Chapter 3

Two-Dimensional Finite Element

3.1 Introduction

The displacements, traction components and distributed body force values are functions of the position indicated by $(x,\, y)$The displacement vector u is given as:

$$\mathbf{u} = [u,\, v]^{\mathrm{T}}$$

where u and v are the x and y components of $\mathbf{u}$. The stresses and strains are given by

$$\sigma = [\sigma_x, \sigma_y, \tau_{xy}]^{\mathrm{T}} \qquad \epsilon = [\epsilon_x, \epsilon_y, \gamma_{xy}]^{\mathrm{T}}$$

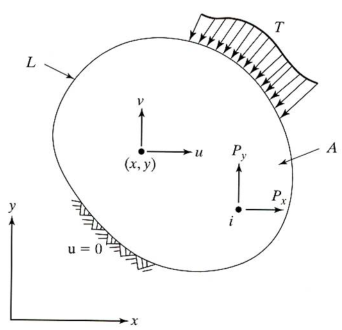


Figure 1:

From Figure 3.1, representing the two-dimensional problem in general setting, the body force, traction vector and elemental volume are given by:

$$\mathrm{f} = [f_x, f_y]^{\mathrm{T}},\ \mathrm{T} = [T_{\mathrm{x}},\, T_{\mathrm{y}}]^{\mathrm{T}} \quad and \quad dV = t\, dA$$

where t is the thickness along the z direction. The strain-displacement relations are given by

$$\epsilon = \left[\frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right]^{\mathrm{T}}$$

Stresses and strains are related by

$$\sigma = \mathrm{D}\epsilon$$

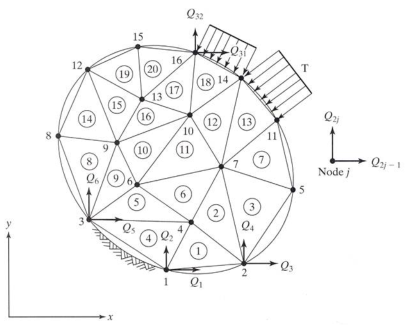


Figure 2:

Figure 3.2 above shows the two-dimensional region is divided into straight-sided triangles. The points where the corners of the triangles meet are called nodes. Each triangle is called an element. The elements fill the entire region except a small region at the boundary. It can be reduced by choosing smaller elements or elements with curved boundaries. For two-dimensional problems, each node is permitted to displace in only the two directions x and y. Thus, each node has two degree of freedom (dof). The numbering scheme for the displacement are taken as Q_{2j-1}in the x-direction and in Q_{2j} the y-direction. We denote the global displacement vector as

$$\mathrm{Q} = [Q_1, Q_2, \ldots, Q_N]^{\mathrm{T}}$$

where N is the number of degrees of freedom. Information about the numbering scheme can be seen from the element connectivity in Table 3.1 below.

Table 1:

Element Number	Three Nodes		
e	1	2	3
1	1	2	4
2	4	2	7
$\vdots$	$\vdots$	$\vdots$	$\vdots$
11	6	7	10
$\vdots$	$\vdots$	$\vdots$	$\vdots$
20	13	16	15

The displacement components of a local node j in Figure 3.3 are represented as q2j-1 and q2j in the x and y directions. Thus, the element displacement vector is

$$\mathrm{q} = [q_1, q_2, \ldots, q_6]^{\mathrm{T}}$$

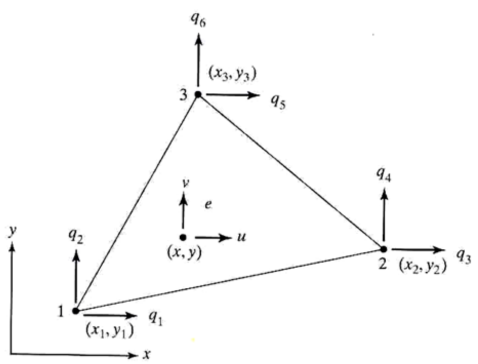


Figure 3:

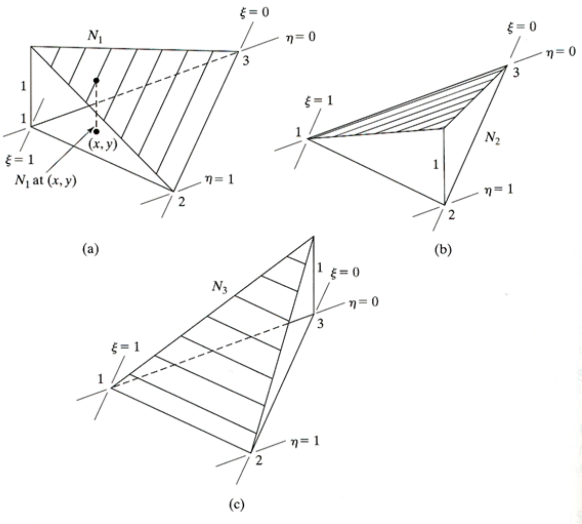


Figure 4:

3.2.1 Displacement

The displacements at points inside an element as shown in Figure 3.4 above is represented as:

$$N_1 + N_2 + N_3 = 1$$

where

$$N_1 = \xi = \frac{A_1}{A}$$

$$N_2 = \eta = \frac{A_2}{A}$$

$$N_3 = 1 - \xi - \eta = \frac{A_3}{A}$$

The displacements can be written using shape functions. We have:

$$u = N_1 q_1 + N_2 q_3 + N_3 q_5$$

$$v = N_1 q_2 + N_2 q_4 + N_3 q_6$$

or becomes

$$u = [q_1 + q_5]\,\xi + (q_3 + q_5)\,\eta + q_5$$

$$v = [q_2 - q_6]\,\xi + (q_4 - q_6)\,\eta + q_6$$

In the x and y coordinates:

$$x = N_1 x_1 + N_2 x_2 + N_3 x_3$$

$$y = N_1 y_1 + N_2 y_2 + N_3 y_3$$

or

$$x = (x_1 - x_3)\,\xi + (x_2 - x_3)\,\eta + x_3$$

$$y = (y_1 - y_3)\,\xi + (y_2 - y_3)\,\eta + y_3$$

or in simpler way

$$x = x_{13}\xi + x_{23}\eta + x_3$$

$$y = y_{13}\xi + y_{23}\eta + y_3$$

3.2.2 Strain

Using chain rule, partial derivatives of u and v with respect to x and y or ξ and η gives.

$$\frac{\partial u}{\partial \xi} = \frac{\partial u}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial u}{\partial y}\frac{\partial y}{\partial \xi}$$

$$\frac{\partial u}{\partial \eta} = \frac{\partial u}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial u}{\partial y}\frac{\partial y}{\partial \eta}$$

which can be written in matrix notation as

$$\left\{ \begin{array}{c} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{array} \right\} = \left[\begin{array}{cc} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{array} \right]$$

where (2x2) matrix is denoted as the Jacobian of the transformation, J:

$$\mathrm{J} = \left[\begin{array}{cc} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{array} \right]$$

