XXXVI Ibero-Latin American
Congress on Computational
Methods in Engineering
CILAMCE
Rio de Janeiro, 22-25 Nov
2015

FINITE ELEMENT METHOD IMPLEMENTATION FOR SOLVING BOUNDARY VALUE PROBLEMS IN 2D

Nasser Alkmim

Lineu José Pedroso

nasser.alkmim@gmail.com

lineu@unb.br

Universidade de Brasília, Departamento de Engenharia Civil e Ambiental - Grupo de Dinâmica e Fluido-Estrutura (GDFE). Caixa Postal 04493, Campus Darcy Ribeiro, CEP 70919-970, Brasilia-DF

Asa Norte, Brasilia, 70910-900, DF, Brazil

Abstract. *This work presents a simple computer implementation of the Finite Element Method for solving boundary value problems in two dimensions, more specifically the heat transfer and plane elasticity problems. The code was made in Python and the following libraries were used: numpy for handling matrices operations; scipy for solving the system of linear equations; matplotlib for plotting the contour maps with the solution distribution within the domain and networkx for drawing the domain and boundary condition. The mesh was created using "Gmsh", Geuzaine & Remacle (2009), which is a free mesh generator, and then parsed inside the python code.*

Keywords: *Finite Element Method, Plane Elasticity, Heat Transfer, Python*

1 INTRODUCTION

The Finite Elements Method is a widely used discretization technique in structural mechanics and also in non structural problems. Its popularity on engineering analysis started in the mid of the XX century together with the advancements on computation technology.

This work aims to show the implementation aspects of the method using the Python programming language which is known for its simple syntax that provides readability to the code. Its property of Object-Oriented programming allows a clean and intuitive script for the code, e.g., passing all the mesh attributes and methods as a function argument is as simple as passing the object “mesh”. The large amount of libraries available represents another advantage.

2 PROBLEM FORMULATION

This section presents the mathematical model for the plane elasticity and heat transfer problems. Both topics were widely discussed on the literature such as Zienkiewicz (2005) and Fish & Belytschko (2007).

2.1 Elasticity

The elasticity problem can be stated as “Given the body geometry, applied loads, support conditions, material stress-strain and initial stress; Calculate the body displacements and the corresponding strains and stresses” Bathe (1996). Its mathematical model consists of three equations; the first one is the equilibrium equations, the second, kinematics; and the third the constitutive relations.

$$\begin{aligned} &\sigma_{ij,j} + b_i = 0 \\ &\varepsilon_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \qquad \text{with} \qquad \begin{aligned} u_i &= u_i^d \quad \text{on} \quad \partial\Omega^d \\ f_i &= n_j \sigma_{ij}^n \quad \text{on} \quad \partial\Omega^n \end{aligned} \\ &\sigma_{ij} = \mathbb{C}_{ijk\ell}\, \varepsilon_{k\ell} \end{aligned} \tag{1}$$

where σ_{ij}, ε_{ij} and u_i are the stress, strain and displacement fields, respectively. $\mathbb{C}_{ijk\ell}$ is the elasticity tensor. The distributed body forces per unit volume is represented by b_i. For this model small displacements assumptions were made. The boundary of the domain is $\partial\Omega = \partial\Omega^d \cup \partial\Omega^n$ and n_j the unit vector normal to the domain.

2.2 Heat transfer

The heat transfer problem in its transient form is modeled by,

$$k\theta_{,ii} + q = c\rho\frac{\partial\theta}{\partial t} \qquad \text{with} \qquad \begin{aligned} \theta &= \theta^d \quad \text{on} \quad \partial\Omega^d \\ k\theta_{,i} &= s^n \quad \text{on} \quad \partial\Omega^n \end{aligned} \tag{2}$$

which represents the equilibrium of heat flow. In the equation, k is the the thermal conductivity, c is the specific heat of the material and ρ the density. The temperature field is represented by θ and q the internal energy source. The boundary conditions of the problem are imposed on the domain boundary, $\partial\Omega$, and the initial condition is given by $\theta(x_i, 0) = \theta_0$.

3 COMPUTER IMPLEMENTATION

The computer implementation for both problems, scalar and vector fields, has the same architecture. The framework used was the Direct Stiffness Method, where all elemental matrices were computed for later be assembled in the global system of equations. The first part of this process is depicted in Fig 1.

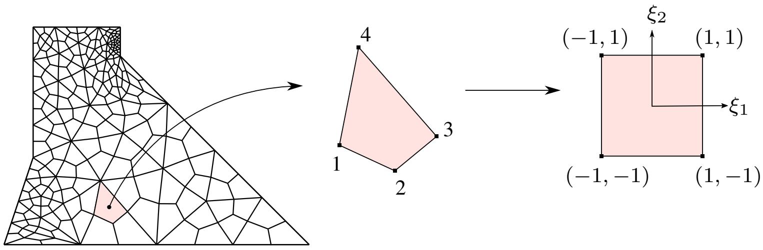

Figure 1: Discretization process and element coordinate transformation for integral computations. ξ_1 and ξ_2 are the coordinates of the isoparametric domain.

