Webfem manual

by webfem team

NOTICE: User manual.

Manual for the user

1. Introduction

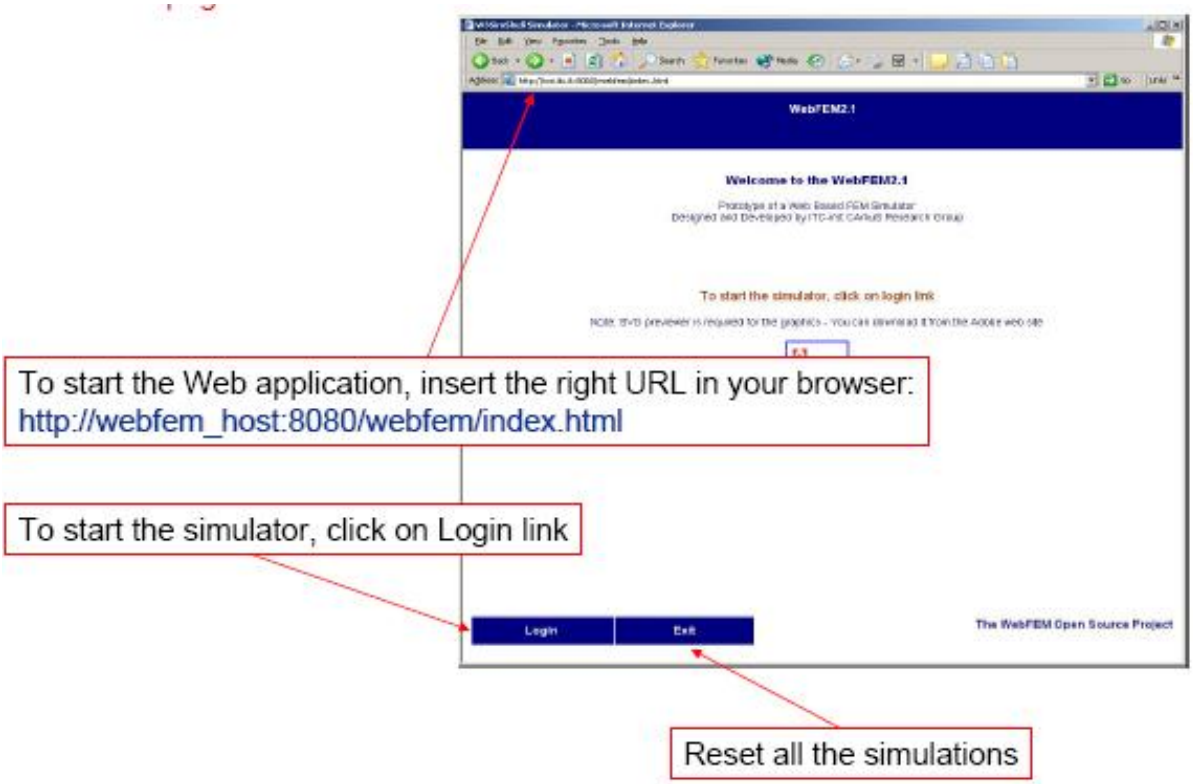
This is a simple manual.

FIXME (PC):

We describe only the use of the SVG interface.

