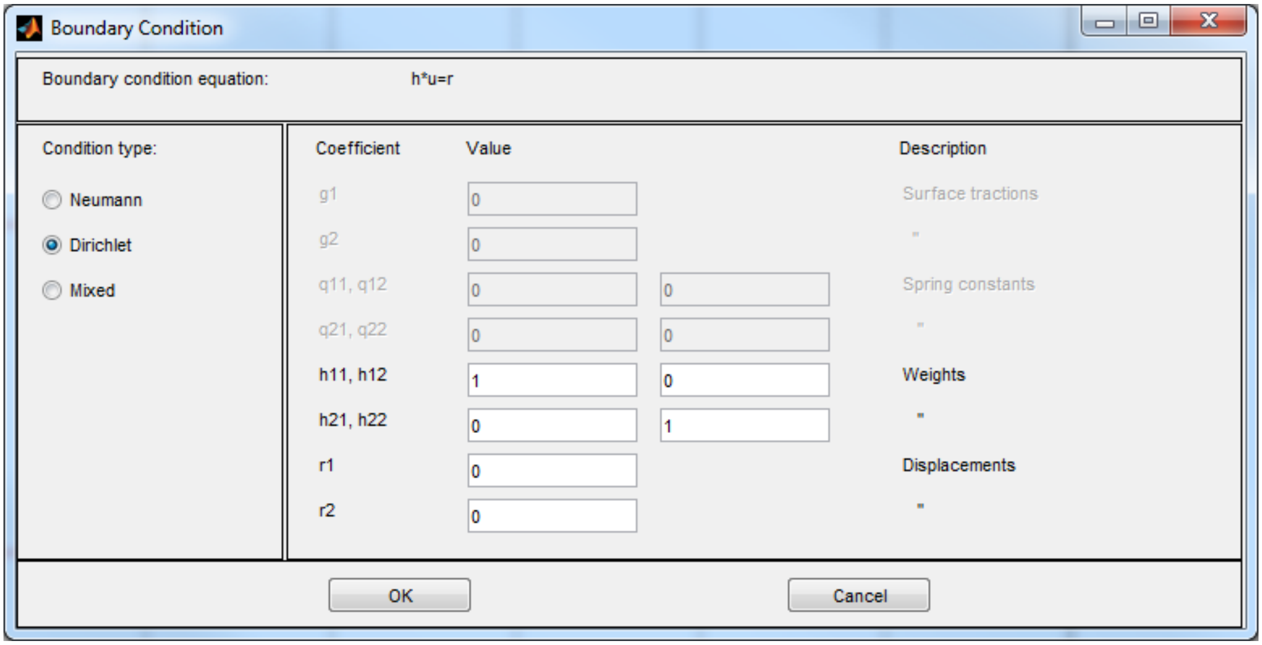
Use mouse to draw rectangle

Click at (0,0.5) and press and drag to (3,0)

Step 2 (Set Boundary condition)

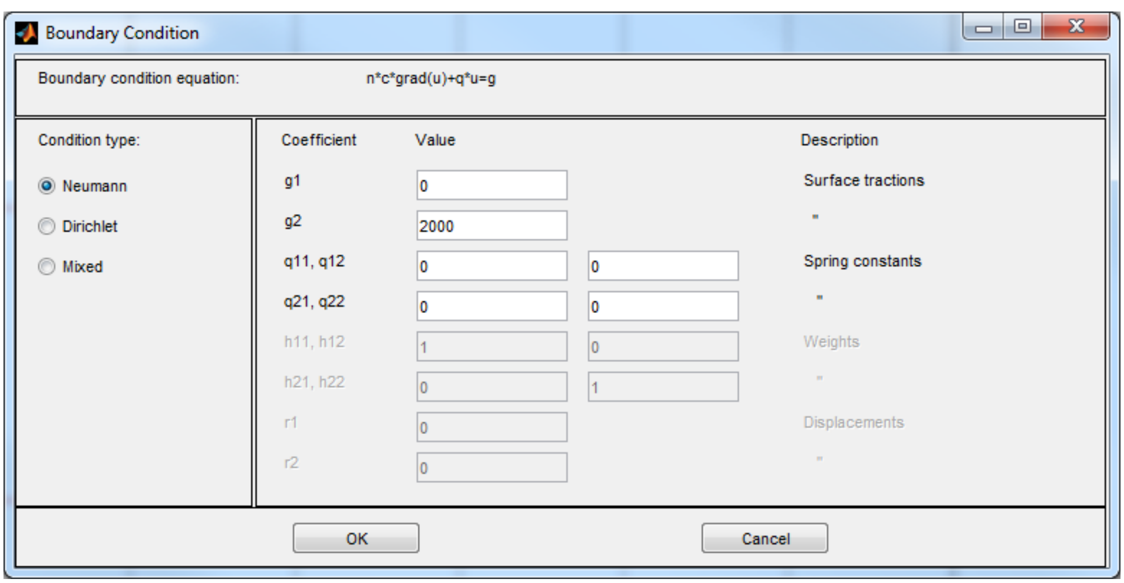
Boundary → remove all subdomain boundaries Click boundary conditions button

