Computational Structural Mechanics and Dynamics
Assignment 2

Federico Valencia Otálvaro

Master's in Numerical Methods in Engineering
Universitat Politècnica de Catalunya

February 26^{th}, 2020

Contents

List of Figures

1 Problem Description

1.1 Assignment 2.1

1. Identify the symmetry and antisymmetry lines in the two-dimensional problems illustrated in the figure. They are:

   (a) a circular disk under two diametrically opposite point forces (the famous "Brazilian test" for concrete)
   (b) the same disk under two diametrically opposite force pairs
   (c) a clamped semiannulus under a force pair oriented as shown
   (d) a stretched rectangular plate with a central circular hole
   (e) and (f) are half-planes under concentrated loads.

2. Having Identified those symmetry/antisymmetry lines, state whether it is possible to cut the complete structure to one half or one quarter before laying out a finite element mesh. Then draw a coarse FE mesh indicating, with rollers or fixed supports, which kind of displacement BCs you should specify on the symmetry or antisymmetry lines.

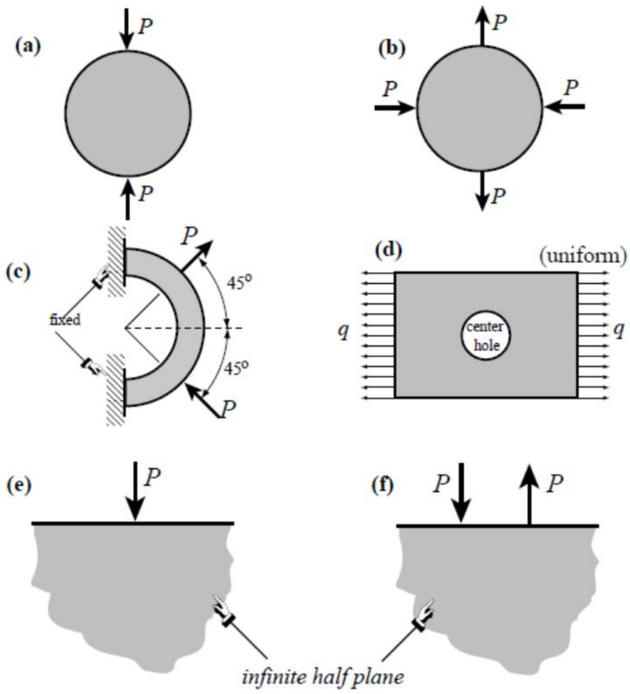


Figure 1: Problems for Assignment 2.1

1.2 Assignment 2.2

Explain the difference between "Verification" and "Validation" in the context of the FEM Modelling procedure.

1.3 Assignment 2.3

A tapered bar element of length l and areas A_i and A_j with A interpolated as:

$$A = A_i(1 - \xi) + A_j\xi \quad (1)$$

and constant density ρ rotates on a plane at uniform angular velocity ω (rad/sec) about node i. Taking axis x along the rotating bar with origin at node i, the centrifugal axial force is $q(x) = \rho A \omega^2 x$ along the length in which x is the longitudinal coordinate $x = x^e$.