2. Welcome page

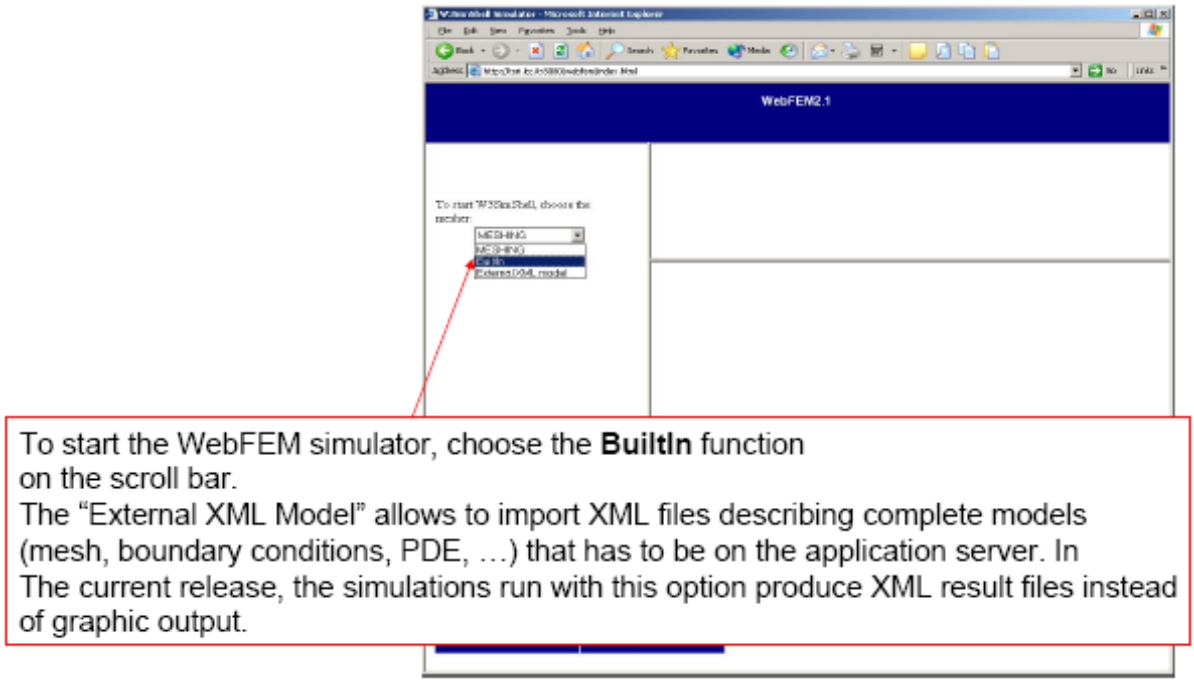
This is the first page of the simulator.



Welcome page

3. Mesher choiche

We can import a mesher file or use the simple web client to define the boundary condition.

