

Programming for Engineers and Scientists

Part 1: FE program design

Domingo Eugenio Cattoni Correa

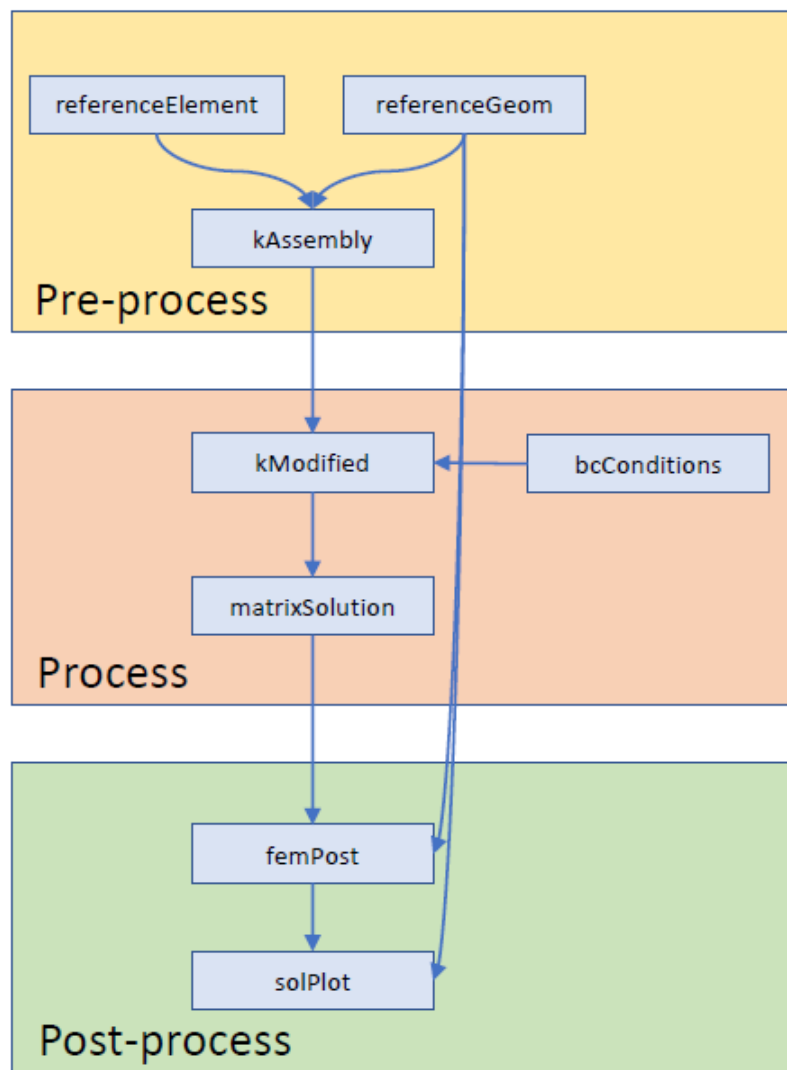
Oriol Call Piñol

Juan Diego Iberico Leonardo

For the first part of the assignment 1, in the present document is developed the main structure of the program. It is based in the decomposition of the necessary *functions* as well as *structures* where the variables are saved by packages.

The main scheme of the program in terms of functions is the following:

Functions



In the flow chart the structure were divided in the three main processes for a Fe program.

- Pre-Process: The functiones defined in the frist part are mainly fouced to input data definning the geometry. On the other side, the type of reference element i choosen by the user, selecting from the ones that are implemented on the code (linear, trianluar, quadrilater and the order).
- Process: Once the assembled matrid is defined, the functions related with this step, evaluated the boundary (bcConditions) conditions and modify the master stiffness matrix (kModified). Finally the linear system of rquations are solved calling the function matrixSolution.
- Post-Process: Once the function of matrixSolution is computing, the main values of interest are computed nd evaluated on the functions defiened as post-process. Furthermore, a plot is showed presenting the defomring structure.

Next it is presented the differend types of structures used to arrange the information according the insterest of the variable or input. This scheme is devided in three parts according the proeces where it is taking part.

MATLAB Structures

```
mesh.connectivity  
mesh.nCoordinates  
elements.shapefun  
elements.xShapefun  
elelemnts.xxShapefun
```

```
bc.neumann  
bc.Dirichlet  
  
modified.kGlobal  
modified.force
```

```
result.displament  
result.stress  
result.strain  
result.force
```

referenceGeom

Input: -

Output: mesh.nCoordinate, mesh.connectivity.

Description: Read txt files with information of connectivity matrix and nodes coordinates.

Uses: (txt files)

Used by: kAssembly, Plot

Comments: Introduce the geometry problem to Matlab. That can be done from txt files generated from GID.

referenceElement

Input: dimension, order, element

Output: element.shapefun,
element.xShapefun, element.xxShapefun

Description: The user define what kind of element and order. The function gives the shape function and its derivatives (in natural coordinates).

Uses: -

Used by: kAssembly

Comments:

1. Line
2. Square
3. Triangle

kAssembly

Input: mesh.connectivity mesh.nCoordinates
element.shapefun, element.xshapefun,
element.xxshapefun

Output: kglobal

Description: Compute the elemental stiffness matrix and makes the mesh assembly

Uses: referenceGeom, referenceElement

Used by: kModified

Comments: at the end, we get kglobal

kModified

Input: kglobal, bc.neumann, bc.dirichlet

Output: modified.kGlobal and modified.force.

Description: Applies the Boundary Conditions to the matrix.

Uses: kAssembly, bcConditions

Used by: matrixSolution

Comments:

bcConditions

Input: -

Output: bc.neumann, bc.dirichlet.

Description: Read txt files with information of boundary conditions matrices.

Uses: (txt files)

Used by: kModified

Comments: Introduce the boundary conditions to Matlab. That can be done from txt files generated from GID.

femPost

Input: result.displacement

Output: result.stress, result.strain, result.force

Description: Compute different values of interest as a post-process step.

Uses: matrixSolution

Used by: solPlot

Comments: the user can define the different values of the output depending the problem.

solPlot

Input: result.displacement,
mesh.nCoordinate, mesh.connectivity

Output: plot

Description: Show figures of the deformed geometry

Uses: refGeometry, matrixSolution, femPost

Used by: -

Comments: furthermore, can show stress and strain under color figures.

For each function, the figure describe the main features as the input and output and comment about their interaction between them in the main scheme.