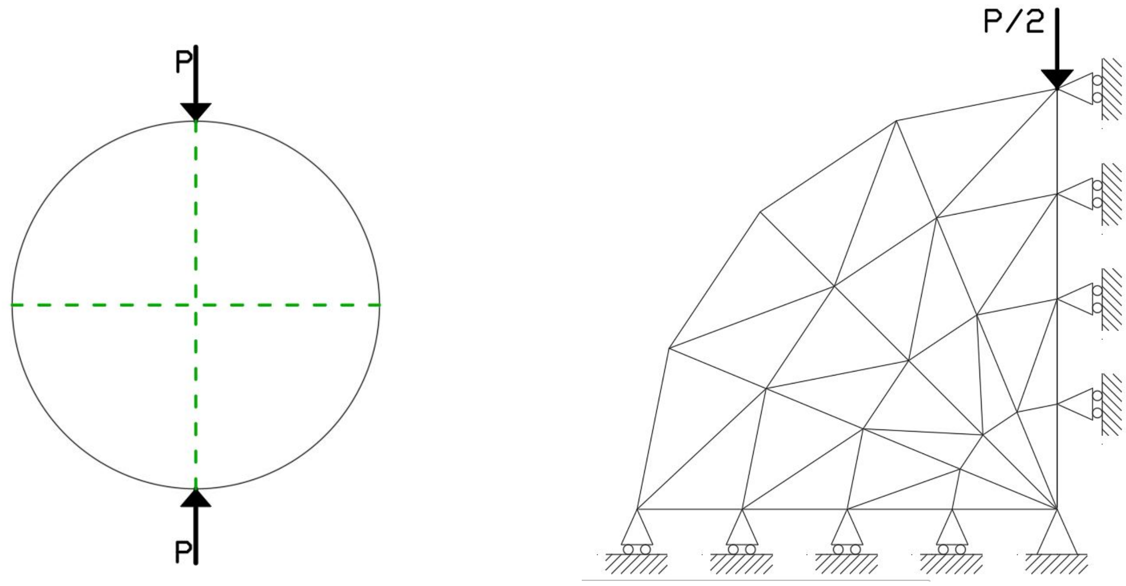
Find the consistent node forces as functions of ρ, A_i, A_j, ω and l, and specialize the result to the prismatic bar $A = A_i = A_j$.

2 Solution

2.1 Assignment 2.1

2.1.1 Problem A

Due to the symmetric configuration of both the structure and the loads, two orthogonal symmetry lines are present in this case. Hence, it is possible to consider only one quarter of the domain to perform a finite element analysis.



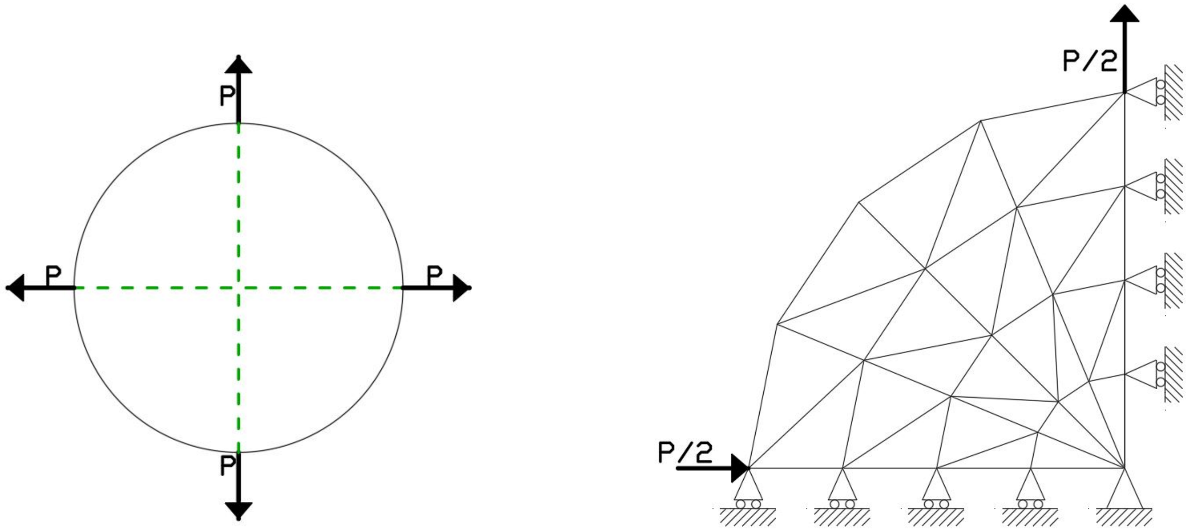
(a) Symmetry Axis (b) FE Discretization

Figure 2: Problem A

Since the two symmetry axis intersect at the center of the circle, the node located at the intersection must be restrained for both vertical and horizontal displacements. The nodes along the horizontal symmetry axis are restrained for vertical displacements and the ones along the vertical symmetry axis are restrained for horizontal displacements. Load P is reduced by one half due to the fact that it is placed exactly along the vertical axis of symmetry, and since the stiffness of the nodes along the axis is being reduced, the load must be reduced as well. The reduced domain was then discretized with a mesh of 28 triangular finite elements (FE).