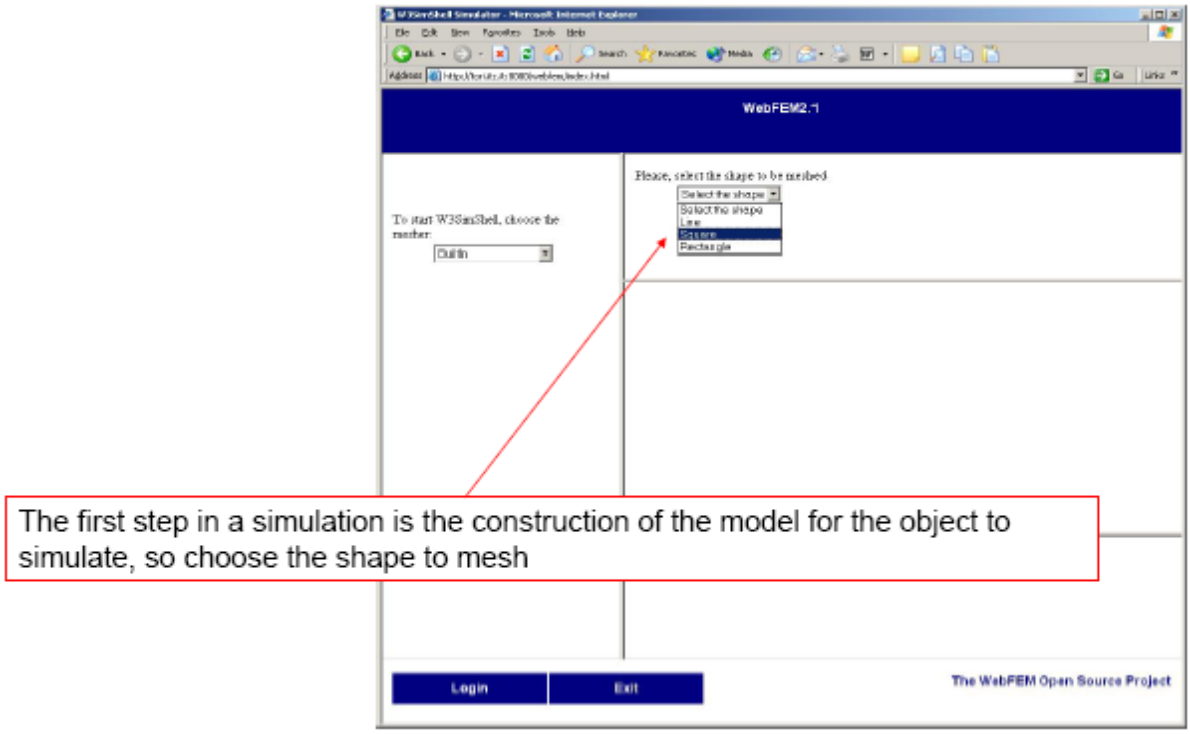


Mesher choiche

4. Shape configuration.

Select the shape to mesh.