On taking the derivatives of x and y,

$$\mathrm{J} = \left[\begin{array}{cc} x_{13} & y_{13} \\ x_{23} & y_{23} \end{array} \right]$$

Strain, ϵ

$$\epsilon = \left\{ \begin{array}{ccc} \frac{\partial u}{\partial x} & & \\ \frac{\partial v}{\partial y} & & \\ \frac{\partial u}{\partial y} & + & \frac{\partial v}{\partial x} \end{array} \right\}$$

$$\epsilon = \frac{1}{\det \mathrm{J}} \left\{ \begin{array}{ccccccccccc} y_{21}q_1 & + & y_{31}q_3 & + & y_{12}q_5 & & & & & & \\ x_{32}q_2 & + & x_{13}q_4 & + & x_{21}q_6 & & & & & & \\ x_{32}q_1 & + & y_{23}q_2 & + & x_{13}q_3 & + & y_{31}q_4 & + & x_{21}q_5 & + & y_{12}q_6 \end{array} \right\}$$

This equation can be written in matrix form as $\epsilon = \mathrm{Bq}$, where

$$\mathrm{B} = \frac{1}{\det \mathrm{J}} \left[\begin{array}{cccccc} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{12} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{array} \right]$$

3.2.3 Potential-Energy Approach, Π

The potential energy of the system, $\Pi$is given as:

$$\Pi = \frac{1}{2}\int_A \epsilon^{\mathrm{T}}\mathrm{D}\epsilon t\, dA - \int_A \mathrm{u}^{\mathrm{T}}\mathrm{f}t\, dA - \int_\ell \mathrm{u}^{\mathrm{T}} t\, d\ell - \sum_i \mathrm{u_i}^{\mathrm{T}}\mathrm{P}_i$$

3.2.4 Galerkin Approach

Variational form is given by:

$$\int_A \sigma^{\mathrm{T}}\epsilon(\phi)\, t\, dA - \left(\int_A \phi^{\mathrm{T}}\mathrm{f}t\, dA + \int_\ell \phi^{\mathrm{T}}\mathrm{T}t\, d\ell - \sum_i \phi_{\mathrm{i}}{}^{\mathrm{T}}\mathrm{P}_i\right) = 0$$

3.2.5 Element Stiffness

Element strain energy is given by :

$$\begin{aligned} U_e &= \tfrac{1}{2}\int_e \epsilon^{\mathrm{T}}\mathrm{D}\epsilon t\, dA \\ &= \tfrac{1}{2}\int_e \mathrm{q}^{\mathrm{T}}\mathrm{B}^{\mathrm{T}}\mathrm{DBq}t\, dA \\ &= \tfrac{1}{2}\mathrm{q}^{\mathrm{T}}\mathrm{B}^{\mathrm{T}}\mathrm{DBq}t_e \int_e dA \\ &= \tfrac{1}{2}\mathrm{q}^{\mathrm{T}} t_e A_e \mathrm{B}^{\mathrm{T}}\mathrm{DBq} \\ &= \tfrac{1}{2}\mathrm{q}^{\mathrm{T}}\mathrm{k}^e\mathrm{q} \end{aligned}$$

where k^e is the element stiffness matrix is given by:

$$\mathrm{k}^e = t_e A_e \mathrm{B}^{\mathrm{T}}\mathrm{DB}$$

then

$$\mathrm{U} = \sum_e \frac{1}{2}\mathrm{q}^{\mathrm{T}}\mathrm{k}^e\mathrm{q}$$

$$\mathrm{U} = \sum_e \frac{1}{2}\mathrm{q}^{\mathrm{T}}\mathrm{KQ}$$

3.2.6 Force Terms

1) Body force term $\int_e \mathrm{u}^{\mathrm{T}}\mathrm{f}t\, dA$ is given by :

$$\int_A \mathrm{u}^{\mathrm{T}}\mathrm{f}t\, dA = \mathrm{q}^{\mathrm{T}}\mathrm{f}^e$$

where f^{e} is the element body force vector given by:

$$\mathrm{f}^{e} = \frac{t_e A_e}{3}\left[f_x,\ f_y,\ f_x,\ f_y,\ f_x,\ f_y\right]^{\mathrm{T}}$$

Global force vector **F** is:

$$\mathrm{F} = \sum_{\mathrm{e}} \mathrm{f}^{\mathrm{e}}$$

2) Traction force term, $\int_{\ell} \mathrm{u}^{\mathrm{T}}\mathrm{T}t\, d\ell$is given by:

$$\int_{\ell_{1-2}} \mathrm{u}^{\mathrm{T}}\mathrm{T}t\, d\ell = \left[q_1,\ q_2,\ q_3,\ q_4\right] \mathrm{T}^{e}$$

where $\mathrm{T}^{e}$is:

$$\mathrm{T}^{e} = \frac{t_e \ell_{1-2}}{6}\left[2\mathrm{T}_{x1} + \mathrm{T}_{x2},\ 2\mathrm{T}_{y1} + \mathrm{T}_{y2},\ \mathrm{T}_{x1} + 2\mathrm{T}_{x2},\ \mathrm{T}_{y1} + 2\mathrm{T}_{y2},\right]^{\mathrm{T}}$$

where,

$$\ell_{1-2} = \sqrt{\left(x_2 - x_1\right)^2 + \left(y_2 - y_1\right)^2}$$

3) Point load term, $\mathrm{u}^{\mathrm{T}}\mathrm{P}_i$is given by:

$$\mathrm{u}^{\mathrm{T}}\mathrm{P}_i = Q_{2i-1}P_x + Q_{2i}P_y$$

The contribution of body forces, traction forces and point loads to the global force **F** can be represented as:

$$\mathrm{F} \leftarrow \sum_{e}\left(\mathrm{f}^{e} + \mathrm{T}^{e}\right) + \mathrm{P}$$

The total potential energy, $\Pi$is given by:

$$\Pi = \frac{1}{2}\mathrm{Q}^{\mathrm{T}}\mathrm{KQ}{-}\mathrm{Q}^{\mathrm{T}}\mathrm{F}$$

Then,

$$KQ = F$$

where **K** and **F** are modified stiffness matrix and force vector.

3.2.7 Stress Calculation

Using the stress-strain relations and element strain-displacement relations, stress, is given by:

$$\sigma = \mathrm{DBq}$$

3.2.8 Temperature Effects

The element temperature load is given by:

$$\theta^e = t_e A_e \mathrm{B}^{\mathrm{T}} \mathrm{D} \epsilon_0$$

For plane stress, $\epsilon_0 = [\alpha \Delta \mathrm{T}, \alpha \Delta \mathrm{T}, 0]^{\mathrm{T}}$

For plane strain, $\epsilon_0 = (1 + v) [\alpha \Delta \mathrm{T}, \alpha \Delta \mathrm{T}, 0]^{\mathrm{T}}$

Example: For the two-dimensional loaded plate shown in Figure 3.5 below, determine the displacement of nodes 1 and 2 and the element stresses using plane stress conditions. Body force may be neglected in comparison with the external forces.

