KING FAHD UNIVERSITY OF PETROLEUM AND MINERALS

MECHANICAL ENGINEERING DEPARTMENT

SEMESTER (031)

ME-498

(FEA in Mechanical Engineering)

Final Project Report

Due 14-Jan 2004

Prepared For

Dr. Arif

By

Ayman Abdallah

973610

Sec # 01

Table of contents

Introduction

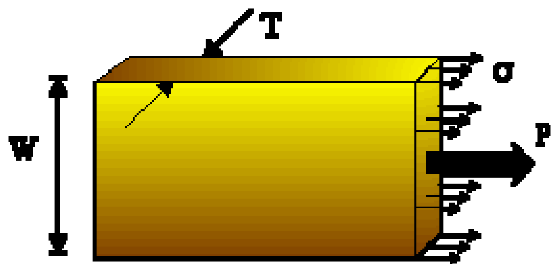
Basic stress analysis calculations assume that the components are smooth, have a uniform section and no irregularities. But in practice virtually all engineering components have to have changes in section and/or shape. Common examples are shoulders on shafts, oil holes, key ways and screw threads. Any discontinuity changes the stress distribution in the vicinity of the discontinuity, so that the basic stress analysis equations no longer apply. Such discontinuities cause local increase of stress referred to as 'stress concentration'. So that we can say stress concentrations arise from any abrupt change in the geometry of a specimen under loading. As a result, the stress distribution is not uniform throughout the cross-section.

The maximum local stress, σ_{max}, normally occurs at these discontinuities. This maximum local stress is many times greater than the nominal stress, σ_{nom}, of the member. Thus, the discontinuities cause areas of stress concentration within the component, and are often called "stress raisers". In ideally elastic members, the ratio of the maximum stress to the nominal stress is designated, the theoretical stress concentration factor, K. The nominal stress is calculated for a particular loading and net cross section assuming a stress distribution across the section that would be obtained for a uniform geometry. The theoretical stress concentration factor is solely dependent on the geometry and the mode of loading. The maximum stress present in a component can be determined by multiplying the nominal stress by the appropriate theoretical stress concentration factor:

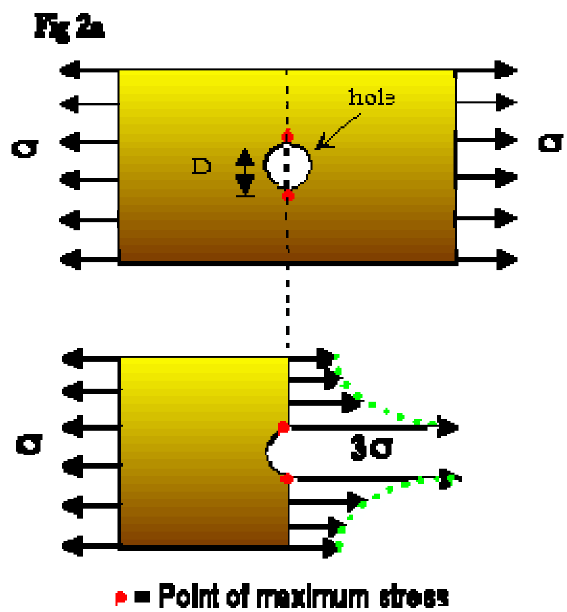
$$\sigma_{max} = K * \sigma_{nom}$$

Background about Stress Concentration Factor (SCF)

The uniform plate shown in the Figure is subjected to a tensile force **P**. The stress **σ** in the plate is uniform everywhere with: (**σ= P/WT)**



It is often necessary to drill a hole in the plate. When the load **P** is applied, the presence of the hole disturbs the uniform stresses in the plate. The profile of the stress at the section through the center of the hole has the form shown in figure below when the diameter **D** is small by c son with the width **W**.



Notice the maximum stress **σ_{max}** is **3σ** and occurs at the edge of the hole In the expression (**σ_{max}** = **3σ)**. The factor **3** is known as the **stress concentration factor (SCF).**

Failures such as fatigue brittle cracking and plastic deformation frequently occur at points of stress concentration. It is for this reason that stress concentration factors play an important role in design. The value of **SCF**s depend on the shape and dimensions of the component being designed and can be calculated using finite element methods, but there are many examples where the equations are known and the values can be easily calculated.

Problem Description

The problem is to design and develop charts to determine the stress concentration factors for two types of plates with triangular types of holes.

General FEA Analysis Approach

For ***Analysis Type Selection*** you need to determine the kind of analysis necessary to predict your physical model's behavior (Structural, Thermal, Magnetic, Electric, Flow or Coupled-Field). In the case of structural analysis you need to decide if quasi-static load (static) or dynamic load (vibration) is present.