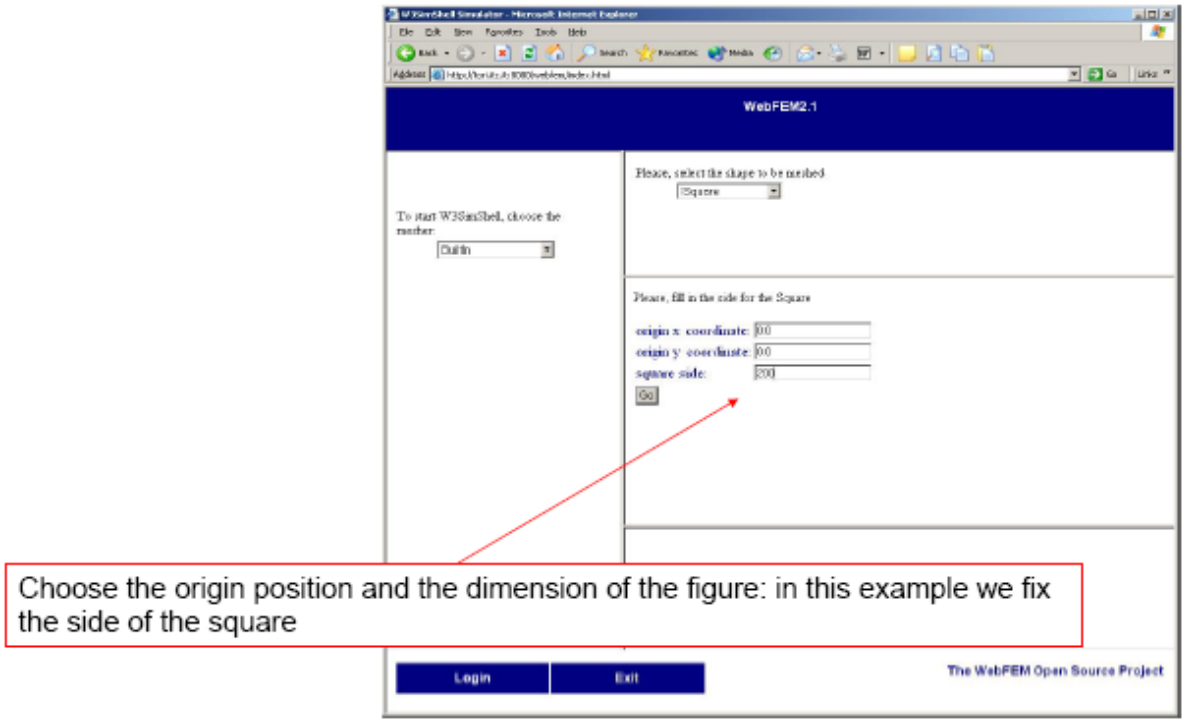
Note:
At the moment, 2D regular geometries are available



Shape config.

