Table of Contents

# 2. Problem solving with cane Multiphysics

cane Multiphysics is a framework for numerical simulations in engineering. It's input is made externally, using GiD pre-processing software to describe the parameters of the problem, whereas the output files of cane can be postpocessed in Paraview, an open source post-processing software. cane is capable of performing many different analyses that need to be performed in order to accurately model physical phenomena that occur in engineering problems. Such analyses are:

- Computational Fluid Dynamics analyses (CFD)
- Computational Fluid-Structure Interaction analyses (FSI)
- Contact Mechanics analyses
- Thermal Conduction analyses
- Plate in membrane action analyses (Plane Stress, Plane Strain)
- Isogeometric analyses (IGA)

This chapter is devoted to giving a brief introduction on how to use cane to solve Solid and Fluid Mechanics problems both separately and when interacting with each other as a coupled system. Results are validated by comparing them with known results from benchmark problems.

2.1. Steady state plane stress problem

In this problem, a uniformly distributed load is subjected to a thin plate structure as shown in the figure below. The plate is discretized using two linear triangular elements for illustration purposes. More elements must be used in order to obtain reliable results.
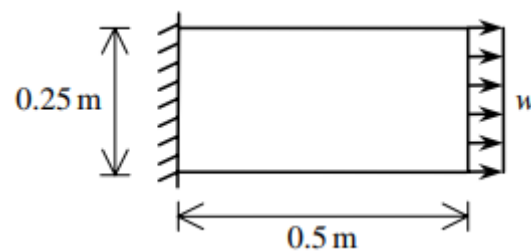

The abovementioned problem will be solved using the following parameters:

- Modulus of Elasticity: E = 210 GPa
- Poisson's ratio: v = 0.3
- Plate thickness: t = 0.025m
- Distributed load: w = 3000 kN/m2

Since the thickness is relatively small compared to the other dimensions of the plate, we can assume plane stress state for the static analysis.

2.1.1 Preprocessing with GID

Firstly, the user must specify the MATLAB GiD problem type. Select Data -> Problem Type -> matlab.
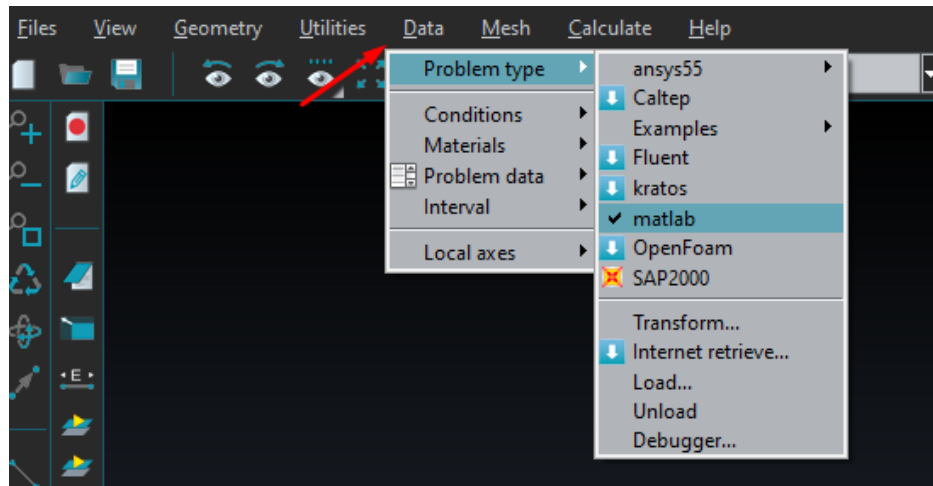
### 2.1.1.1. Geometry Setup