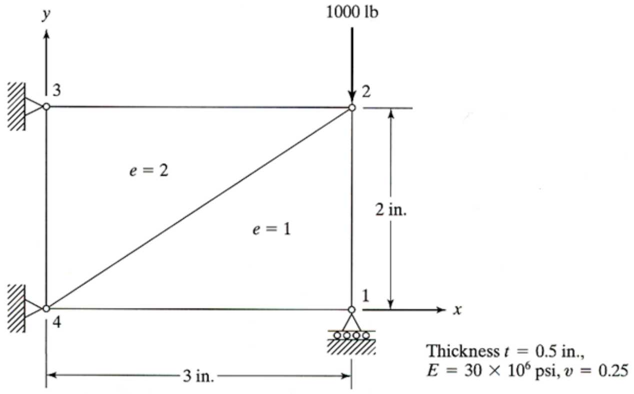


Figure 5:

Solution:

For plane stress conditions, the material property matrix is given by:

$$D = \frac{E}{1-v^2}\begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1-v}{2} \end{bmatrix} = \begin{bmatrix} 3.2\text{x}10^7 & 0.8\text{x}10^7 & 0 \\ 0.8\text{x}10^7 & 3.2\text{x}10^7 & 0 \\ 0 & 0 & 1.2\text{x}10^7 \end{bmatrix}$$

Using the local numbering pattern, we establish the connectivity as follows:

Element Number	Three Nodes		
	1	2	3
1	1	2	4
2	3	4	2

We know that:

$$\mathrm{B^e} = \frac{1}{\det \mathrm{J}}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{12} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix}$$

where, $\det \mathrm{J} = x_{13}y_{23} - x_{23}y_{13}$

On performing the matrix multiplication $\mathrm{DB^e}$, we get

$$\mathrm{DB}^1 = 10^7 \begin{bmatrix} 1.067 & -0.4 & 0 & 0.4 & -1.067 & 0 \\ 0.267 & -1.6 & 0 & 1.6 & -0.267 & 0 \\ -0.6 & 0.4 & 0.6 & 0 & 0 & -0.4 \end{bmatrix}$$

$$\mathrm{DB}^2 = 10^7 \begin{bmatrix} -1.067 & 0.4 & 0 & -0.4 & -1.067 & 0 \\ -0.267 & 1.6 & 0 & -1.6 & -0.267 & 0 \\ 0.6 & -0.4 & -0.6 & 0 & 0 & -0.4 \end{bmatrix}$$

These two relationship will be used later in calculating stresses using $\sigma^e = \mathrm{DB^e q}$

The multiplication $t_e A_e \mathrm{B}^{e\mathrm{T}}\mathrm{DB^e}$ gives the element stiffness matrices,

$$\mathrm{k}^1 = 10^7 \begin{bmatrix} 0.983 & -0.5 & -0.45 & 0.2 & -0.533 & 0.3 \\ & 1.4 & 0.3 & -1.2 & 0.2 & -0.2 \\ & & 0.45 & 0 & 0 & -0.3 \\ & & & 1.2 & -0.2 & 0 \\ & & & & 0.533 & 0 \\ \mathit{Symetric} & & & & & 0.2 \end{bmatrix}$$

$$\mathrm{k}^2 = 10^7 \begin{bmatrix} 0.983 & -0.5 & -0.45 & 0.2 & -0.533 & 0.3 \\ & 1.4 & 0.3 & -1.2 & 0.2 & -0.2 \\ & & 0.45 & 0 & 0 & -0.3 \\ & & & 1.2 & -0.2 & 0 \\ & & & & 0.533 & 0 \\ \mathit{Symetric} & & & & & 0.2 \end{bmatrix}$$

In the problem under consideration, Q_2, Q_5, Q_6, Q_7 and Q_8 are all zero.

Using the elimination approach, it is sufficient to consider the stiffness associated with the degree of freedomQ_1, Q_3 and Q_4. Since the body force is neglected, the first vector has the component F_4 = -1000lb. The set of equation is given by the matrix representation:

$$10^7 \begin{bmatrix} 0.983 & -0.45 & 0.2 \\ -0.45 & 0.983 & 0 \\ 0.2 & 0 & 1.4 \end{bmatrix} \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -1000 \end{Bmatrix}$$

Solving for Q_1, Q_3 and Q_4we get,

$$Q_1 = 1.913\text{x}10^{-5}\text{in}, \quad Q3 = 0.875\text{x}10^{-5}\text{in}, \quad Q_4 = -7.436\text{x}10^{-5}\text{in}$$

For element 1, the element nodal displacement vector is given by:

$$\mathrm{q}^1 = 10^{-5}\begin{bmatrix} 1.913 & 0 & 0.875 & -7.436 & 0 & 0 \end{bmatrix}^{\mathrm{T}}$$

The element stresses, $\sigma^1$are calculated from $\mathrm{DB}^1\mathrm{q}$

$$\sigma^1 = \begin{bmatrix} -93.3 & -1138.7 & -62.3 \end{bmatrix}^{\mathrm{T}} \text{psi}$$

Similarly,

$$\mathrm{q}^2 = 10^{-5}\begin{bmatrix} 0 & 0 & 0 & 0 & 0.875 & -7.436 \end{bmatrix}^{\mathrm{T}}$$

$$\sigma^2 = \begin{bmatrix} 93.4 & 23.4 & -297.4 \end{bmatrix}^{\mathrm{T}} \text{psi}$$

Solution using CST program

Input data file for CST program

2D STRESS ANALYSIS USING CST

```
   NN        NE     NM        NDIM    NEN        NDN
    4         2      1           2      3          2
   ND        NL   NMPC
    5         1      0
Node#         X      Y
    1         3      0
    2         3      2
    3         0      2
    4         0      0
Elem#        N1     N2          N3   Mat#  Thickness  TempRise
    1         4      1           2      1        0.5         0
    2         3      4           2      1        0.5         0
DOF#   Displacement
    2         0
    5         0
    6         0
    7         0
    8         0
DOF#   Load
    4     -1000
 MAT#         E     Nu      Alpha
    1  3.00E+07   0.25   1.20E-05
B1       i       B2      j        B3      (Multi-    constr.  B1*Qi+B2*Qj=B3)
                                          point
```

Result from CST program:

Node#		X-Displ	Y-Displ				
	1	1.908E-(	-5.862E-09				
	2	8.733E-06	-7.416E-05				
	3	1.922E-09	-1.184E-09				
	4	-1.922E-09	-9.709E-11				
Elem#		SX	SY	Txy	S1	S2	S3
	1	-93.122379	-1135.592	-62.08159	-89.4383	-1139.28	-3.3961
	2	93.122379	23.2642918	-296.612	356.8549	-240.468	-41.6419
DOF#		Reaction					
	2	820.6532					
	5	-269.02021					
	6	165.75423					
	7	269.02021					
	8	13.592574					

The computer result may differ slightly since the penalty approach for handling boundary conditions is used in the computer program.

