



An example solution of a elastic panel

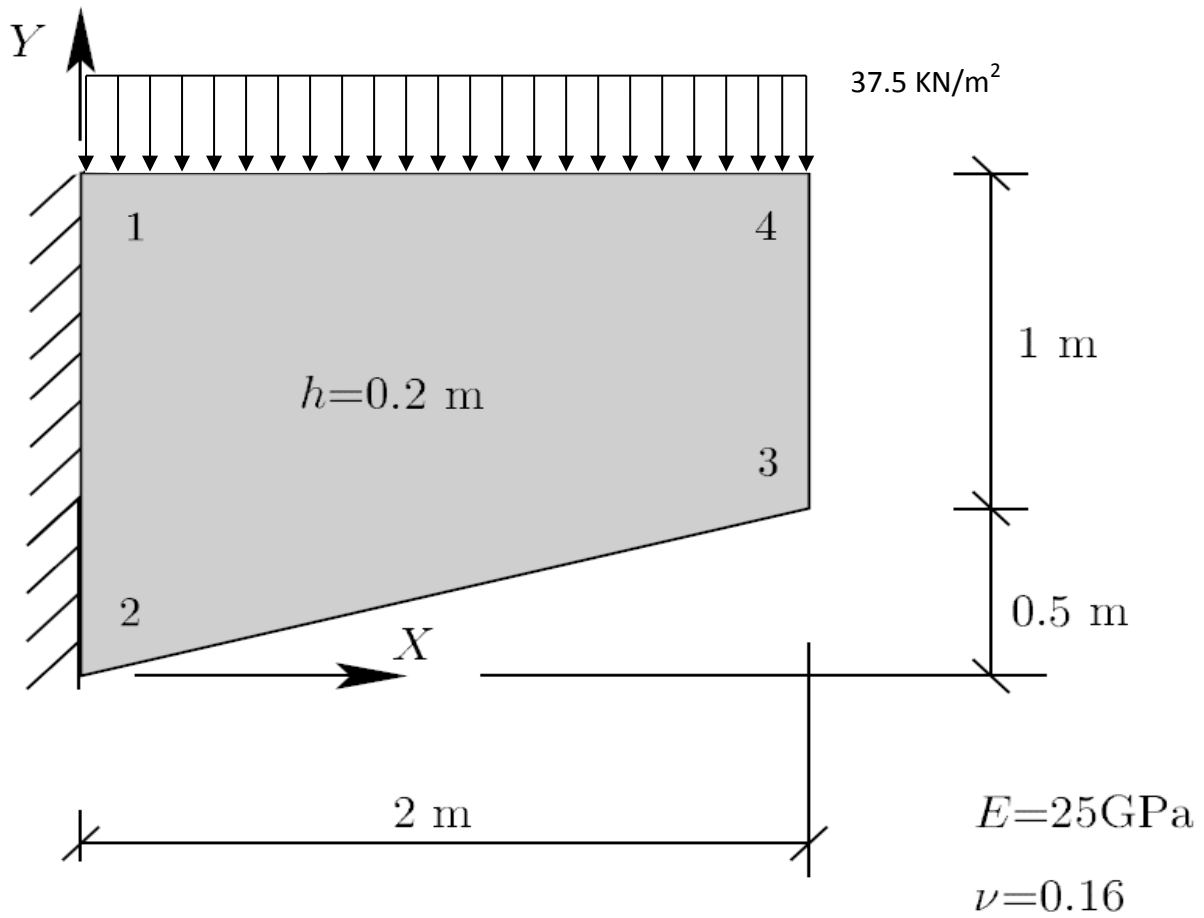


Piotr Mika

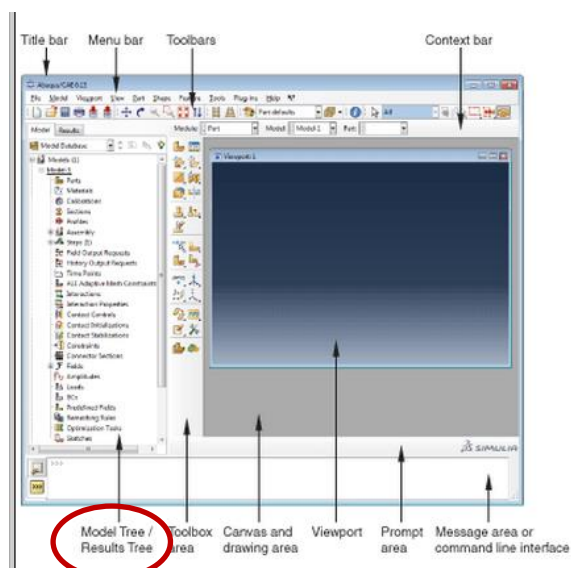
maj, 2014

1. Example- solution of the panel using ABAQUS program

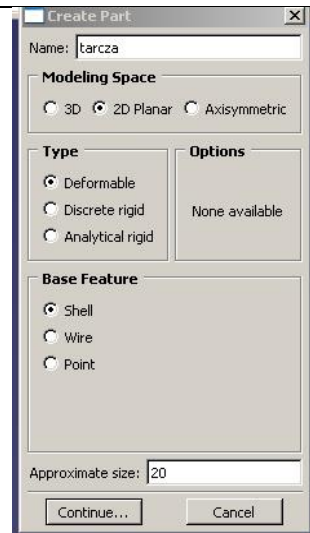
Using ABAQUS /CAE, generate a model for a panel:





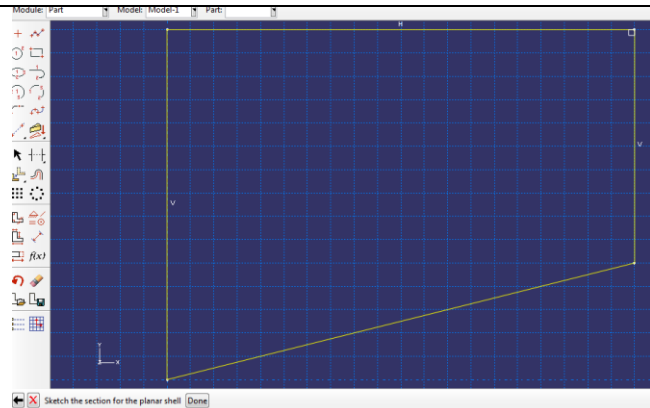
Components of the main window



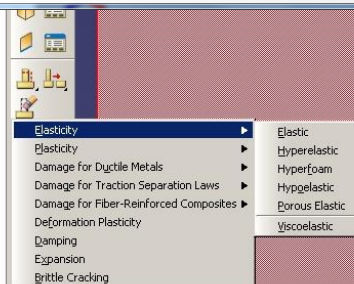
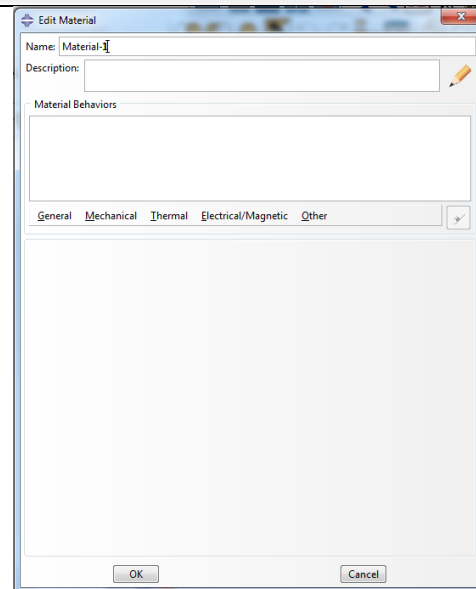
Module: Part
 [Menu/Part/Create];
 or double-click (##MB) on the Part in model tree (MT) - Create Part, Part, Name [tarcza], 2D, Deformable, Shell, Approximate size [20], Cont.



Creating a panel
 Menu/Add/lines/Connected lines
 Or select icon  Create lines,
 In the dialog box below the workspace (WS) enter the coordinates (0, 1.5), (0, 0), (2, 0.5), (2, 1.5), (0, 1.5); click mouse button 2 anywhere in the viewport (# 2MB) to finish using the selected tool; Done, F6 = 