2.1.2 Problem B

Problem B is very similar to problem A. The geometry of the structure is the same but the load configuration changes. However, the new loads are applied in such way that the symmetry axis proposed for problem A may be used in this case as well. Hence, the reduced domain and FE mesh created for problem A was also employed for problem B. Loads were reduced in the same manner as well.

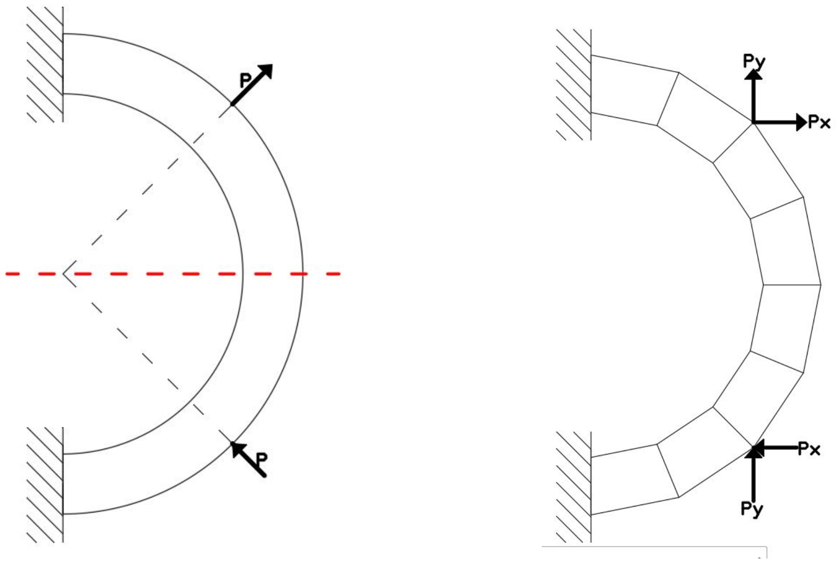


(a) Symmetry Axis (b) FE Discretization

Figure 3: Problem B

2.1.3 Problem C

Unlike problems A and B, problem C is asymmetric. In this case, the antisymmetry axis is a horizontal passing along the center of the semi-circular structure.



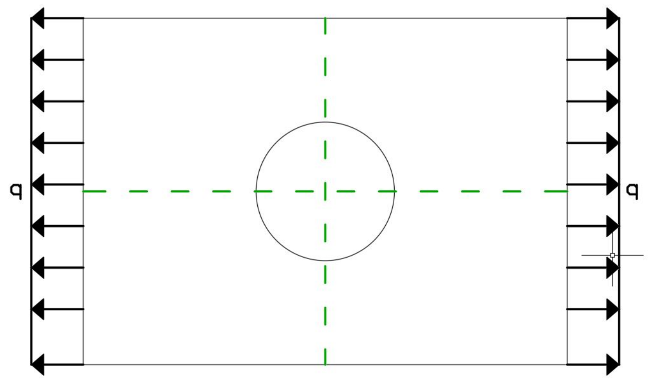
(a) Antisymmetry Axis (b) FE Discretization

Figure 4: Problem C

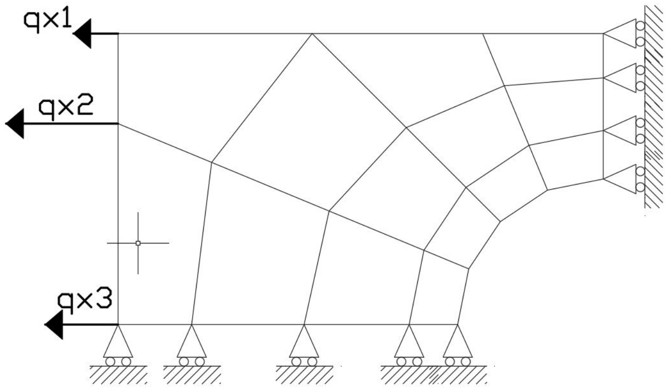
Since the loads are not parallel to the antisymmetry axis, the domain cannot be reduced. The complete domain was discretized with a mesh of 8 quadrilateral FE. All four nodes located at the ends of the structure must be fixed and the loads must be separated into vertical and horizontal components in order to apply them to vertical and horizontal degrees of freedom respectively.