3.3 Basic Modeling and Simulation Procedures Using Femlab

Model Navigator

The Model Navigator appears when we start Femlab or when you restrart from stracth within femlab by selecting New from the File menu or clicking on the New toolbar button. In the Model Navigator, we specify the application mode, names of dependent variables and the nature of the problem; stationary(static), eigenfrequency, time dependent(transient), quasi static, parametric or frequency response.

Options and Settings

This section reviews basic settings for example those for the axes and grid spacing. All settings are accessible from the Options menu. It is possible to define material, loads and constraints in a user defined coordinate system.

Geometry Modeling

- Either the model geometry is set up inside Femlab or imported from other software in 1D, 2D or 3D.
- DXF fail format for 2D geometry.
- IGES fail format for 3D geometry.

Physic Settings

Material properties are usually defined on the subdomain.

Application Mode Properties

Application mode properties are global properties controlling the analysis starting with which analysis to perform. Make all corresponding settings in the Application Mode Properties dialog box, which you open from Properties in the Physics menu.

Application Scalar Variables

Application scalar variables are global variables defining an analysis. The Structural Mechanics Module application modes have only one variable, freq, the excitation frequency in a frequency-response analysis. To open the Application Scalar Variables dialog box go to the selection Scalar Variables in the Physics menu.

Point Settings

We define loads and constraints on points in Point mode. Point settings are used in all Structural Mechanics Module application modes, whereas point masses are defined in the Beam application modes.

Edge Settings

Edge settings are used only in these 3D application modes: shell, solid, and beam. In the Beam application mode we define physical properties as well as loads and constraints, whereas the other application modes define only loads and constraints.

Boundary Settings

In Boundary Selection mode, we specify loads and constraints on the edges (in 2D) or faces (in 3D). For In-plane Euler beams and shells the physical properties are also defined.

Subdomain Settings

In Subdomain Selection mode, we specify the properties of the material. We can also control the element to use from this dialog box. For all continuum application modes we can select the order of the Lagrange element. In addition, for time-dependent and nonlinear problems we set initial conditions for subdomains in this mode.

Mesh Generation

At this stage the software meshes the geometry. We simply click on one of the Mesh buttons in the Main toolbar; in other cases it is necessary to set some mesh parameters using the Mesh menu and the Mesh Parameters dialog box.

Computing the Solution

To compute the solution, press the Solve Problem toolbar button on the Main toolbar. We can specific settings for each solver in the Solver Parameters dialog box.

Postprocessing and Visualization

FEMLAB's powerful visualization tools are accessible in the program's Postprocessing mode, but to use them we must be familiar with the Postprocessing menu.

Steps to model and simulate using Femlab are stated below:

1. Geometry modeling :
   • Either the model geometry is set up inside Femlab or imported from other software in 1D, 2D or 3D.
   • DXF file format for 2D geometry.
   • IGES file format for 3D geometry.

2. Defining material data or material properties.

3. Defining constraint dan applied loads.

4. Mesh generation.

5. Defining type of analysis.

6. Computing the solution.

7. Postprocessing and visualization.

Static Analysis

Below is a geometry and structure with height of 12 cm and 5 cm width. The top portion of the structure is constraint and 4000N load is applied at the bottom section.

Model Geometry:

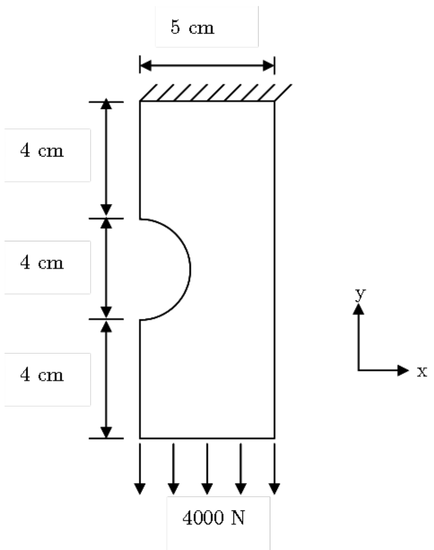


Figure 6:

Material and element data:

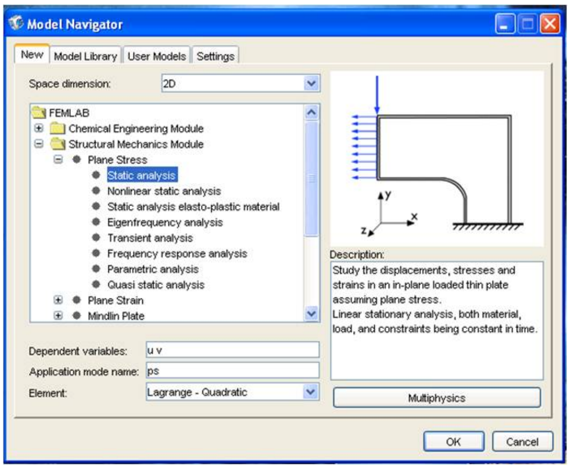
1. Material is structural steel.
2. Top portion is constrained and load of 4000 N is applied at the bottom section.
3. Geometry thickness is 1 cm.

Note : All unit in Femlab is SI unit.

PROCEDURES :

1) Model Navigator

1. Double click on the Femlab icon on the desktop.
2. Go to the New page in the Model Navigator, then select 2D from the Space dimension list.
3. On that same page, select Structural Mechanics Module, Plane Stress, Static analysis.
4. Click on OK to close the Model Navigator.



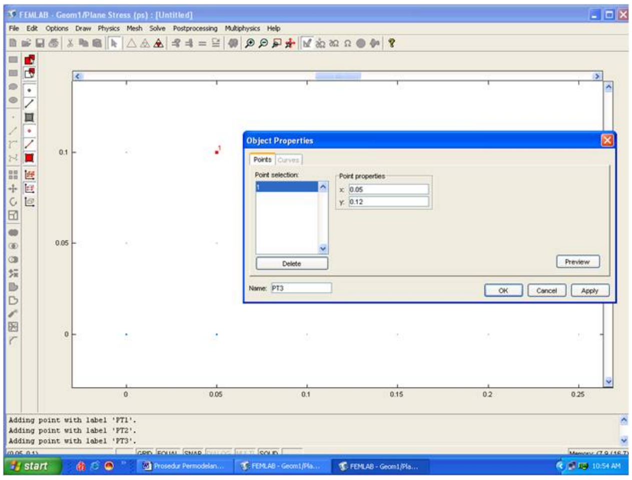
2) Geometry Modeling

Defining axes and grid size:

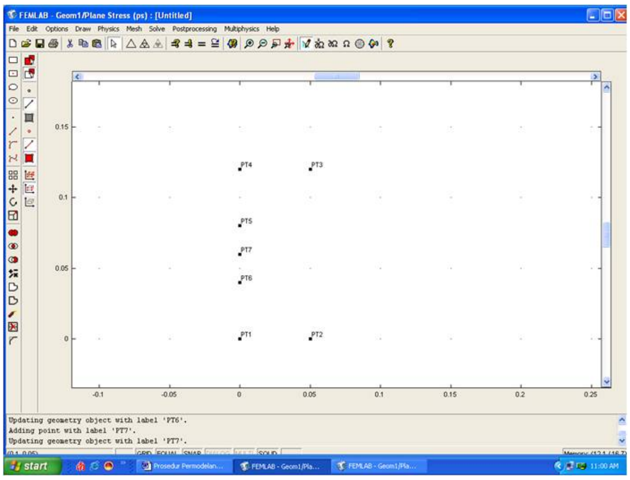
- From **Options** menu, choose **Axis/Grid Settings**. Click on the **Grid** tab and click once more on the **Auto** box to define axes or grid size manually. Set 0.05 for x, y and z-spacing. Click on **Apply** and click **OK** to close the dialog box.

Create Points:

- Using the **Point** button at the side toolbar, create two points at (0,0) and (0.05,0).
- Click on the **Zoom In** button at the top toolbar to enlarge image.
- Create a point at (0.05,0.12). To do this, first, make a point at (0.05,0.1). Click twice on that point. **Object Properties** dialog bix will appear. Choose 1 in the **Point selection** and write 0.05 for x and 0.12 for y in **Point properties**.
- Click **Apply** and **OK**.



- Similarly, make four more points at (0,0.12), (0,0.08), (0,0.06), (0,0.04).
- All created points are shown below.

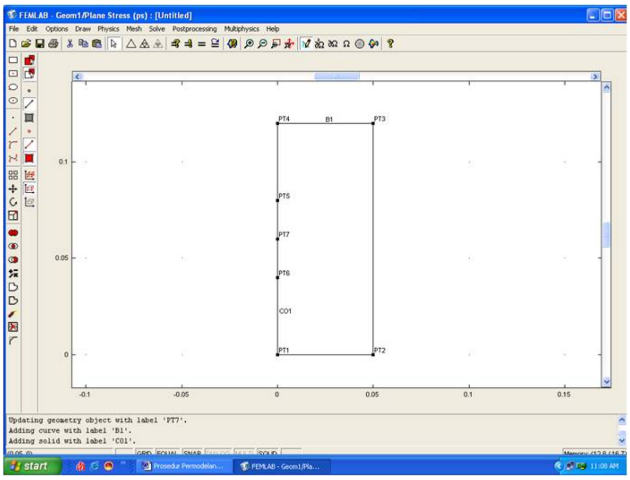


Enlarge Image:

- To enlarge image, click the **Zoom In** , **Zoom Window** or **Zoom Extents** button placed at the top toolbar.

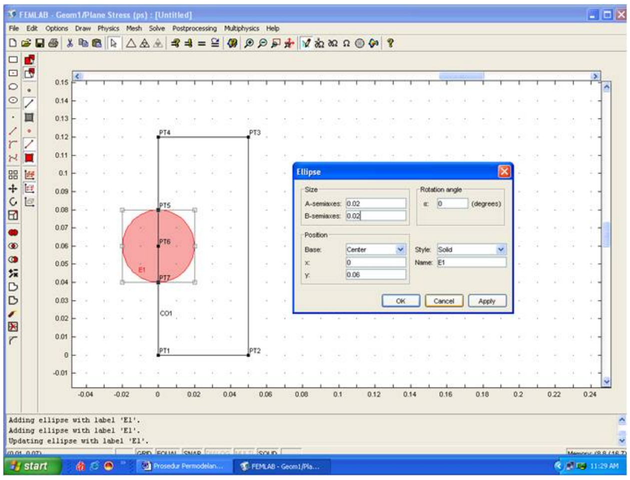
Create lines connecting the points:

- Click on the **Line** button at the side toolbar to make lines.
- To make a line connecting points 1 and 2, left click on point 1 and left click once again on point 2. Then right click.



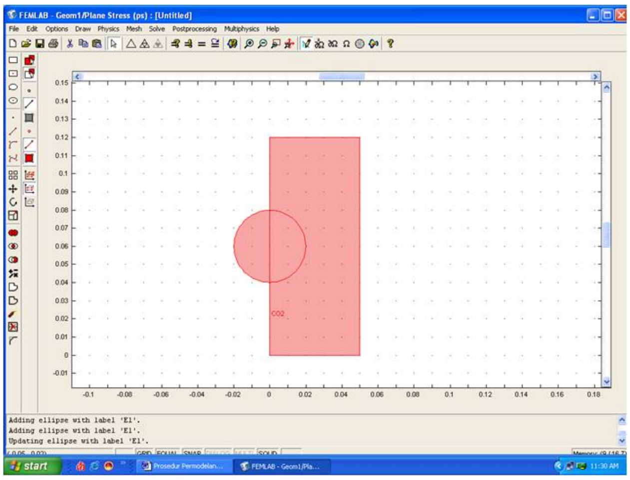
Make circle :

- Make a circle at coordinate (0,0.06) with diameter of 0.04 m by clicking on the **Ellpise/Circle (Centered)** button placed at the side toolbar.
- To make it easier, grid size will be reduced to 0.01. Click on the **Options** menu and choose **Axis/Grid Settings**. Set 0.01 in the x, y and z spacing.
- Click on the **Ellpise/Circle** (Centered) button. Click at coordinate of (0,0.06) and make a circle.
- Click twice on the circle. In the Ellipse dialog box, make sure that **A-semiaxes** and **B-semiaxes** is 0.02. In the **Position space**, **Base** is center, x:0 and y:0.06.
- Click **Apply** and **OK**.

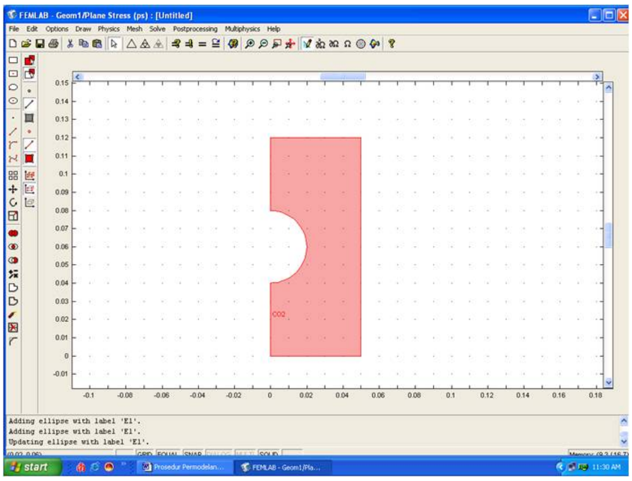


Create solid object:

- Choose/click all entities (lines, points, circle) and click **Coerce to Solid** button placed at the side toolbar. Use the shift button on the keyboard to select more than one entities.



