Modeling with CMU Mini-FEA Program

Introduction

Finite element analysis (FEA) allows you analyze the stresses and displacements in a body when forces are applied. FEA determines the stresses and displacements by analyzing the body as a set of small connected elements. The equations that relate stresses, strains and displacements are solved approximately within each element. Generally, the more elements you use, the more accurate will be the stresses, strains and displacements. These equations of elasticity are similar in concept to equations you learn in mechanics of materials, but they are more general. They allow for the analysis of more complex shaped bodies, subjected to complex loadings.

The CMU Mini-FEA program uses the general equations, but only applies them to limited types of problems, as explained below. However, the main steps in the Mini-FEA program are the same as those in a commercial finite element program.

Here, we first give definitions and an overall picture of the steps in FEA. Then, we will give some modeling techniques in FEA. Step by step instructions in using the program can be found at the end.

Definitions and Overall Picture

Geometry

The program can only analyze a body shaped like a box. You prescribe two dimensions – the length is horizontal, height is vertical. The third dimension into the screen is 1.

Material

The body must be elastic. You specify the Young's modulus (E) and Poisson ratio (ν). The units must be consistent with those of the geometry and loads.

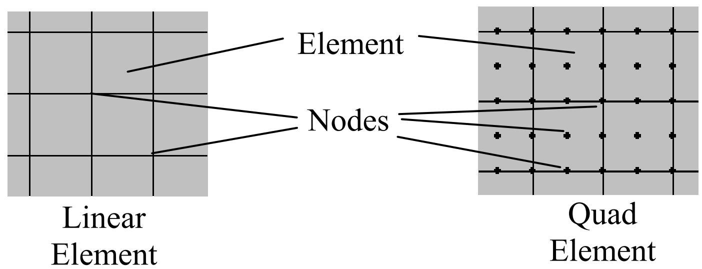
Meshing

This is where you break the body into elements. In this program the elements must be of uniform size and shape (rectangular). You specify how many elements in the x-direction and how many in the y-direction. You also specify one of two types of elements.

Linear elements: the displacement within an element is assumed vary linearly with x and y, so the stress is constant in that element.

Quad(ratic) *elements*: the displacement is quadratic and the stress varies linearly. Since the stresses in Quad elements vary linearly with position, these elements calculate stresses more accurately in problems that involve bending.

Nodes are at the corners of Linear elements and also along the sides and at the center of Quad elements (all dots).



Loads

This is where you describe the forces acting on the body, as well as regions where the body is restrained. Sometimes these are called boundary conditions. The mini-FEA program only allows you to specify forces acting in the x-y plane (the plane of the rectangle Length by Height) or displacements (motion) in the x-y plane. Forces or displacements can only be specified at nodes.

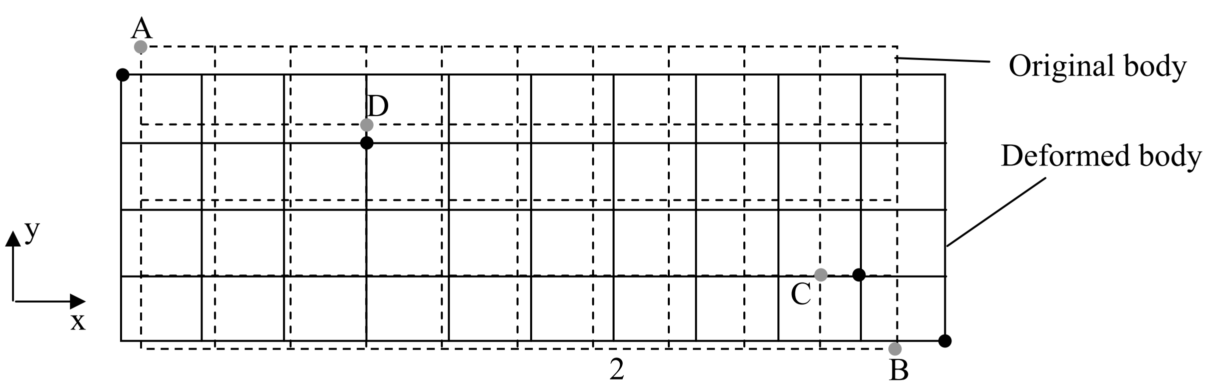
Forces
Fx is the external force at a node in the x-direction. If Fx > 0, force acts to the right. If Fx < 0, forces acts to the left. If Fx = 0, there is no external force in the x-direction.

Fy is the external force at a node in the y-direction. If Fy > 0, force acts upward. If Fy < 0, forces acts downward. If Fy = 0, there is no external force in the y-direction.