2.1.4 Problem D

Problem D has two symmetry axis which divide the plate in halves. This allows is to reduce the domain to one quarter of the original one.



(a) Symmetry Axis



(b) FE Discretization

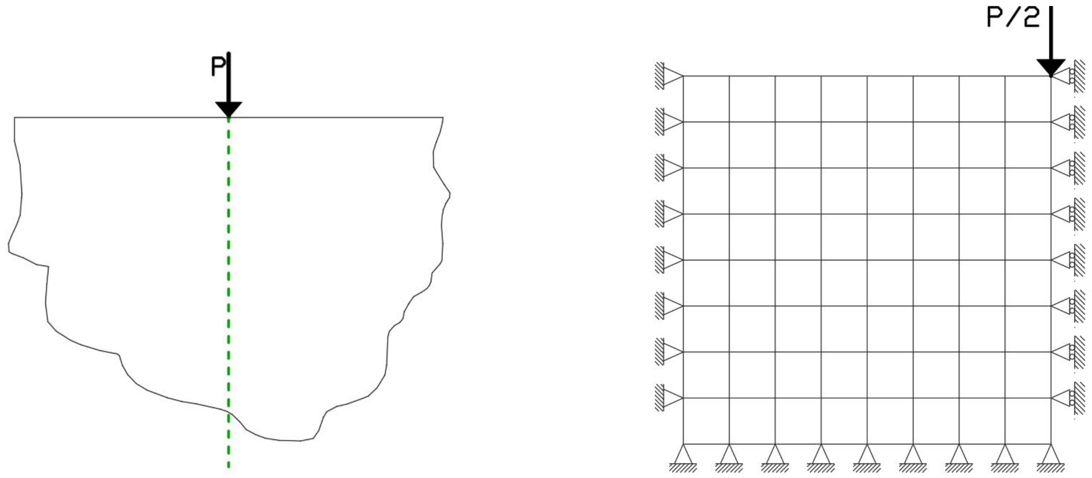
Figure 5: Problem D

The reduced domain was discretized with a mesh of 14 quadrilateral elements. The nodes along the horizontal symmetry axis are restrained for vertical displacements while the nodes along the vertical symmetry axis are restrained for horizontal displacements. The distributed load q must be assigned as horizontal nodal loads on the nodes along the left edge of the reduced domain. It is worth noting that since the elements along the left edge are not uniform, the loads on the nodes will vary between one another, since the afferent length for each one of them will be different.

2.1.5 Problem E

Problem E illustrates a single vertical load being applied at the upper surface of an infinite half plane. Since there is only one load present, there is a symmetry axis along the line of application of the load.

Since an infinite domain cannot be analyzed with the finite element method, the reduced domain had to be given boundaries for its discretization. The domain was reduced by half due to the existing symmetry axis and the nodes along the vertical symmetry line were restrained for horizontal displacements. The finite elements domain has to be large enough for the influence of load P to be negligible in the boundary nodes where the plane is supposed to be infinite. The nodes at these boundaries are restrained for both vertical and horizontal displacements. load P was reduced to half for the same reason as in problem A.