- The circle will be separated from the geometry to produce our final geometry. Click on the **Split object** button. Then click on the **Difference** button. All this button placed at the side toolbar.



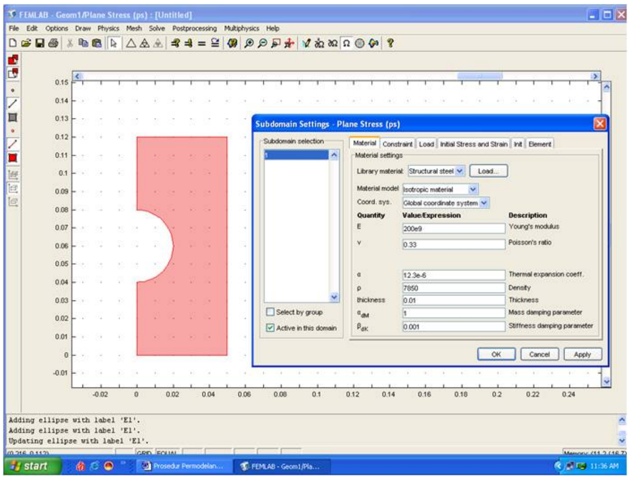
3) Defining material data or material properties.

After setting up the geometry, second step is to define material data or material properties. For this problem, the material data is taken from the Femlab Material library.

Click on the Physics menu and choose Subdomain Settings.

- In Subdomain Settings dialog box, click at the Material tab.
- In the Subdomain selection , choose 1 and click on the Load button to choose material.
- Choose Structural steel and click Apply. Then click OK to close the dialog box.
- In the Subdomain Settings dialog box, set the element thickness as 1 cm by writing 0.01 in the Thickness edit field.

- Click Apply and OK to close the dialog box.

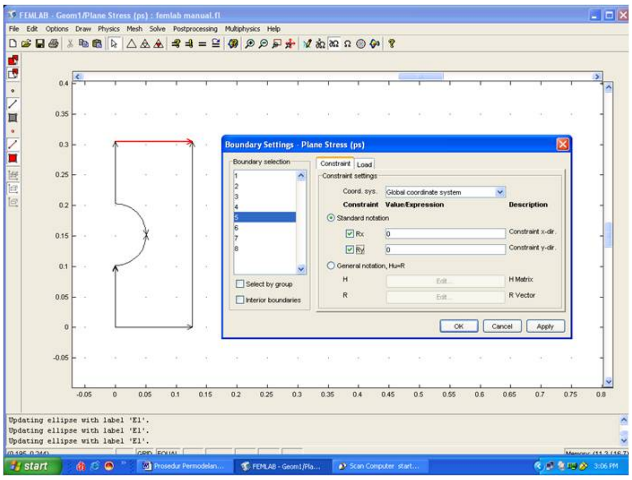


4) Defining constraint and applied loads

In this third step, constraint ang applied loads will be defined.

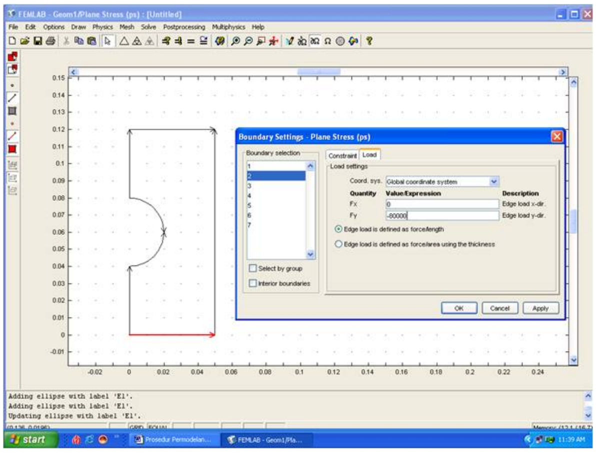
Defining constraint:

- In **Phsyics** menu, choose **Boundary Settings**.
- Click on the line on top of the geometry and set **Constraint** in the x and y direction by clicking on the **Rx** and **Ry** box.
- Click **Apply**.



Defining applied loads:

- Click on the bottom line of the geometry.
- Click on the **Load** tab.
- Make sure that the **Edge load** is defined as **force/length** is on.
- In the Fy edit field, write -80000 N/m . Negative sign shows that the load is downward. (Note: 4000/0.05 = 80000 N/m).

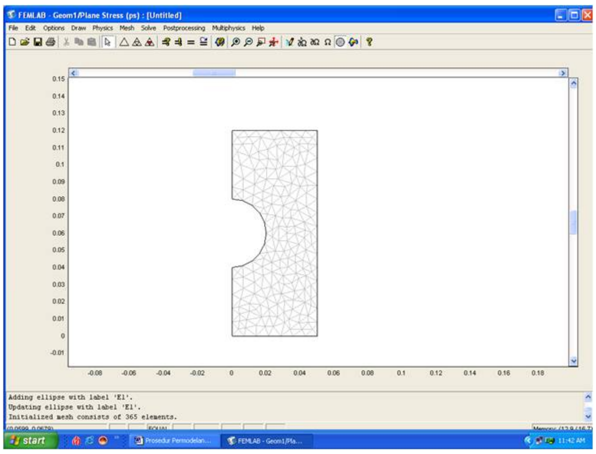


- Click on the **Apply** button and **OK** button to close the dialog box

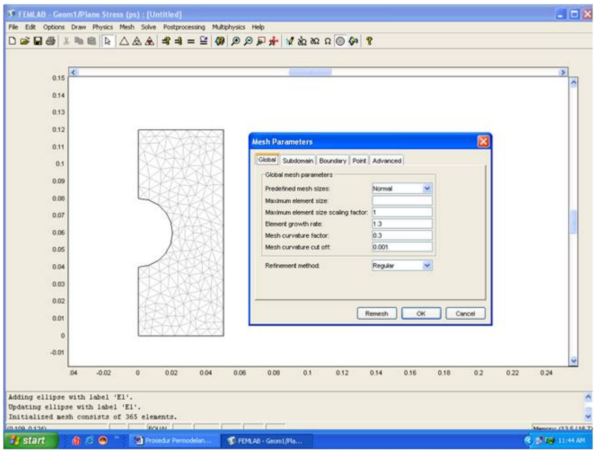
5) Mesh generation

After constraint and applied loads are defined, mesh will be generated.

- Mesh is generated automatically by clicking the **Initialize Mesh** button. Mesh size can be reduced to obtained better result by clicking the **Refine Mesh** button.
- Now, click on the **Initialize Mesh** button placed on the top toolbar.