Displacements
Ux is the displacement of the body at a node in the x-direction due to the loading. If Ux > 0, the point displaces to the right. If Ux < 0, the point displaces to the left. If Ux = 0, the point does not displace in the x-direction.

Uy is the displacement of the body at a node in the y-direction due to the loading. If Uy > 0, the point displaces upward. If Uy < 0, the point displaces downward. If Uy = 0, the point does not displace in the y-direction.

In this example, Point A moves left and down ($U_x < 0$, $U_y < 0$). Point B moves right and up ($U_x > 0$, $U_y > 0$). Point C moves right and doesn't move up or down ($U_x > 0$, $U_y = 0$). Point D doesn't move left or right and moves down: ($U_x = 0$, $U_y < 0$).



For each node, you must do one of two things:

1. For most nodes, you don't specify the displacement or the force. In FEA, this is the same as saying there is zero external force at the point, or Fx = Fy = 0. (Ux and Uy will not be zero at this point, in general; they will be determined by the FEA program.).

2. You choose displacement (Ux) or force (Fx) in the x-direction and give it a value. And, you choose displacement (Uy) or force (Fy) in the y-direction and give it a value. Then, in the CMU mini-FEA program you click on a node, or you click on two nodes, depending on whether you want to specify displacements and/or forces on a single node or on an area (or line) of nodes. (This is explained in the step-by-step instructions.) So if you want the force Fx to have a value, and Fy to be zero, then, besides setting Fx to its value, you must make sure that Fy (and not Uy) is set to zero.

In FEA, you cannot apply only forces to a body and no displacements at all. Even if the forces are in equilibrium, the displacement of the whole body is not fully determined and you will get an error message. You must specify at least three displacements. A longer explanation of how to do this will be found under modeling techniques.

Solve

Once you have completed the above steps, you click on Solve for the program to complete the analysis. When it is done, you can visualize how the body deforms, and you determine the displacements, strains and stresses at any point.

Extract

Say you want to do calculations with the results on the computer calculator or in a spreadsheet. You can chose a set of nodes, and all forces, displacements, strains, and stresses at those nodes will be placed in a new window with tabs between each quantity. You can copy from that window and paste into a spreadsheet for further calculations.

Units

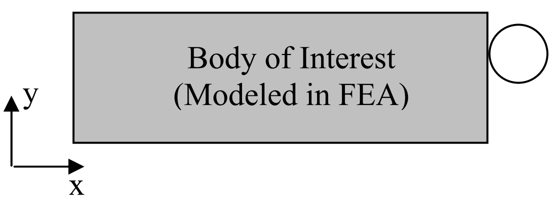
The length and height, the modulus E, the forces and the displacements must be in a consistent set of units, say, newtons and meters or pounds and inches. The results for stress and displacements will be in the associated units. For example, the following are consistent: length and height in meters, modulus E in newtons/meter^2, forces in newtons, and displacements in meters. Then, stress will be in newtons/meter^2. Another set of consistent units is: length and height in inch, modulus E in pounds/inch^2, forces in pounds, and displacements in inch. Then, stress will be in pounds/inch^2.

Modeling with FEA

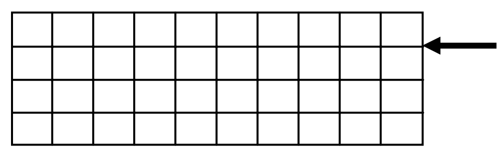
<u>Application of Forces</u>

Concentrated Force
In FEA, forces are applied at nodes. Say you want to simulate a force that acts on localized region on the boundary of the body. Then, prescribe the force (Fx in this case) at a single node.

Contact force between cylinder and block is known to be 100 N

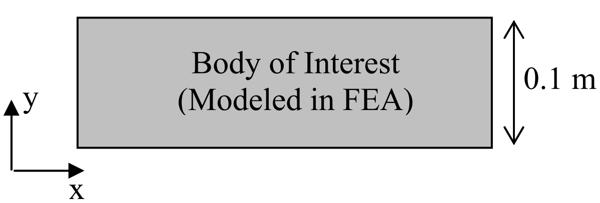


Prescribe Fx = -100 at node coinciding with contact point

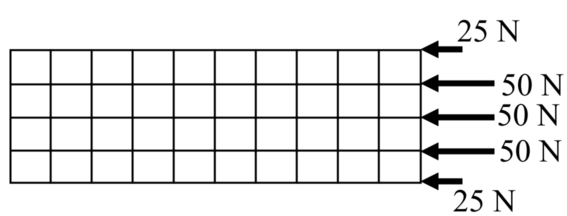


Distributed Force
Say you want to simulate a force per length of 2000 N/m that is uniformly distributed over the right end (that is, 200 N total). Say there are five nodes on the right end, how much force do we put on each node? We associate with each node the area half way to the next node. So, the top and bottom right corner nodes each have an associated area of 0.1/8. The three central nodes have 0.1/8 above and 0.1/8 below or 0.025 total. The net force on the corner nodes is (2000)(0.1)/8 = 25. The net force on the three central nodes is (2000)(0.2)/8 = 50.