5. Shape configuration 2

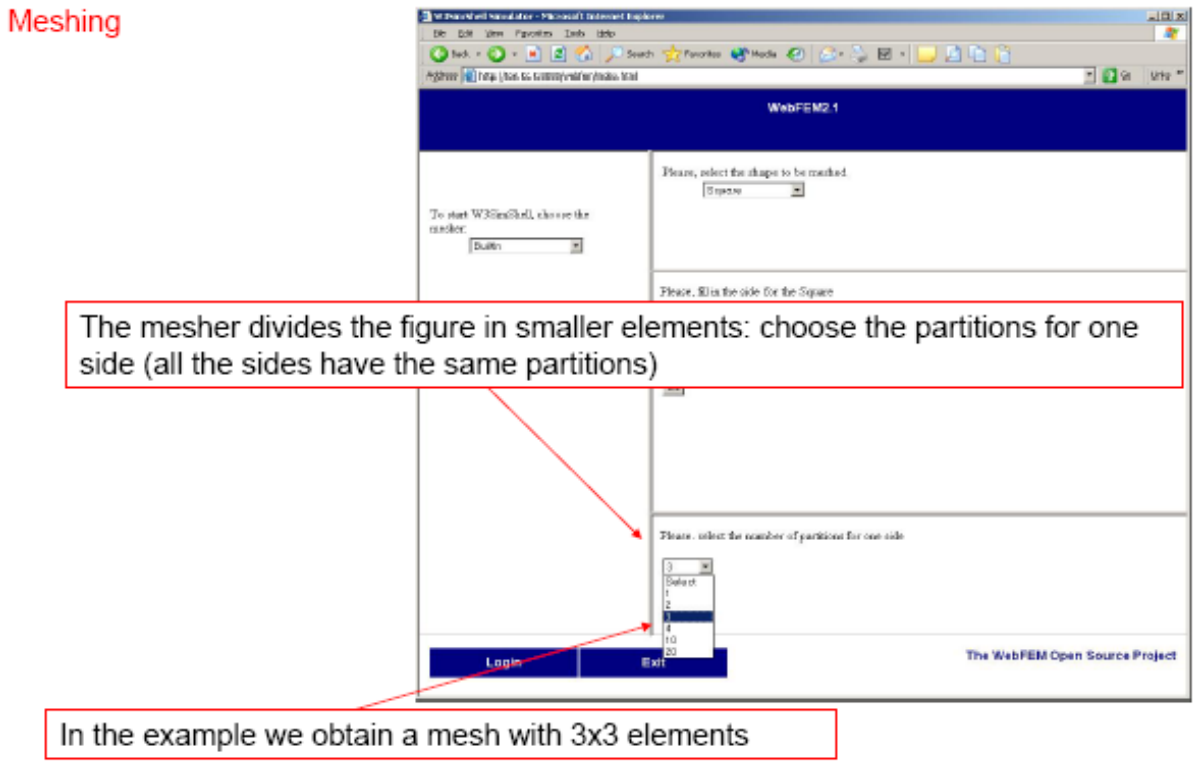
Define the geometry properties: dimensions and origin coordinates



