Application of MATLAB environment to solution of boundary and initial boundary value problems

Partial Differential Equation Toolbox (PDETOOL)

PDETOOL is a GUI application that computes Finite Element Method approximation of solutions to partial differential equations (including eigen problems) in 2D spatial domains and time.

Selected problems:

I. Plane strain state, static analysis
II. Modal analysis (eigen vibrations)

Modeling process:

1. Type command *pdetool* in MATLAB Command Window without arguments to start the application.

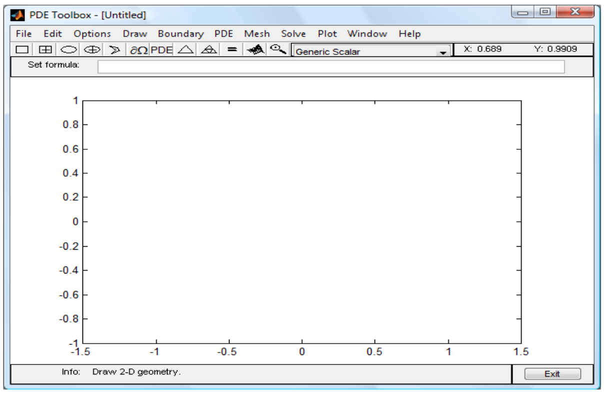

Fig. 1: GUI of PDE Toolbox

2. Select the analyzed problem (Structural Mechanics, Plane Strain) toggling down the list with *Generic Scalar* option set as the default PDE type.

3. Specify your problem parameters by clicking the *PDE* icon in the main toolbar or select *PDE→PDE Specification* in the main menu and select either

I. Elliptic problem or
II. Eigen modes

and enter E – Young modulus, nu – Poisson ratio, Kx, Ky – components of body forces, rho – density.

4. Geometry definition

Set the limits of the coordinate system *(Options→Axes Limits*) and turn on the grid (*Options→ Grid*). In order to define the geometry of the analyzed domain, use main toolbar icons (or *Draw*→… from the main menu).

Created geometrical figures are automatically named. One can modify a figure and its name clicking twice within the object. *Snap to grid* function may be turned on/off in *Options* menu in order to align objects to the grid.

Defined objects are summed up by default (e.g. R1+P1 denotes union of two figures). The default formula can be easily modified at the *Set formula* line. In order to delete an object, select it by a single click (its boundary is marked black then) and press *Delete* key.

5. Boundary condition types:

- Dirichlet b.c. (respective edges are marked with red color),
- Neumann b.c. (blue color)
- mixed (black color).

Homogeneous Dirichlet b.c. are set for each edge by default. To modify it, use the respective icon from the main toolbar or select *Boundary→Specify Boundary Conditions* from the main menu. One can just click twice the respective edge and the same dialog box should pop out. Select multiple edges with *Shift* key pressed. Subsequently specify carefully respective boundary conditions.

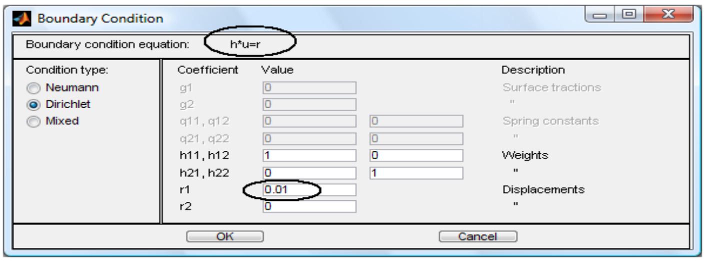

Fig. 2: Non-zero Dirichlet b.c. definition

6. Mesh generation
Use one of two icons from the main toolbar in order to create either a coarse or fine mesh. The same can be done using main menu (*Mesh→ Initialize Mesh/Refine Mesh*).

7. Solving the problem
Click the '=' button or select *Solve→ Solve PDE* from the main menu. Solution plots will be presented.

II. Upper and lower bounds of the eigen values that are to be found can be specified in the main menu (*Solve→ Parameters*).

8. Postprocessing (visualization)
Select the respective icon in the main toolbar or select *Plot→ Parameters* from the main menu in order to select solution maps, contour plot, deformed mesh, FE mesh options. *Colormap hsv* is recommended.

II. Mode shapes for selected range of eigen values can be plotted using *Eigenvalues* option.

All the data can be exported to the MATLAB Workspace in a matrix form. In particular, option *Mesh→ Export* enables saving the information about the topology (p– node coordinates, e– edges, t – triangles), option *PDE→ Export* saves the equation coefficients, option *Solve→ Export* saves the solution values.

I. After exporting the necessary information (topology + equation coefficients and solution values) one can compute components of the strain and stress tensor and the Mises equivalent stress using intrinsic MATLAB procedure pdesmech, e.g.:

```
sx=pdesmech(p,t,c,u,'tensor','sxx')
mises=pdesmech(p,t,c,u,'tensor','von Mises','application','pn','nu',0.3)
```

The prepared model data can be saved as an M-file and modified using MATLAB Editor. After running such a file, GUI is opened automatically.