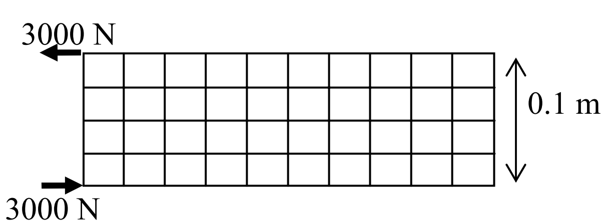
Uniformly distributed compressive force of 2000 N/m applied at right end of body.

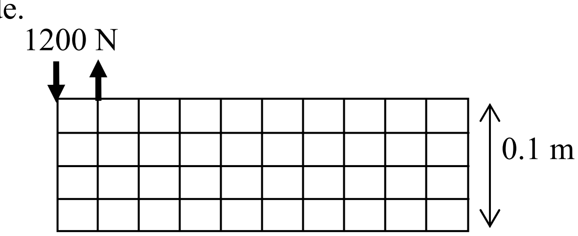


If there are four elements (5 nodes), prescribe forces Fx as shown



Couple
Say you want to simulate a counter clockwise couple of magnitude 300 N-m applied to the left end of the body. There are many ways of doing this. Two of them are shown. In both cases the forces are equal and opposite and so produce zero net force. The forces also result in a net couple of the desired magnitude.



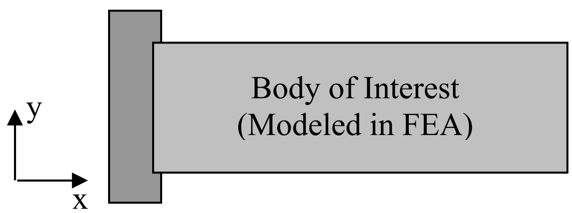


<u>Constraints on Body</u>

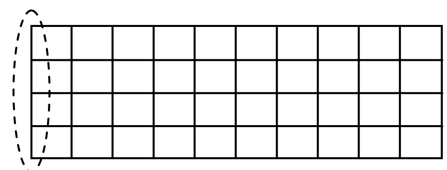
Fixed Boundary

In strength of materials, we often study the stresses in bodies that are fixed at some region. For beams, this is drawn as a cantilever. This means that all displacements at the end of the body that is cantilevered are prevented from displacing. We model this in FEA by setting the displacements at the fixed end equal to zero.

Body is perfectly attached to a rigid wall



Set Ux = Uy = 0 at nodes at end which is attached to the supporting wall

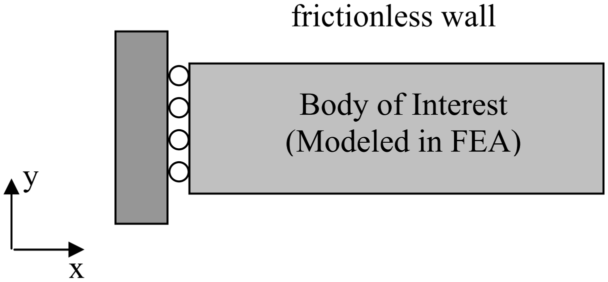


Contact with frictionless boundary

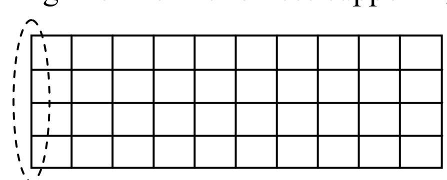
Alternatively, the body could press against, but is not bonded to, a surface that does not deform. Further, we could assume that the friction between the surface and the body is negligible. This means that there are no forces that resist the sliding of the body parallel to the surface. The sliding could occur, if the body expands laterally while being compressed against the surface. You may have seen this drawn with roller supports.

We model this is FEA by setting the displacement normal to the surface (Ux for this surface) equal to zero, and the force parallel to the surface (Fy = 0 for this surface) equal to zero.