Shape config. 2

6. Meshing

Define the meshing properties: number of partitions

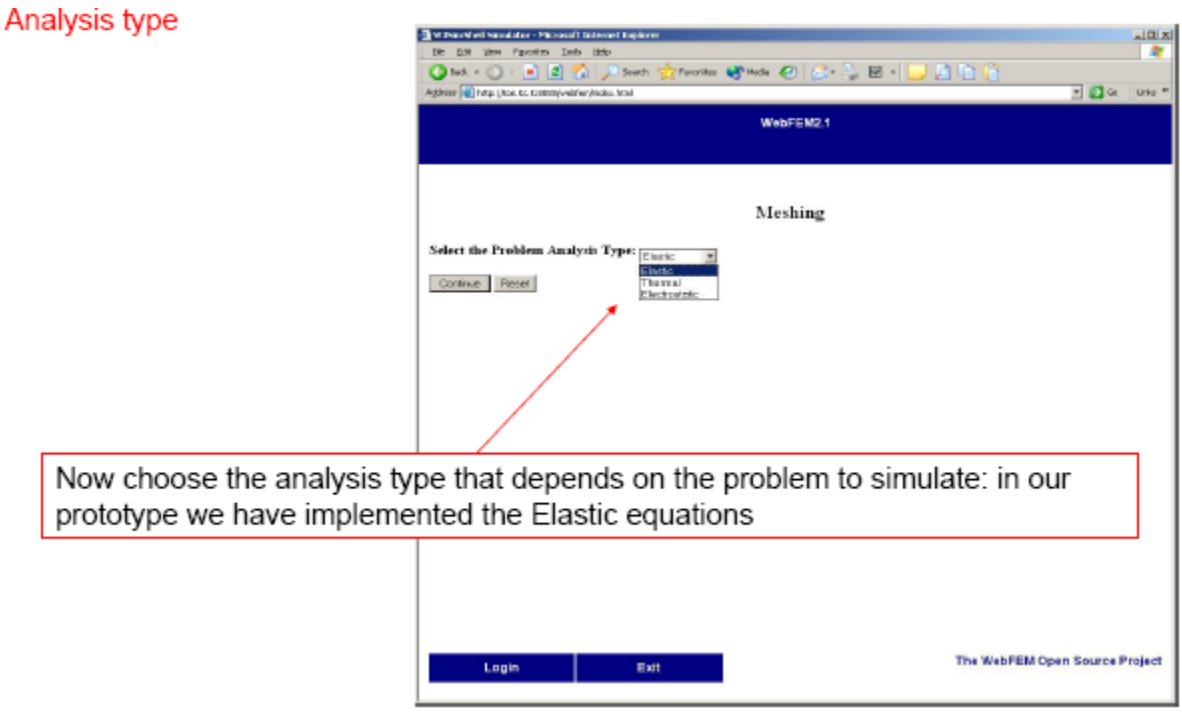


Meshing

7. Analysis type

Define the type of analysis.

Note:
At the moment Static Elastic Equation only.



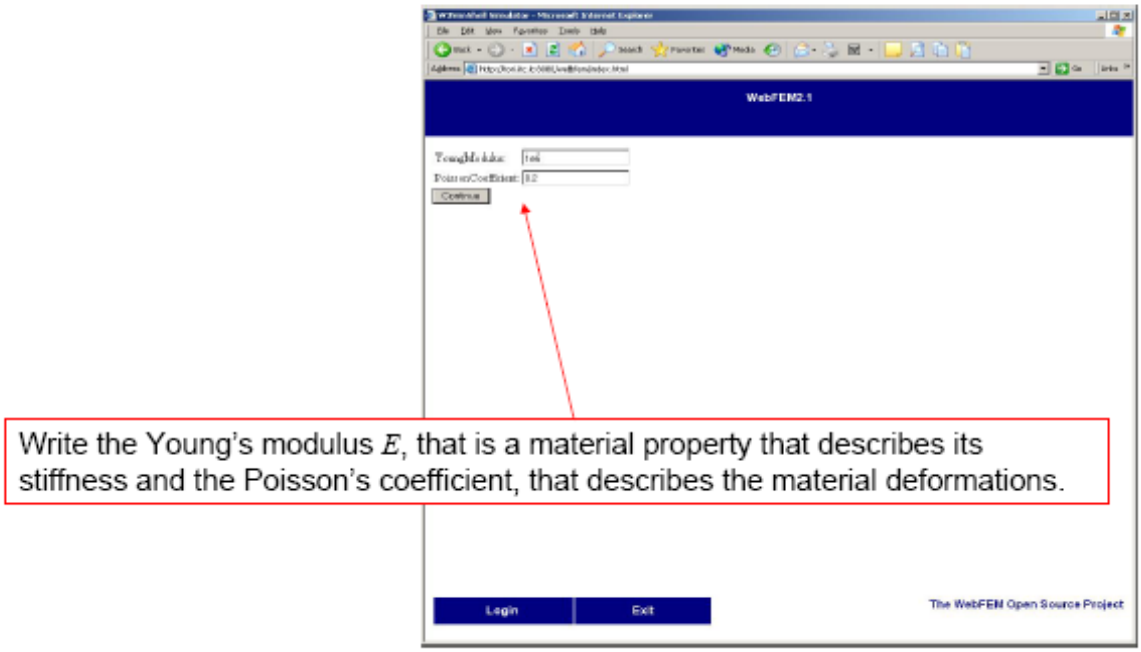
Analysis type

8. Material properties

Define material properties.