(i) Left Edge - Double click left edge (or x = 0 edge). There us no displacement (u = v = 0). Remember u in the figure below is the vector of u and v. We will use the Dirichlet Boundary condition.

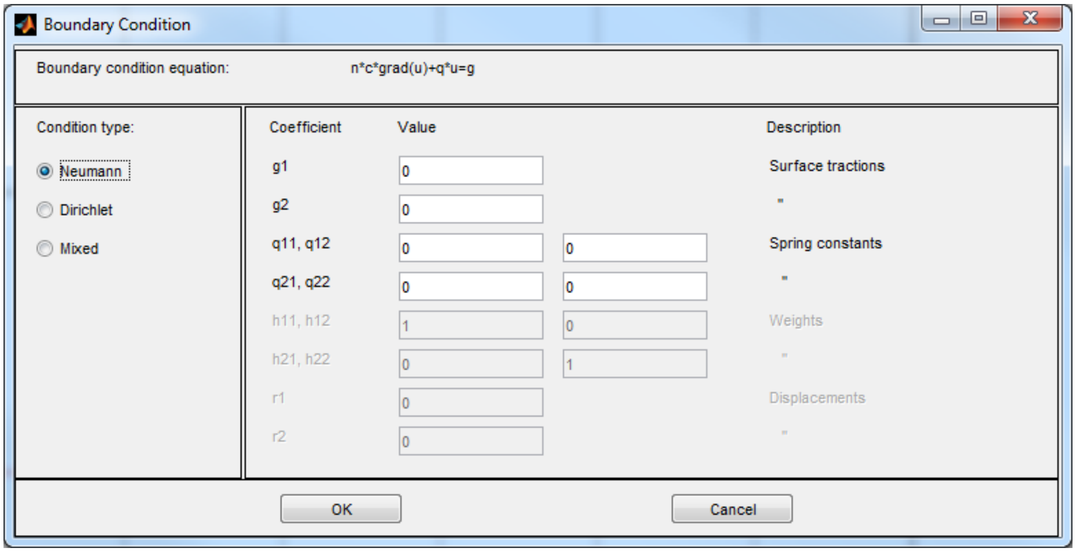


(ii) Right Boundary: This carries the load P. You cannot set up a point load. You have to specify it as a traction force - or Force/Area on the edge. That is P/b = 2000 N. We have to specify traction using the Neumann boundary condition

Double click right edge (or x = 3 edge)



(iii) Top Edge (y = 0.5) . All traction are zero. Use Neumann to accept default values



(iv) Same for Bottom edge

Step 3. Click on the PDE button to set the material properties for Aluminum

density = 2710 kg/m^3

Modulus of Elasticity (E) = 70 GPa Modulus of rigidity (G) = 26 GPa

Poisson ratio (nu) : calculate from G = E/(2(1 + nu))