To create a geometry, select create object → rectangle. The rectangle can be drawn by clicking on the drawing plane or by specifying the coordinates of its corner edges (Fig. 1). The coordinates must be typed in the format x y z. They must contain white space between each coordinate, whereas omitting coordinates are assumed by default to be zero. Enter the first point (0, 0) in the command line and confirm with esc. Now enter the second point (0.5, 0.25).
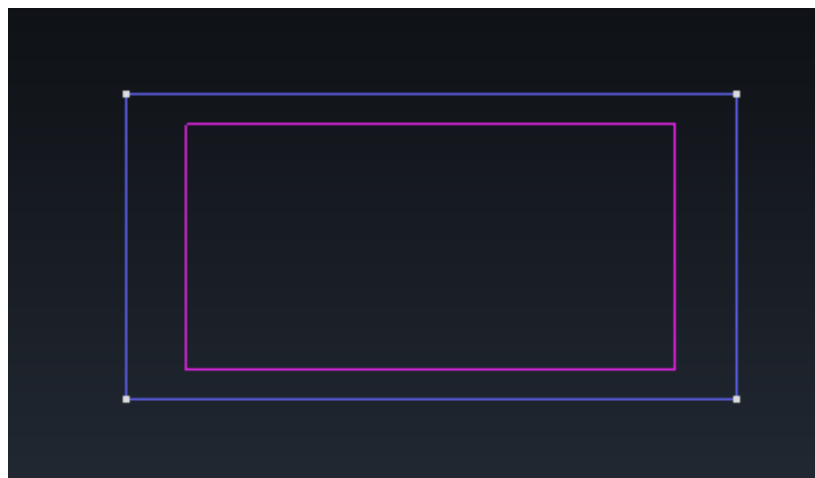
2.1.1.2 Boundary Conditions

To specify the Dirichlet boundary conditions select Data → Conditions → Constraints. Select lines (line icon) as selection type and select Structure-Dirchlet.
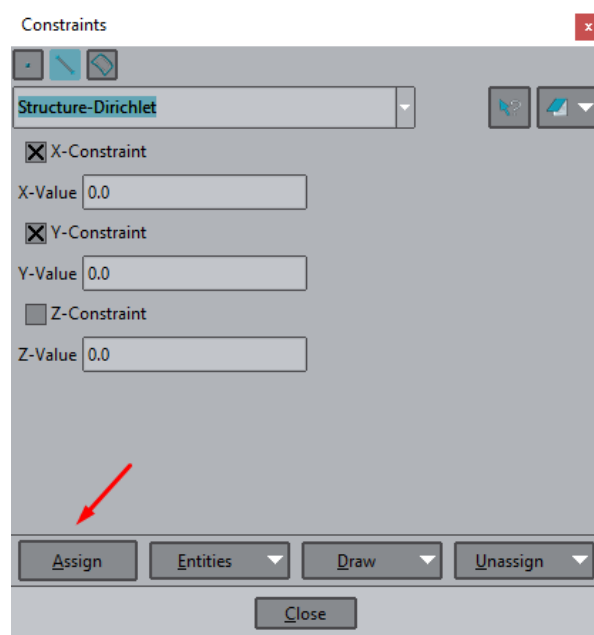

Then click on the Assign button and select the appropriate side to apply the boundary condition. Here, both translations on x and y are so we choose the value 0.

To assign the constant horizontal load select Data -> Conditions -> Loads

2.1.1.3. Definition of the computational domain

The computational mesh needs then to be assigned to a domain, so that the nodes and the elements for the chosen domain are written out to the desirable input file. Select Data → Conditions → Domains, choose Structure-Nodes from the drop-down menu and select the whole surface. Confirm with esc. Then, select Structure-Elements and repeat the previous step to assign the elements to the computational domain.
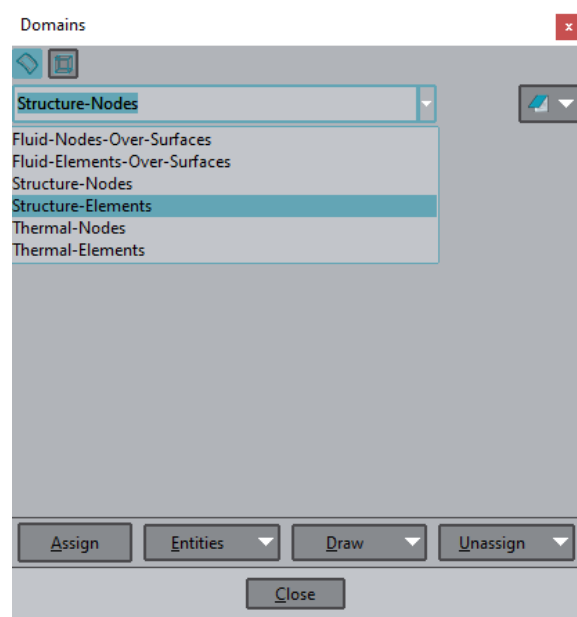
2.1.1.4 Selection of material properties