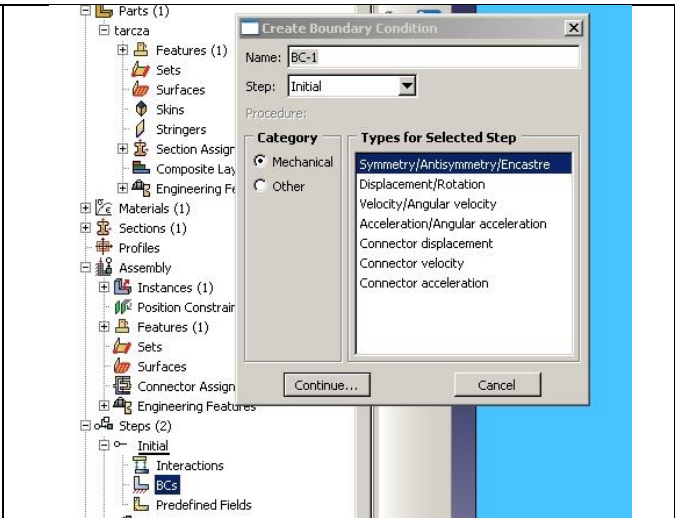


Defining Material
 Module: Property
 [Menu/Material/Manager]; Create,
 Or (##MB) on Materials in MT, Name [Material-1], Mechanical, Elasticity, Elastic, Isotropic, Young's modulus [2.5e7], Poisson's ratio [0.16], OK

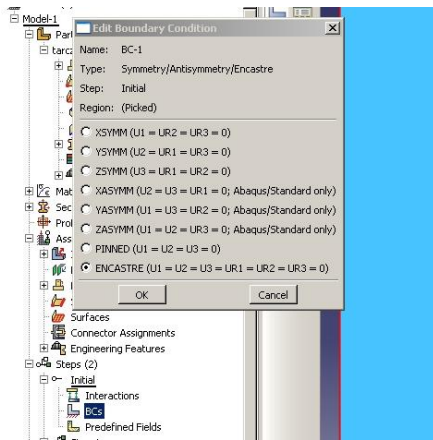


<p>Module: Property Defining Section Properties [Menu/Section/Manager/Create]; Or (##MB) on Sections in MT, Name [Section-1], Solid, Homogeneous, Cont., Material [Material-1], Plane stress/strain thickness [0.2] OK</p>	
<p>Assign Section [Menu/Assign/Section]; or in MT: under Parts/tarcza/Section Assignments, pick the panel, Done, section [Section-1], OK</p>	
<p>Module: Assembly Assembly [Menu/Instance/Create]; or (#MB) on "+" next to Assembly in MT, (##MB) Instances, Parts [tarcza], Independent, OK</p>	

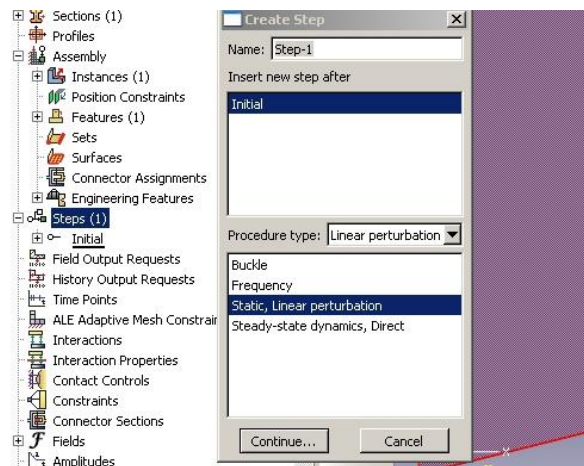
Boundary Conditions
 Module: Step
 [Menu/BC/Manager];
 or (##) on BCs in MT, Initial, name: [BC-1],
 Mechanical, Sym/Ant/Encastre, Cont.,



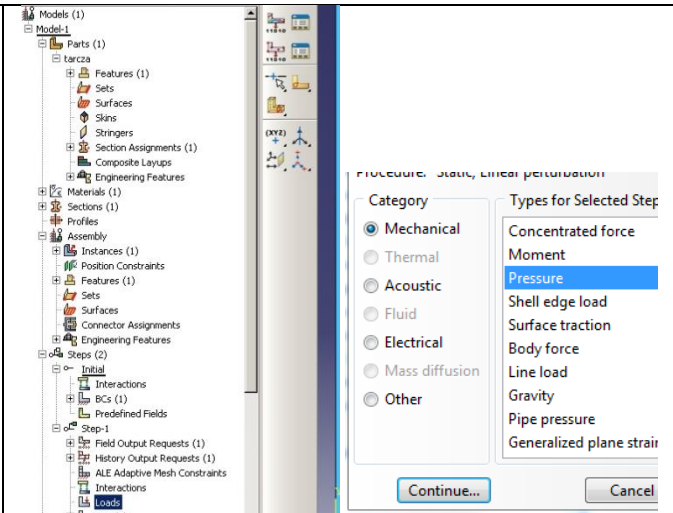
Pick the left edge of the panel, Done,
 Encastre, ok