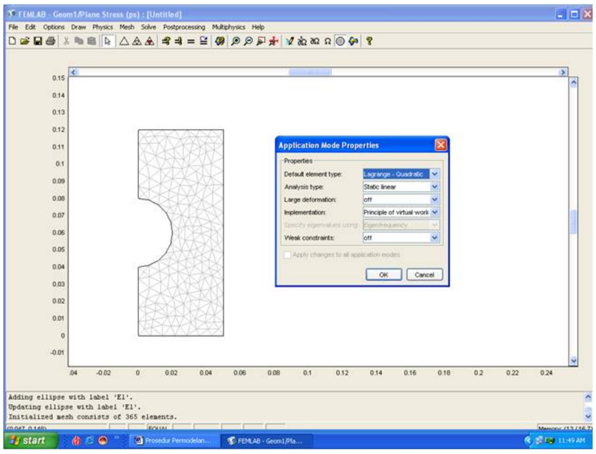
- Mesh parameter such as size, element growth rate and curvature factor can be change by choosing **Mesh Parameters** from the **Mesh** menu list.



6) Defining type of analysis.

After mesh is generated, type of anlaysis need to be defined. Analisis bagi struktur ini adalah analisis statik dan ubahbentuk struktur adalah dalam kawasan linear memandangkan daya yang diaplikasikan adalah lebih rendah berbanding takat alah bahan.

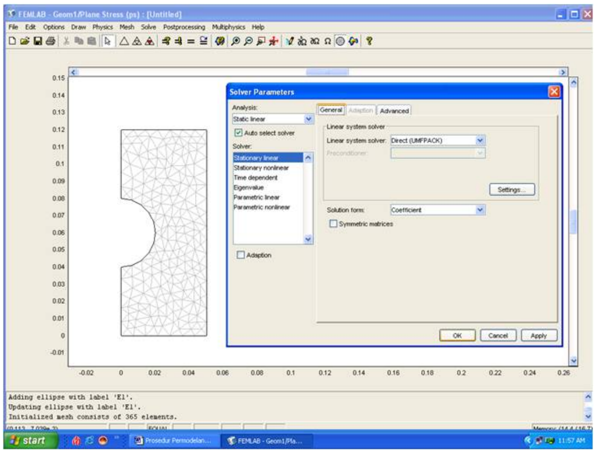
- Click on the **Physics** menu and choose **Properties**.
- In the **Application Mode Properties** dialog box, choose **Static linear** as the **Analysis** type.



- Large deformation and Weak constraint are set off.
- Click on the OK button to close the dialog box.

7) Computing the solution

- Click on the **Solve** menu and choose **Solver Parameters**.
- Make sure the Analysis is **Static linear** and Solver is Stationary linear.
- Click **Apply** button and then **OK** button.
- Simulation is performed by clicking the **Solve** button placed on the top toolbar.

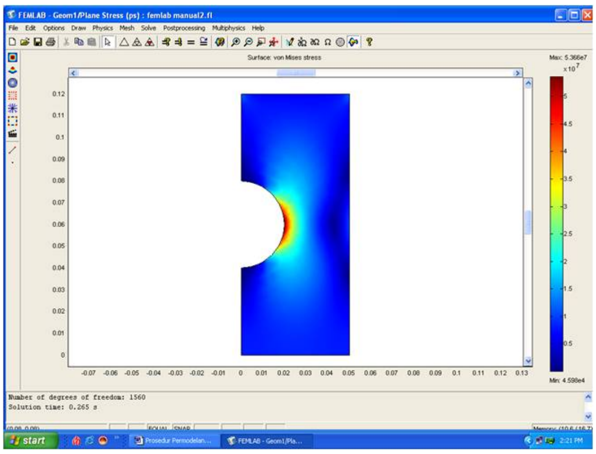


8) Postprocessing and visualization

In this postprocessing and visualization, analysis result such as von mises stress, x displacement and y displacement will be visualized.

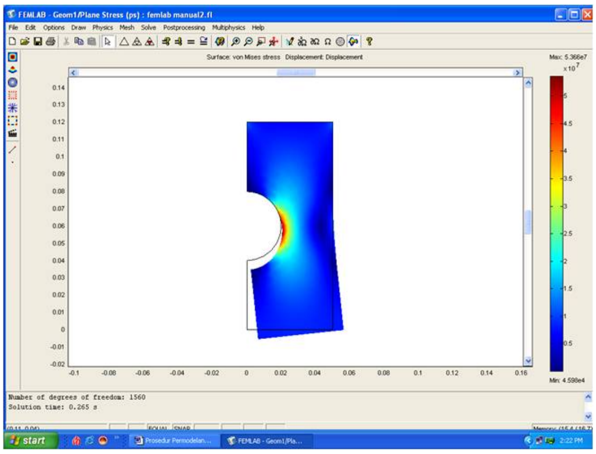
i) Von Mises Stress.

- The result of von mises stress on the surface structure is shown below.



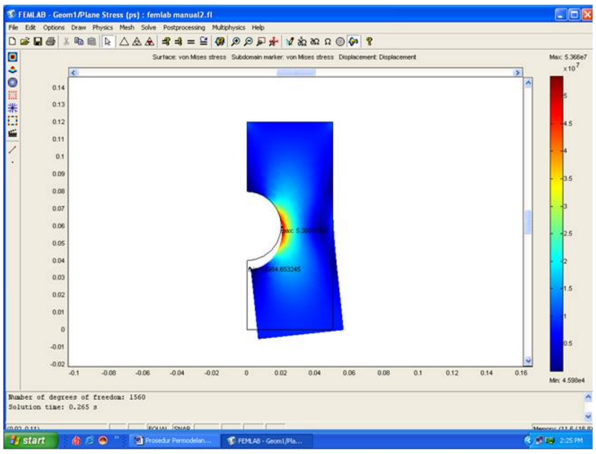
ii) Structure deformation:

- Click on the **Postprocessing** menu and choose **Plot Parameters**.
- Click on the **General** tab. Click on the **Deformed shape** box. Then click **Apply** and **OK**.
- The result of structure deformation under applied load is shown below.