In order to select the material properties, select Data → Materials → Solids. There one can select the default material Steel from the drop-down menu or just change the given parameters to adjust material properties. Apply the material to the geometry of the problem by selecting Assign -> Surfaces and choose the surface of defining the problem's domain. Save the changes if asked so. The user may also expand the materials selection by editing the corresponding files under the folder matlab.gid.
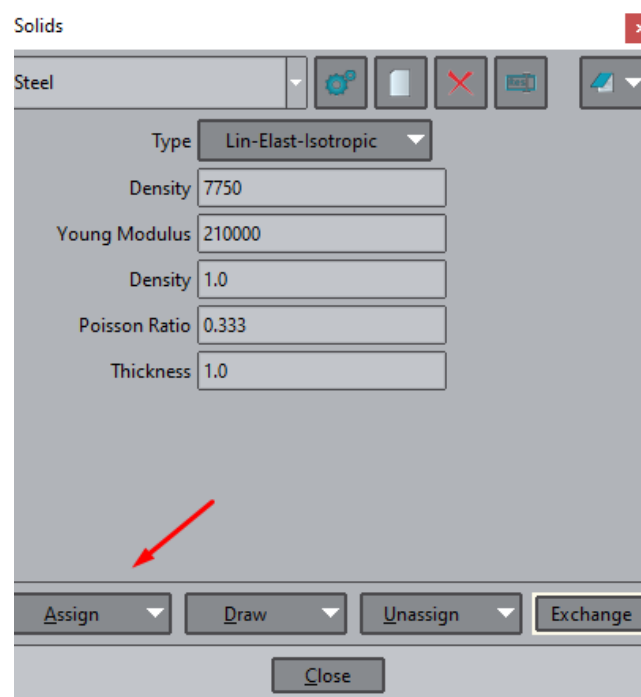
2.1.1.5 Generation of the computational mesh

Lastly, the finite element mesh needs to be generated. The simplest way is to go to Mesh →
Generate Mesh and specify the element size.
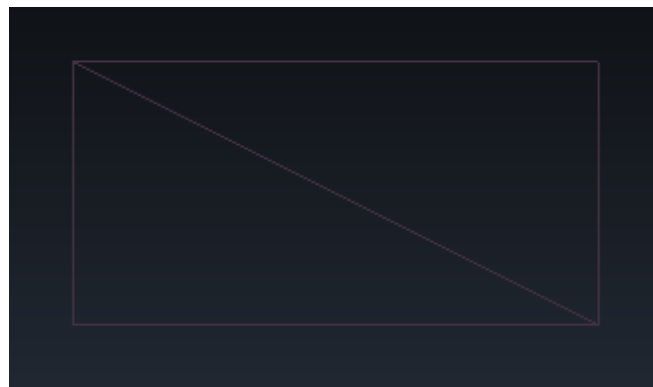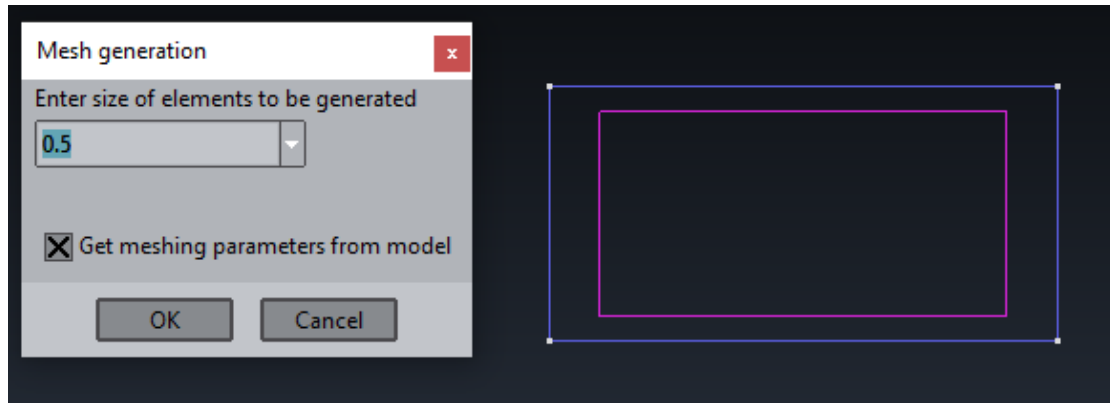
2.1.1.6 Selection of the analysis type and solver setup