In order to use the discretization we first write the equations in the weak form,

$$
\begin{aligned}
&\int_\Omega v_{,i} k \theta_{,i}\, \mathrm{d}\Omega - \int_\Omega v q\, \mathrm{d}\Omega - \int_{\partial\Omega} v s^n\, \mathrm{d}\Gamma = 0 \\
&\int_\Omega w_{i,j} \sigma_{ij}\, \mathrm{d}\Omega - \int_\Omega w_i b_i\, \mathrm{d}\Omega - \int_{\partial\Omega} w_i f_i^n\, \mathrm{d}\Gamma = 0
\end{aligned}
\tag{3}
$$

with, v and w_i been the test function and the vector with test functions. The Neumann boundary conditions are: s^n the heat flux assigned to a delimited boundary and f_i^n the assigned traction.

4 VALIDATION

In order to validate the solver created for the plane elasticity problem two problems are proposed: the first one deals with a rectangular plate under gravity load only; the second, a structure with a minimum set of displacement boundary conditions and with traction applied on both sides, this problem is also reproduced on Mosalam (2011). Figure 2 shows the result for both problems.

In the test with only distributed body forces, 126 elements, the error for the displacement at the bottom end was less than 0.1% and for the normal stress at the top end less than 2% (the stresses were obtained directly on element nodes instead of Gauss points), both compared with analytic solution. The second test resulted in the exact solution for displacement field and stresses.

The heat transfer problem was validated with a simple problem: a square plane with a fix temperature assigned to one of its side and a fix heat flux assigned to the opposite one. The result was exact with the analytic solution.

5 CASE STUDY

The case study used to demonstrate the program capabilities is a dam, shown in profile on Fig.3,on the left, discretized by 2913 quad elements and under the load of the water reservoir

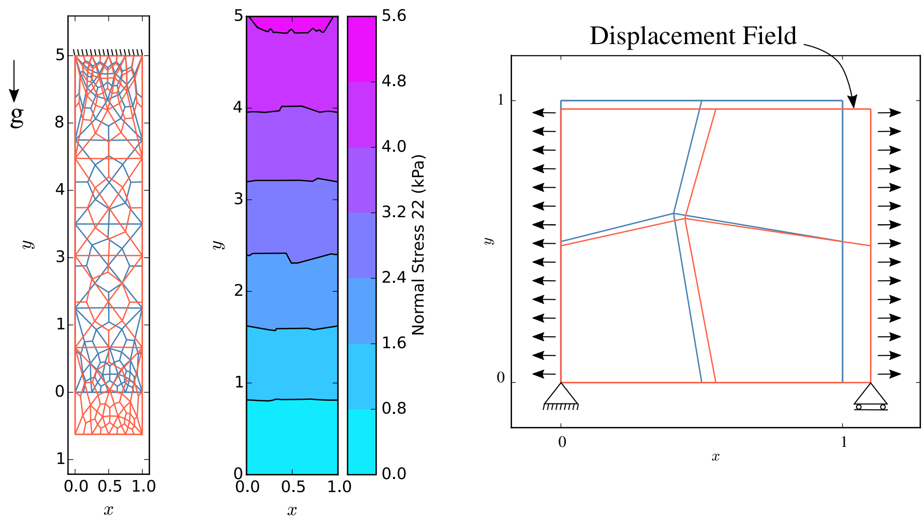

Figure 2: Validation problems for the plane elasticity solver. On the left, the deformed shape of plate under gravity (g) load and normal stress contour plot. On the right, the deformed shape of the structure from the second test.