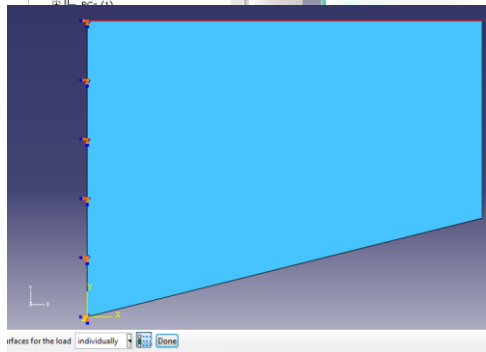
Module: Step
 [Menu/Step/Create];
 or (##MB) on Step-1 in MT, name [Step-1],
 Procedure type: Linear perturbation, Static,
 Linear perturbation, Cont., OK



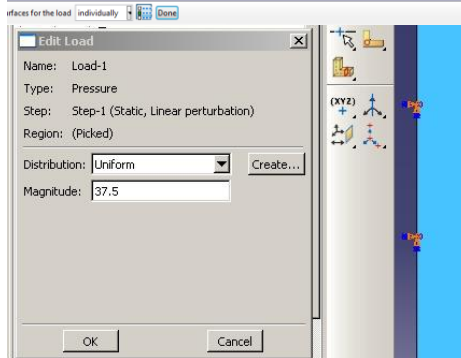
[Menu/Load/Create];
 or (##MB) on Loads in MT, name [pressure],
 step: [Step-1], Mechanical, Pressure, Cont.,



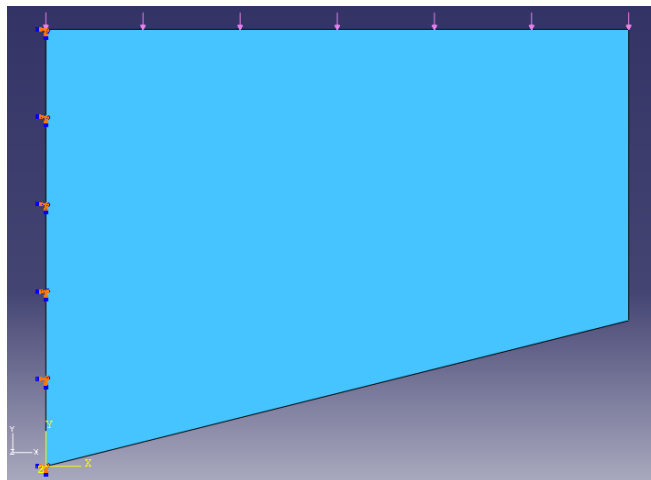
Pick the top edge of the panel, Done,



Uniform [37.5], ok



The model of the panel:



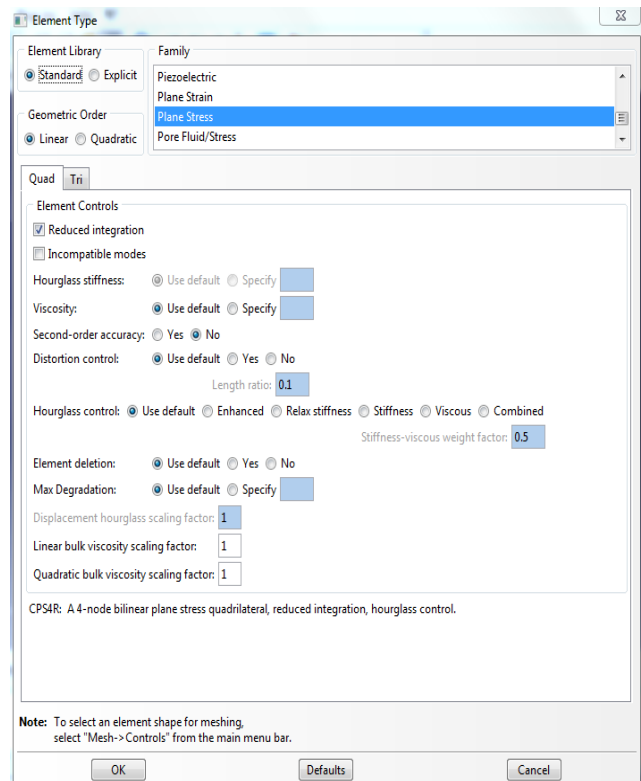
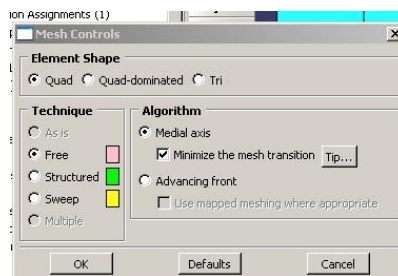
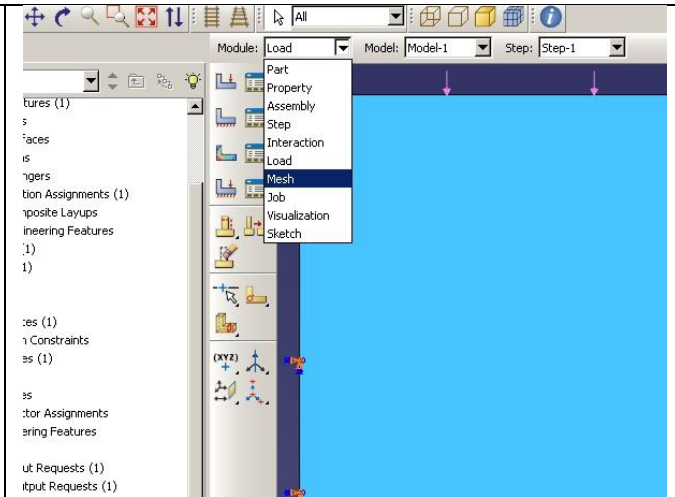
Meshing the part

Module: Mesh

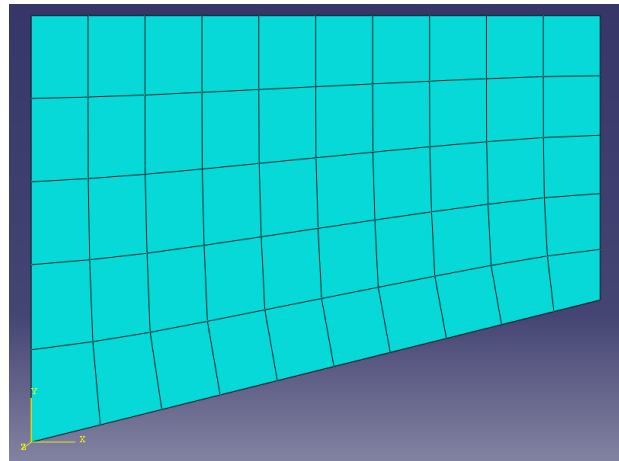
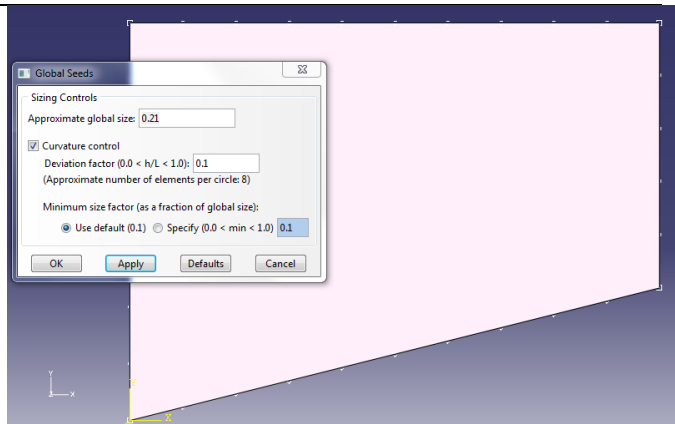
Menu/Mesh/Controls, Quad, Free, Medial axis, OK

Menu/Mesh/Element Type, Pick the whole structure, done Standard, linear, family [Plane stress] (CPS4R)