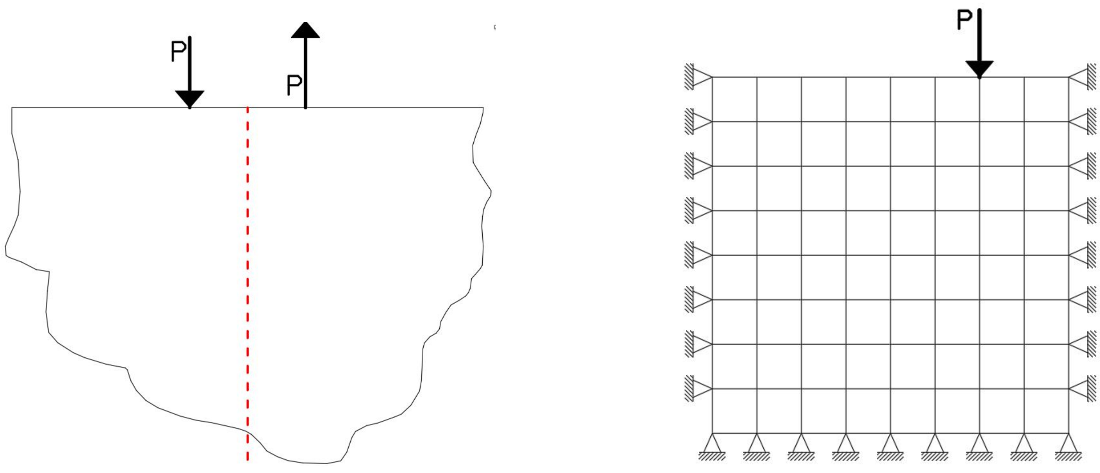
(a) Symmetry Axis (b) FE Discretization

Figure 6: Problem E

2.1.6 Problem F

Problem F presents an antisymmetric problem where there is one vertical antisymmetry axis which crosses the middle point between the two loads P.

Despite the asymmetry of the problem, the domain may still be reduced to a half thanks to the fact that the loads are parallel to the antisymmetry axis. The displacements induced to the structure by the loads will have the same magnitude but opposite directions on both sides of the antisymmetry axis, making the displacements along the axis zero. This effect is represented by restraining horizontal and vertical displacements on the nodes along the antisymmetry line. In the same way as in problem E, the infinite plane was represented by creating a large enough finite elements mesh for the influence of the load to be zero on the nodes at the boundaries where the domain is supposed to be infinite. These nodes are restrained for vertical and horizontal displacements. The left side of the domain was chosen for the modelling, taking into account that the right side will present displacements and stresses of the same magnitude but opposite sign, in a mirroring fashion.



(a) Antisymmetry Axis (b) FE Discretization

Figure 7: Problem F

2.2 Assignment 2.2

The finite elements method is a numerical method for solving partial differential equations. At the same time, the differential equation to be solved is a mathematical model which intends to describe the behavior of a physical system found in nature. The reliability of a numerical solution depends on how precisely it approximates the exact solution of the mathematical model but also, how well the mathematical model describes the behavior of the system it is intended to simulate. This is where Verification and Validation come into play.

Verification consists of comparing the numerical solution with the exact solution of a differential equation (PDE), in order to quantify error and identify and reduce its sources (i.e. discretization errors, computation errors, etc.). There are several error estimation techniques which are used to perform this assessment. Validation, on the other hand, aims to compare the numerical results with experimental data, with the intention of assessing how well the mathematical and numerical models describe the physical problem.

In summary, verification is performed to assess if a mathematical model is being solved with sufficient precision with a numerical solution, while validation is about evaluating if said mathematical model correctly describes the physical problem of interest.

2.3 Assignment 2.3

The consistent nodal force vector for a 1D bar element is given by:

$$f_{ext} = \int_0^1 q \begin{bmatrix} 1-\xi \\ \xi \end{bmatrix} l d\xi \tag{2}$$

Where:

$\xi = \frac{x-x_1}{l} = \frac{x}{l}$ (in this case, $x_1 = 0$)

$q_x = \rho A \omega^2 x = \rho A \omega^2 \xi l$

$A = A_i(1-\xi) + A_j \xi$

Replacing these values in equation (2), we obtain the following expression:

$$f_{ext} = \rho \omega^2 l^2 \int_0^1 \begin{bmatrix} A_i(\xi - 2\xi^2 + \xi^3) + A_j(\xi^2 - \xi^3) \\ A_i(\xi^2 - \xi^3) + A_j \xi^3 \end{bmatrix} d\xi \tag{3}$$

Integrating yields:

$$\boxed{f_{ext} = \frac{\rho \omega^2 l^2}{12} \begin{bmatrix} A_i + A_j \\ A_i + 3A_j \end{bmatrix}} \tag{4}$$

Particularizing for the case of a prismatic bar ($A = A_i = A_j$):

$$\boxed{f_{ext} = \frac{\rho \omega^2 l^2}{6} \begin{bmatrix} A \\ 2A \end{bmatrix}} \tag{5}$$

3 References

- Oñate, E. Introduction to the Finite Element Method. 2008