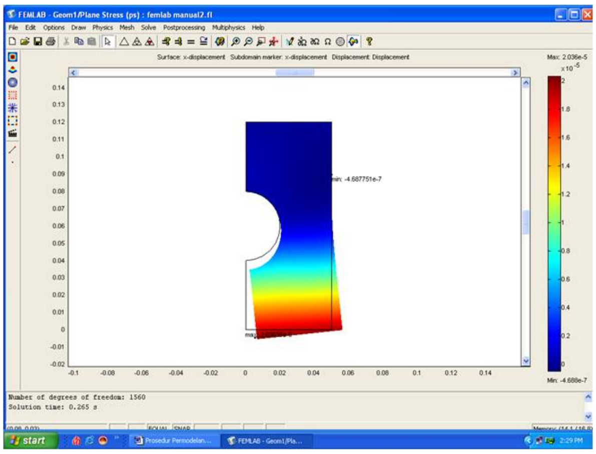
iii) Point of maximum and minimum von mises stress

- Click on the **Postprocessing** menu and choose **Plot Parameter**s. Click on the **Max/Min** tab. In the **Plot Parameter** dialog box, make sure that **Pre-defined quantities** is Von Mises Stress.
- Click on the **Max/min marker** box to activate the marker. Click **Apply** and then **OK**.
- Points where the von mises stress is maximum or minimum is plotted on the geometry with the value.



iv) x-displacement :

- Click on the **Postprocessing** menu and choose **Plot Parameter**.
- Click on the **Surface** tab and in the **Predefined quantities** edit field choose ***x*-displacement**. Make sure that the Surface plot box is marked.
- Click on the **Apply** button.
- Click on the **Max/Min** tab and in the **Predefined quantities** edit field choose ***x*-displacement**. Make sure that the **Max/min marker** box is marked.
- Click on the **Apply** button and then **OK** button to close the dialog box.
- The result is shown below.
- The point where ***x*-displacement** is maximum is plotted on the geometry with its value.
- The y-displacement can be obtained by changing the **Predefined quantities** edit field to **y-displacement**.

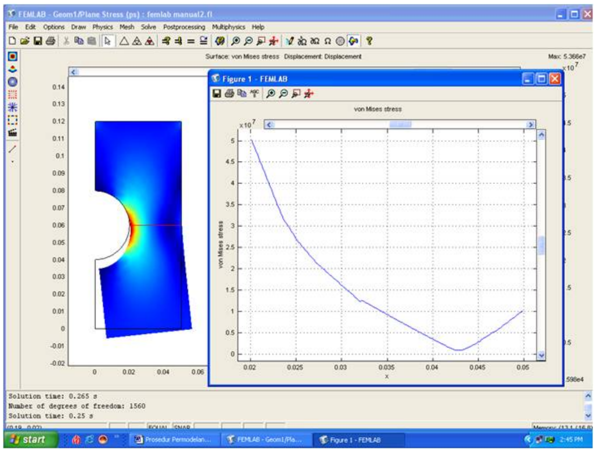


v) Cross section plot of Von Mises Stress:

To see the von mises stress on a cross section, Untuk melihat von mises stress pada keratan struktur, paparan keputusan perlu di ubah kepada paparan von mises stress kembali.

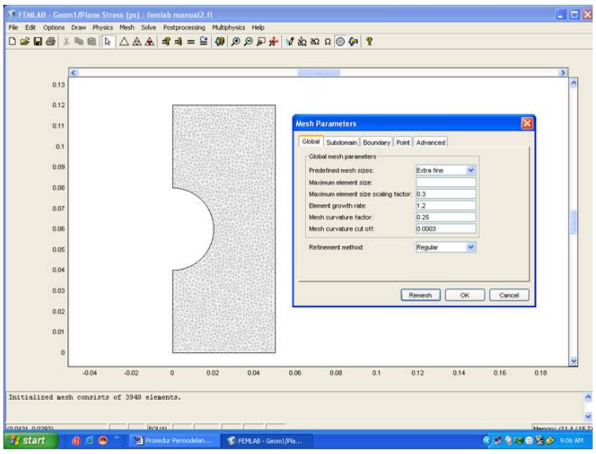
- Click on the **Postprocessing** menu and choose **Plot Parameters**.
- Click on the **Surface** tab.
- Choose von mises stress as **Predefined quantities** and click **Apply**.
- Then click on the **Max/Min** tab and click on the **Max/min marker** box to activate.
- Click on the **Apply** button and after that **OK** button.
- After that, in the **Postprocessing** menu, choose **Cross Section Plot Parameters**.
- Click on the General tab. In the **Plot type** edit field, make sure that the **Line/Extrusion** plot circle is marked.
- Click on the **Line/Extrusion** tab.
- **Predefined quantities** is von mises stress and **x-axis data** is x.

- In the **Cross-section line data** edit field, set x0=0.02, y0=0.06, x1=0.05, y1=0.06.
- Click on the **Apply** button and then OK button.
- The result is shown below.

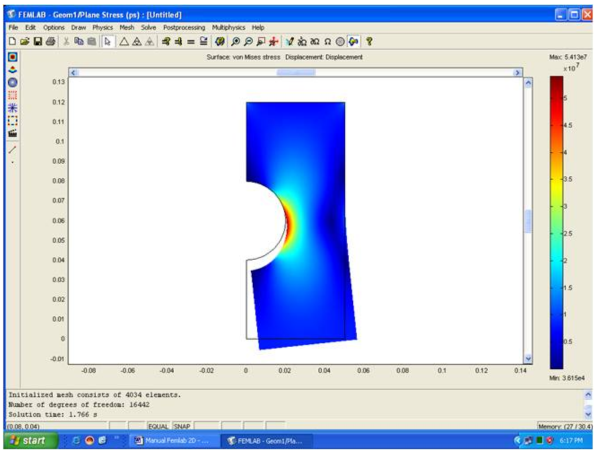


The result is obtained using normal mesh size. Now, we will reduce the mesh size to compare the result.

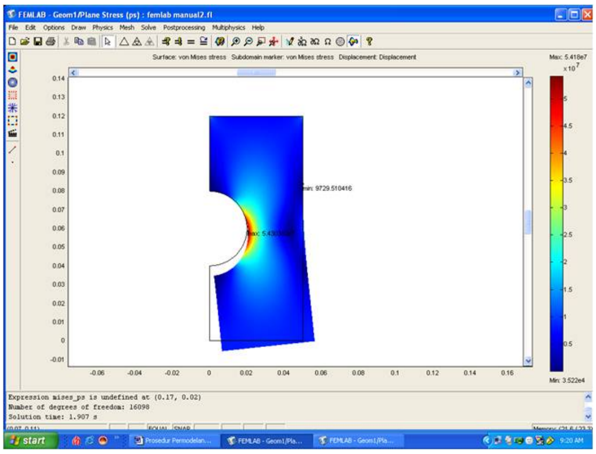
- Click on the **Mesh** menu and choose **Mesh Parameters**.
- In the **Predefined mesh sizes**, choose **Extra fine** and click on the **Remesh** button.



- Click on the **OK** button
- Run the analysis by clicking on the **Solve** button placed at the top toolbar.
- Note that the analysis will take longer time compared by using normal mesh size.



- Now, click on the **Postprocessing** menu and choose **Plot Parameters**.
- Click on the **Max/Min** tab and the **Predefined quantities** is von mises stress. Then, click on the **Max/min marker** box to activate.
- Click on the **Apply** button and then **OK** button.
- The result is shown below.



- It can be noticed that the von mises stress value is different compared by using normal mesh size.
- Now, using **Extra fine** mesh size, repeat steps above for x and y displacement to compare the result.