To select the analysis type and setup the solver, select Data → Problem Data → Structural Analysis. In this window the user can select plane stress or plain strain analysis type and choose between a steady state or a transient analysis. Many more settings are possible, specifically on the time integration schemes and Gauss integration, but they are not needed for this simple case in which the default options are sufficients.
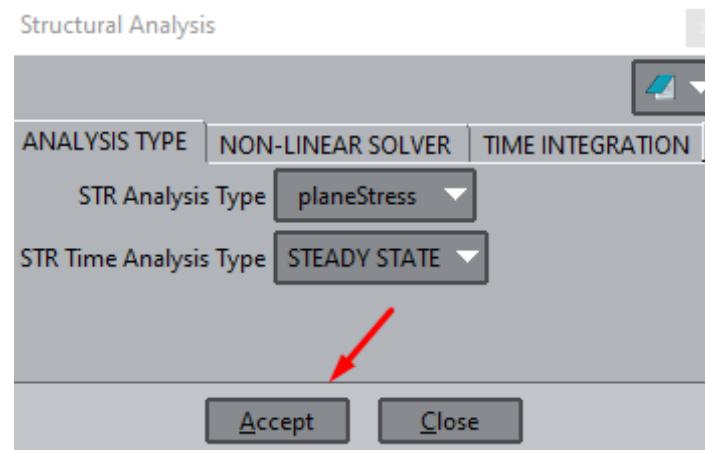
2.1.1.7 Generation of input file for analysis in MATLAB

After the setup is complete, select Calculation → Calculation (F5) or just select F5 to write out the input file which will be later on parsed within cane MATLAB framework. This file has the same name as our project whereas its extension is .dat. The user can also open it with any text editor to check or adjust the data. This file needs then to be placed under folder ./cane/inputGiD/FEMPlateInMembraneActionAnalysis and then a new caseName with the same name needs to be defined in the MATLAB main driver script.

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%
%    Structural Boundary Value Problem
%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

STRUCTURE_ANALYSIS
 ANALYSIS_TYPE,planeStress

STRUCTURE_MATERIAL_PROPERTIES
 DENSITY,7750
 YOUNGS_MODULUS,210000000
 POISSON_RATIO,0.3
 THICKNESS,0.025

STRUCTURE_NLINEAR_SCHEME
 NLINEAR_SCHEME,NEWTON_RAPHSON
 NO_LOAD_STEPS,1
 TOLERANCE,1e-9
 MAX_ITERATIONS,100

STRUCTURE_TRANSIENT_ANALYSIS
 SOLVER STEADY_STATE
 TIME_INTEGRATION EXPLICIT_EULER
 ALPHA_BETA -0.1
 GAMMA 0.6
 START_TIME 0
 END_TIME 10
 NUMBER_OF_TIME_STEPS 100
 ADAPTIVE_TIME_STEPPING true

STRUCTURE_INTEGRATION
 DOMAIN default
 domainNoGP 1
 boundaryNoGP 1
```

2.1.2. Solving with cane Multiphysics

2.1.3. Post-Processing with Paraview

2.2. Modal Analysis of a multihole plate

2.2.1 Preprocessing with GID

2.2.2. Solving with cane Multiphysics

2.2.3. Post-Processing with Paraview

2.3. Transient Analysis of a multihole plate

2.3.1 Preprocessing with GID

2.3.2. Solving with cane Multiphysics

2.3.3. Post-Processing with Paraview