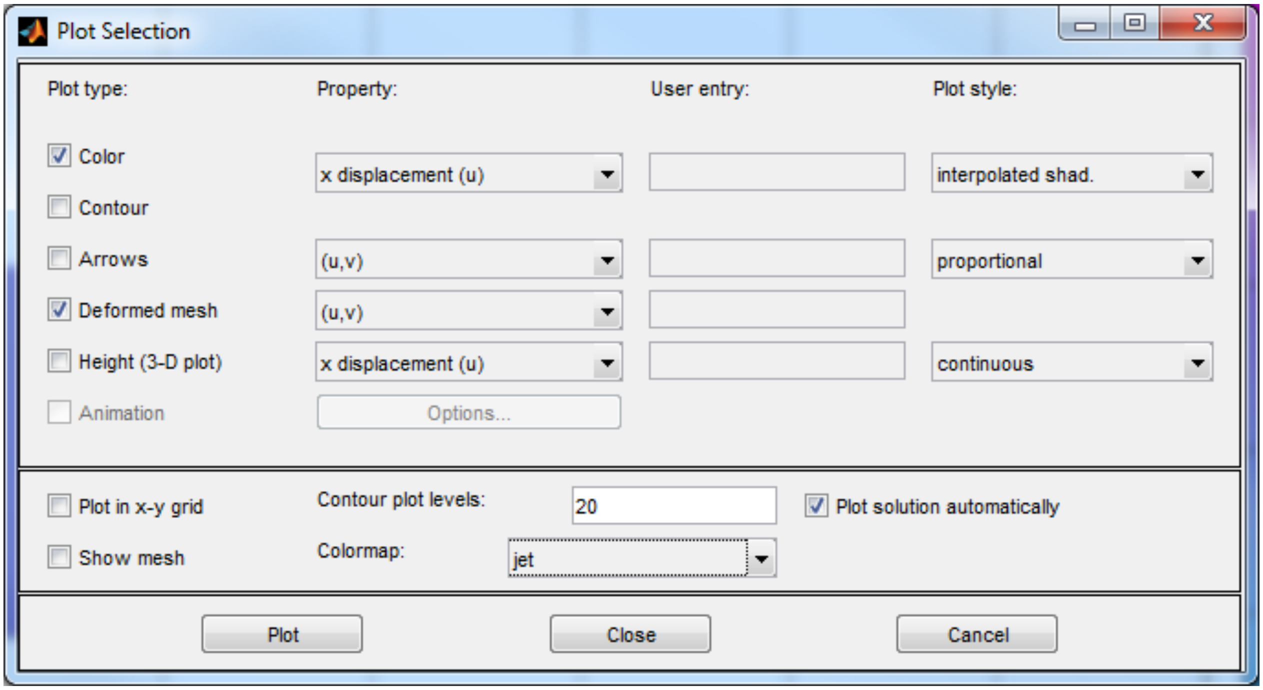
Step 4. Mesh the domain (default meshes) Click on coarse mesh