Note:
Poisson's has to be specified. It must be between 0 and 0.5.

Material properties



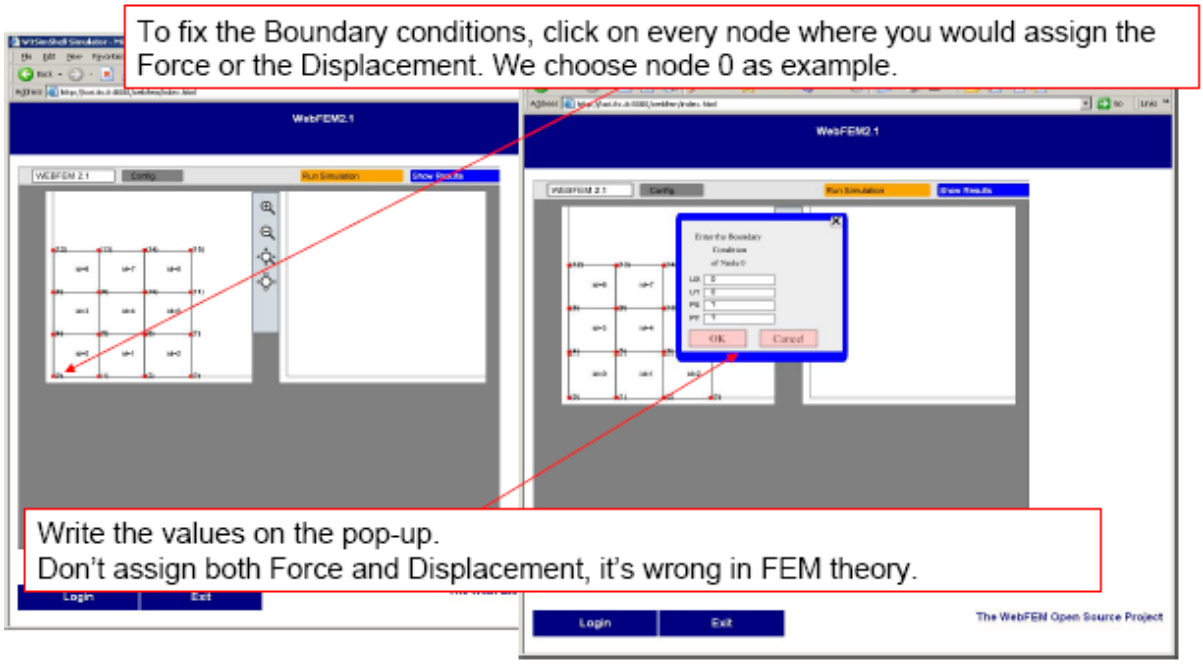
Analysis type

9. Boundary Condition

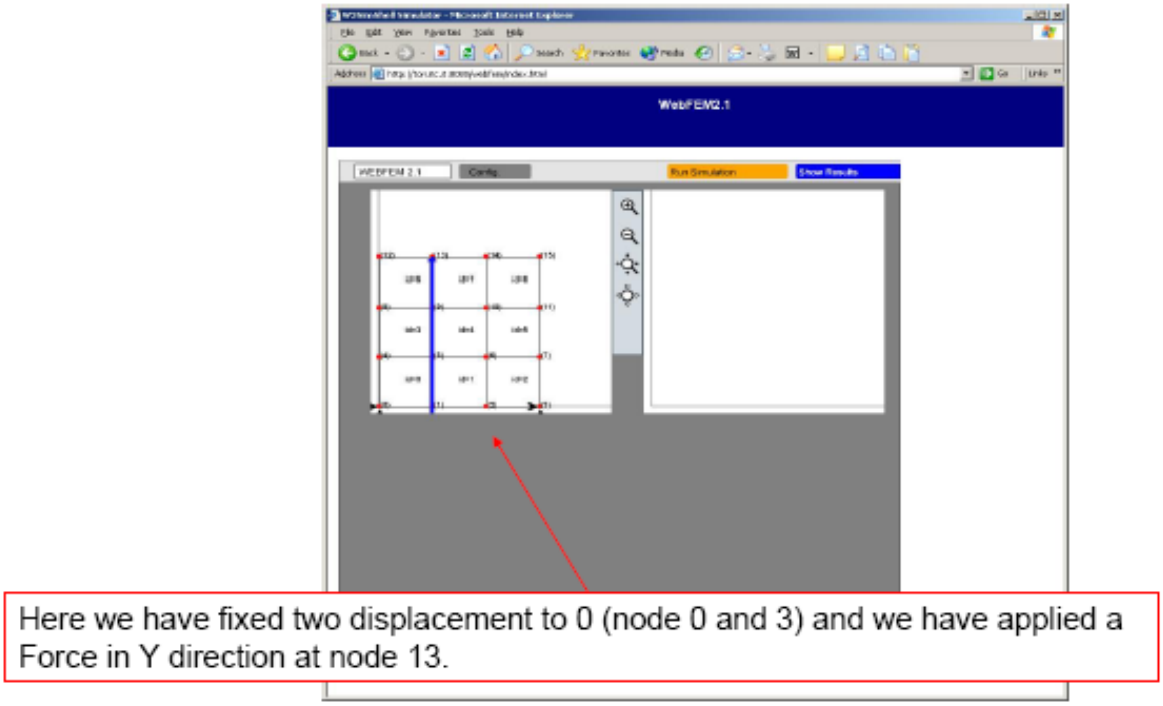
Define the boundary conditions of the problem.

Note:

Avoid assigning both Force and Displacement on the same node, it is redundant in FEM theory.



