

CALFEM Geometry Editor

– Implementing an interactive geometry editor for CALFEM

Examensarbetare: Karl Eriksson

In teaching of the Finite Element Method, a key component is the generation of a mesh which often is defined by a geometry. The CALFEM toolbox did not provide a user interface enabling quick and easy creation and editing of geometries. To address this problem, the interactive editor CALFEM Geometry Editor was developed.

The Finite Element Method (FEM) is a commonly used method to solve differential equations in various areas, such as structural analysis and heat flow. Differential equations are notorious for being very difficult to solve analytically, even for simple applications. FEM is based on the idea that the targeted problem, represented by a geometry, is divided into much smaller parts, so-called finite elements. Each element is built up by nodes and edges, the commonly used triangular element then has three nodes and three edges. These small elements form a mesh, where adjacent elements share the nodes and edges with each other. Instead of solving the global problem using the whole geometry, the problem is broken down to local calculations for each element. By utilizing this, much simpler and efficient calculations may be performed for the smaller elements. Finally, the elements are assembled back together to model the geometry. By using this method, even though very many calculations must be performed, the calculations are very fast and therefore the time and complexity to solve the problems may be drastically reduced.

FEM calculations are usually made using commercially developed software. These software are often, although powerful, difficult and time consuming to learn to use. To address this Structural Mechanics and Solid Mechanics at Lund University developed their own library CALFEM (Computer Aided Learning of the Finite Element Method), implemented in MATLAB and Python. Using this library, the students are provided with a clear connection between the theory and the numerical implementation of FEM application in Python and CALFEM. However, in the Python version, the only way of creating geometries has been through manual input, where every node, edge

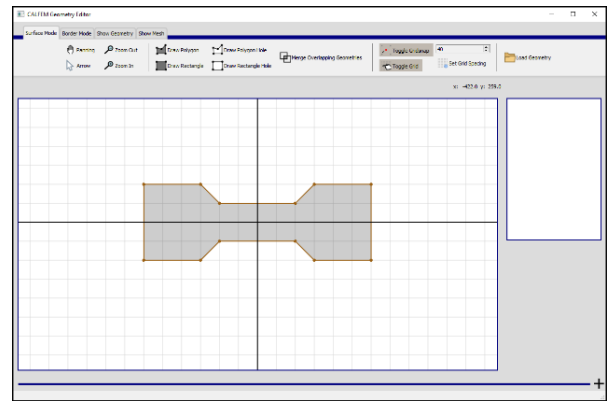


Figure 1: Snapshot of the finished interactive geometry editor

and surface is added by a line of code. This limited the library to simple geometries. Creating more complex geometries is time consuming and error prone.

In this project a program was developed to allow users to create simple geometries. The geometries may then be used together with the CALFEM library to solve FEM exercises. The program was implemented in Python using the library PyQt5 to create a Graphical User Interface. The application used the geometry and canvas classes in PyQt5 for drawing and editing the geometry in the editor. When the user is satisfied, the program converts the geometry description to the one used in CALFEM. In this way the user may draw a geometry using only a few clicks instead of manually defining it in several lines of code. The main functionality of the editor is developed towards use in Python, but option has been included to export mesh results to MATLAB as well.

Examensarbete avslutat 2021: Karl Eriksson: *CALFEM Geometry Editor* – -, Rapport TVSM-5255.

Handledare: Jonas Lindemann, Karin Forsman