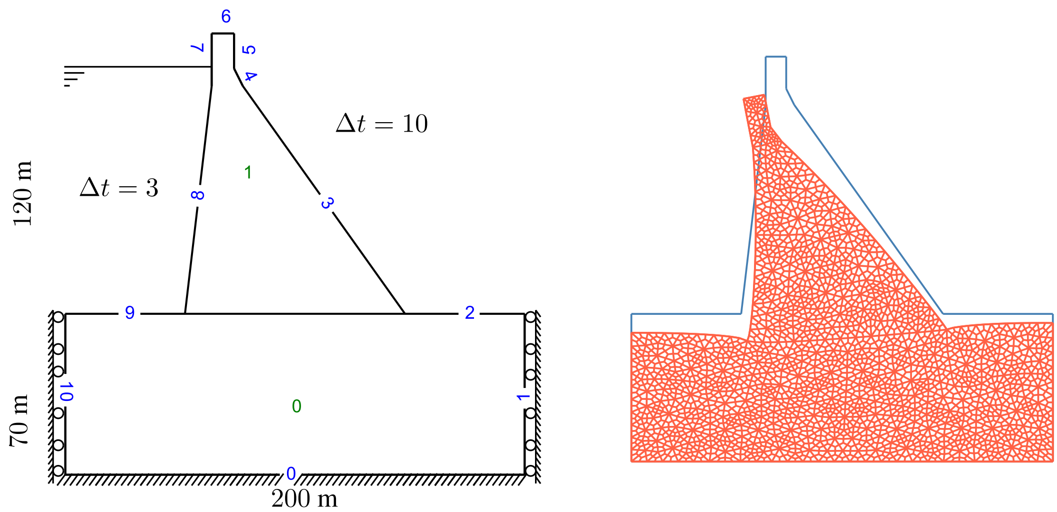

Figure 3: On (left) the dam geometry with the edge labels used to specify boundary conditions and on (right) the displacement field of the structure amplified 1000 times. Δt represents the temperature change.

and the thermal load due a temperature change during a typical day. The foundation Young's modulus is different from the actual dam.

Figure 4 shows the principal stress distribution obtained from averaging the resulted node value weighted by the number of elements sharing that node. From the plot we can see where are the stress concentration regions, which is the heel of the dam.

6 CONCLUSION

This work has shown the versatility of the Finite Element Method with the Direct Stiffness Method framework for solving boundary value problems with complex domain and with material discontinues, such as a dam profile with rock foundation and concrete structure. This

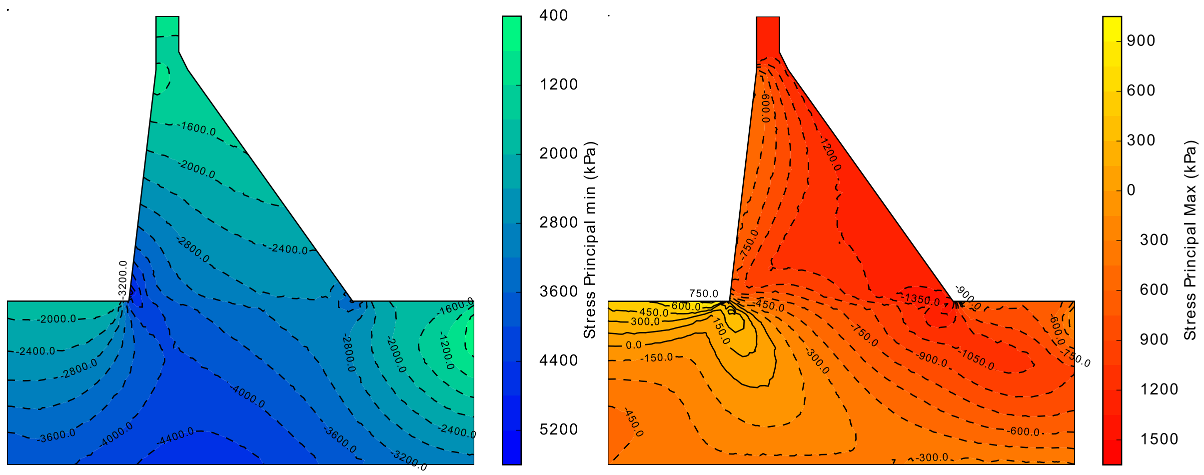

Figure 4: On the (left) the minimum principal stress distribution and on (right) the maximum principal stress distribution.

feature arises from the implementation architecture, which deals with each element individually. The integral over each element sub-domain can be further simplified for computer implementation with the isoparametric formulation where element nodes are mapped to a master standard element.

The produced code was validated using two plane elasticity and one simple heat transfer examples (the later not shown in this paper), all of them with good approximations.

The Python language proved useful, especially the plotter library. The mesh was plotted using an other useful library called NetworkX which handled elements as networks. The boundary conditions were passed to the solver as a functions of space which can be conveniently done since python treats everything as an object.

REFERENCES

Bathe, Klaus-Jrgen. 1996. *Finite element procedures*. Englewood Cliffs, N.J. Prentice Hall.

Fish, Jacob, & Belytschko, Ted. 2007. *A First Course in Finite Elements*. John Wiley & Sons.

Geuzaine, Christophe, & Remacle, Jean-Franois. 2009. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities.

Mosalam, Khalid M. 2011. *CE222 Finite Element Methods course reader*.

Zienkiewicz, O. C. 2005. *The Finite Element Method: Its Basis and Fundamentals - Its Basis and Fundamentals*. 6. aufl. edn. Oxford: Butterworth-Heinemann.