Body is pressed against a rigid, frictionless wall



Set Ux = Fy = 0 at nodes at end which presses against the frictionless supporting wall



Constraints

Say there are forces applied to the body. Even if these forces are in equilibrium, in FEA you still need to specify the displacement at some points. In general, specified displacements should reflect actual constraints on the modeled body, that is, points where the body is prevented from moving.

At a minimum, for the program to run, minimal displacement constraints must ensure that the x-motion, the y-motion and the rotation of the body are constrained. Think about minimal displacement constraints this way. You want to prescribe displacements which

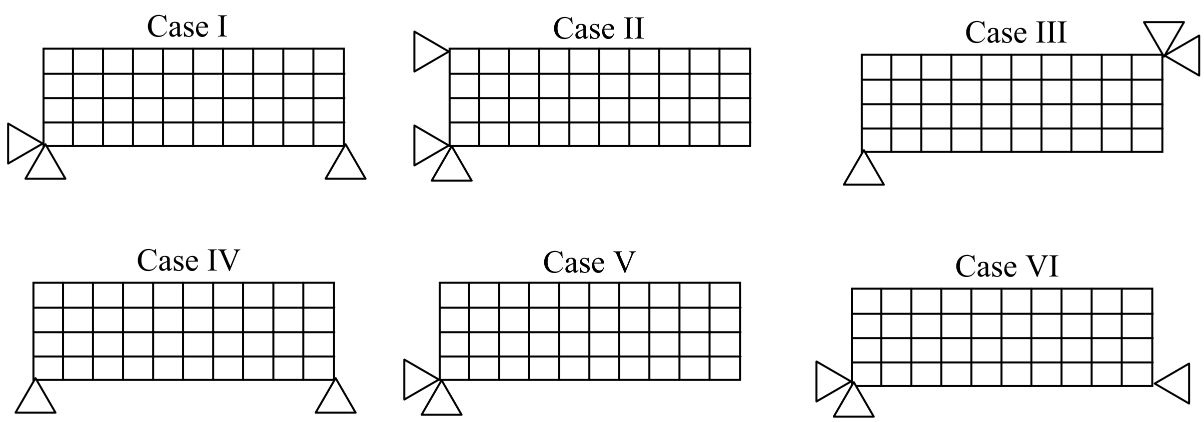
1. Do not allow any extra motion to occur, and
2. Do not prevent any straining of the body

In this examples, and more generally

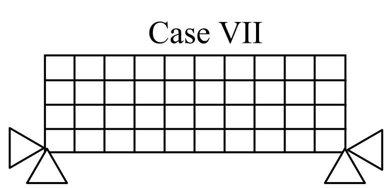
▷ or ◁ means that the horizontal displacement (Ux) is set (often to zero)

△ or ▽ means that the vertical displacement (Uy) is set (often to zero)

Cases I, II and III below are all properly constrained. Case IV is not: it allows a displacement in the x-direction, Case V is not: it allows a rotation about the pinned point (lower left corner). Case VI is not: it still allows a rotation about the lower left corner. A small rotation about that point would cause the lower right point to move up. The constraint on the x-displacement at the lower right point does not restrict such a vertical displacement and so does not prevent rotation.



Case VII below is an example of overly constraining supports that prevent straining that might way to occur. Say this body had an axial loading in the horizontal direction. It would want to expand in the x-direction, but having two constraints on Ux on the same horizontal line prevents that expansion. Now if the body were truly constrained in this way, then one should apply those constraints in FEA. However, if we model a body with just forces applied, and need to have minimal constraints on the displacement for FEA to run, then Case VII would not be acceptable.



Accounting for Thickness of Body

The body analyzed by the CMU Mini-FEA program always has thickness 1. You might want to analyze a body with a different thickness. You can still model the body in FEA with thickness 1, but you need to change nodal forces.

You want the displacements, strains and stresses to be identical in the bodies of two different thicknesses (your thickness and the thickness 1). But, the forces are stress times area, and the area includes the thickness of the plate. Say two bodies have identical planar dimensions, mesh, displacements, strains and stresses. Then, if one body is twice as thick as the other, its nodal forces will be twice as large.

For example, say your body which has thickness 0.01 (in the same units as the length and the height). Say there are forces applied to your body. Then, the forces to be applied to the FEA model should be 1/(0.01) = 100 times as large. In other words, the body of thickness 1 needs 100 times as much force to have the same stress. You must not change the displacements applied to the model. We want them to be the same in the bodies. Conversely, if you extract forces from the FEA results, then in the above example, the actual body of thickness 0.01 would have forces that are 0.01 as large as the FEA results.