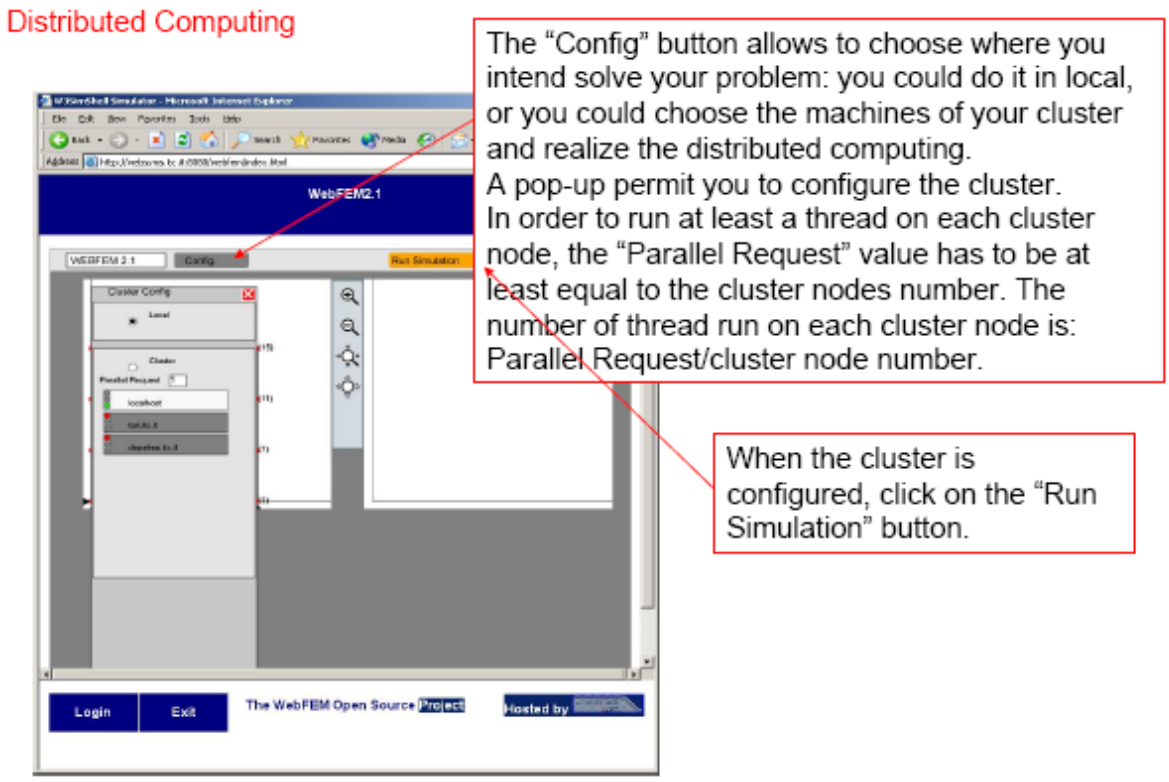
boundarycondition



boundarycondition

10. Configuration

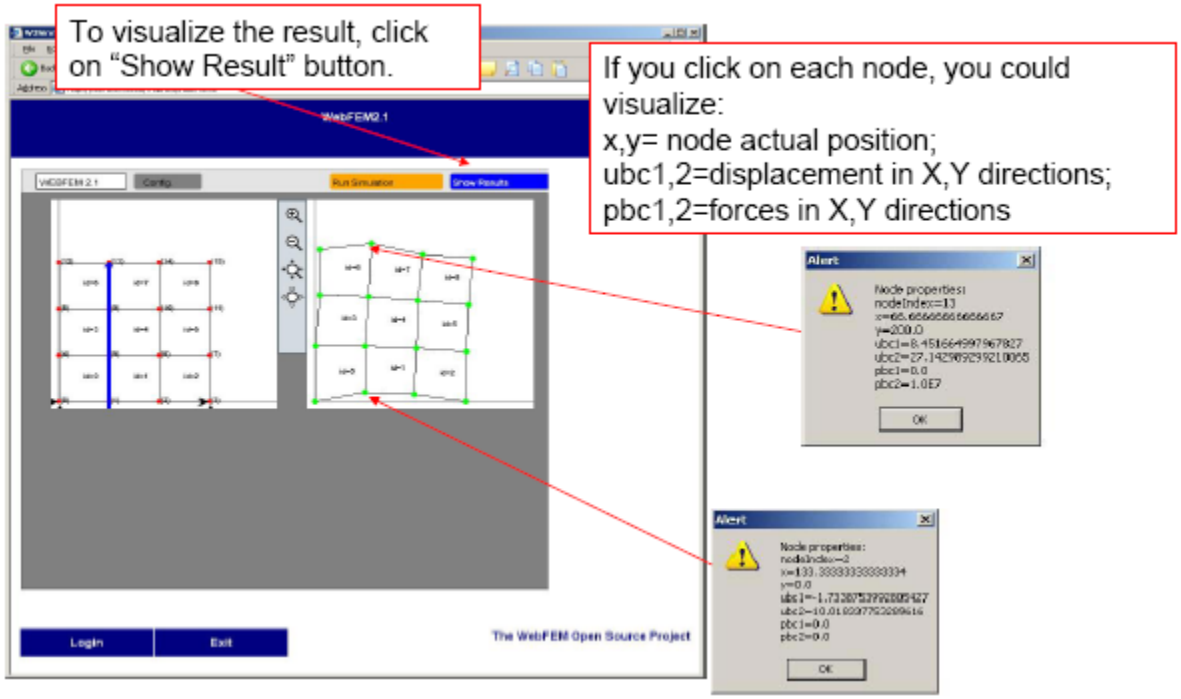
Define the configuration of the distribuited computing.



Config

11. Result

View the result in the SVG gui.



Result