[Menu/Seed, Instance], Approximate global size [0.21], OK

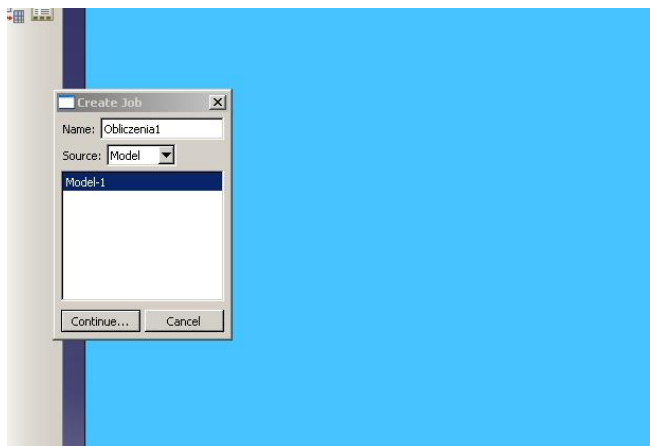


Menu/Mesh/Instance, Yes



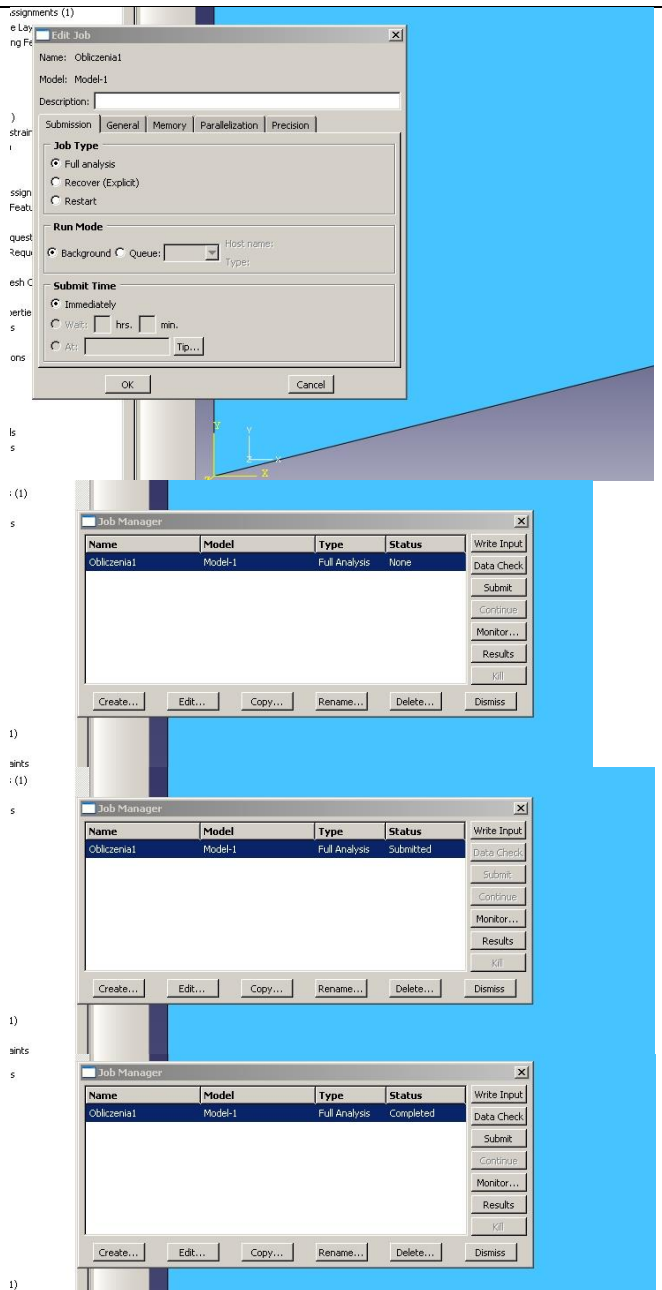
Running the analysis and visualizing the results

Module: job
[Menu/Job, Manager];
Or (##MB) on Jobs in MT, create, name
[obliczenia1], Cont, OK



(#3MB) on job obliczenia1 under Jobs, submit

When the status message indicates „completed” (#3MB) on job obliczenia1 under Jobs, results



Postprocessing

Module: visualisation, contour plot