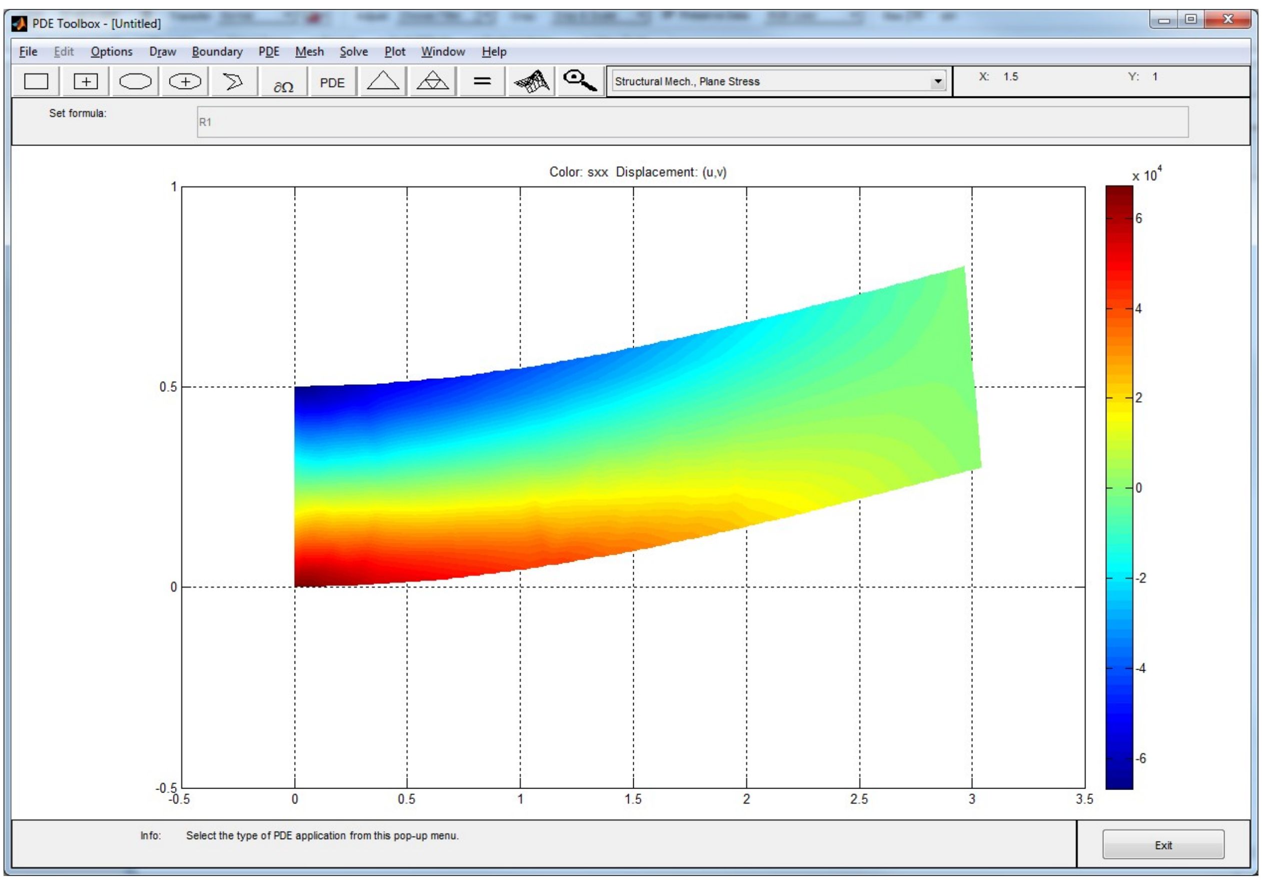
Refine the mesh twice

This program only uses triangular meshes or elements

Step 5 (Solve the problem) Click on = icon

You will see the solution. We will change the plot properties by clicking the plot button


Plot Selection
Plot type:
Property:
User entry:
Plot style:
Color
x displacement (u)
interpolated shad.
Contour
Arrows
(u,v)
proportional
Deformed mesh
(u,v)
Height (3-D plot)
x displacement (u)
continuous
Animation
Options...
Plot in x-y grid
Contour plot levels:
20
Plot solution automatically
Show mesh
Colormap:
jet
Plot
Close
Cancel


PDE Toolbox - [Untitled]
File Edit Options Draw Boundary PDE Mesh Solve Plot Window Help
PDE
Structural Mech., Plane Stress
X: 1.5
Y: 1
Set formula:
R1
Color: sxx Displacement: (u,v)
x 10^4
6
4
2
0
-2
-4
-6
1
0.5
0
-0.5
-0.5
0
0.5
1
1.5
2
2.5
3
3.5
Info:
Select the type of PDE application from this pop-up menu.
Exit

You can now Export the required variables from pdetool and create a MATLAB script

Question 1: The students should program Von Mises/Tresca criteria to conclude on ductile / fragile materials used.

VonMises (check with PDEtool interface)

$$\overline{\sigma} = \sqrt{\frac{3}{2}.\underline{dev}(\underline{\underline{\sigma}}):\underline{dev}(\underline{\underline{\sigma}})} = \sqrt{\sigma_x^2 + 3.\tau_{xy}^2}$$

Tresca

$$Maxi(\sigma_I, \sigma_{II}, \sigma_{III}) < \sigma_r$$

$$\begin{cases} \sigma_I = \frac{\sigma_x}{2} + \sqrt{\left(\frac{\sigma_x}{2}\right)^2 + \tau_{xy}^2} \\ \sigma_{II} = \frac{\sigma_x}{2} - \sqrt{\left(\frac{\sigma_x}{2}\right)^2 + \tau_{xy}^2} \\ \sigma_{III} = 0 \end{cases}$$

Question 2: Empirically fit with $\sigma_{xx} = K.(L - x).y$ Please. Comment !

Question 3: (BONUS) Add a hole center in L/2, h/2 of size defined from the relationship ($D / H = 0.6$) . Make a convergence study on sigma_max. Compare with standard stress concentration factor abaqus if possible.