Element selection

The relative dimension of the components should be used to determine if line, surface or volume elements would be used. If two of the dimensions are much less than the third (15 to 20 times) line elements are recommenced. These elements can be either 2D (i.e. plane frames) or 3D (i.e. space trusses). If only one of the dimensions is much less than the other two, use surface elements of the appropriate dimension (2D for plane stress & plane strain and 3D for shell elements). If both the geometry and the loading are axisymmetric then use axisymmetric elements. If all dimensions are comparable the volume elements (brick or tetrahedral) should be used. In the event that nonlinearities are present special nonlinear elements may be necessary (i.e. contact tension/compression only, etc)

<u>Unit selection</u>

The key is to have *CONSISTENT UNITS.* Select the primary units for Force, Length, Temperature, Time & Angle and make sure that the secondary are consistent. For example are the section and span units the same? Is your density mass or weight? The number one reason for wrong models is inconsistent units.

<u>Boundary condition and loading selection</u>

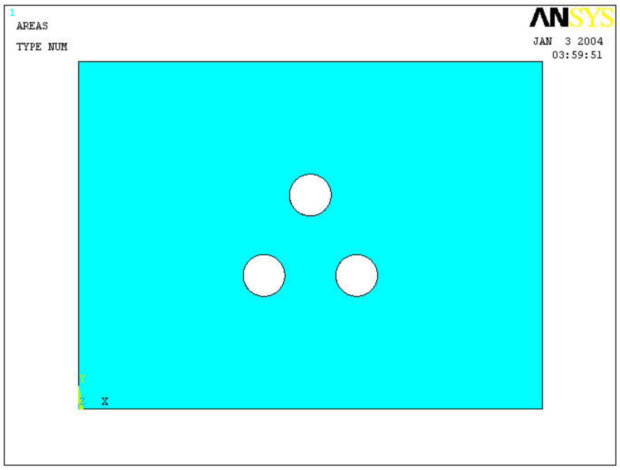
make sure that you consider ALL load cases by examining the construction sequence loads (element birth and death technique), the assembly loading (point, line, contact), the moving loads (influence lines), life time loading (creep), the reference temperature, etc.

<u>The verification of results</u>

You must estimate the results before you begin with a simpler approximate solution. Don't forget that without FEA, we built airplanes, ships, the Eiffel Tower, the Empire State building, etc. Your approximate solution must be in the same order of magnitude as the FEA solution. Also perform the *sanity checks*, like does ... the gravity pull downward? a spinning object moves radially outward? a heated object grows? a bending load generates compression and tension on the right sides? does your axisymmetric model have any hoop stress? You also need to check global equilibrium. And answer the following questions: Is the sum of reactions equal to the applied and inertia loads? Is your mesh density sufficient? What is your Discretization error? (% error in energy norm < 10%) Is buckling possible? Do the axial loads influence the stiffness? Is the behavior consistent with assumptions? Does the magnitude of the displacements require nonlinear analysis? (max □ ~ t) Is the maximum stress less than yield? Does contact occur? Did the solution converge (if appropriate).

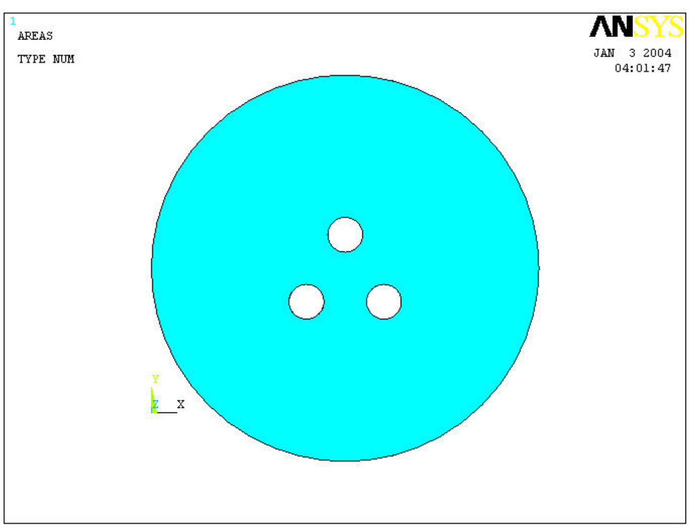
FEA Model for Rectangular Plate with Triangular Pattern hole

The study here involves applying different types of loading on a rectangular plate with triangular pattern holes.



FEA Model for Circular Plate with Triangular Pattern hole

The study here involves applying different types of loading on a circular plate with triangular pattern holes.



Solution Procedure

As a start I am going to implement a uniaxial load on a rectangular plate with three circular holes and check the results. Also, considering the distance between the centers of the circles is constant and equal to l and every time the radius will be changed to different d/l.

Uniaxial Load

Geometrical Modeling for Uniaxial Load on Rectangular Plate

1) Creating rectangular area with dimensions 2 m (width) and 1.5 m (height).

2) Creating three circular area with different radius ranging from 0 m 0.2 m, and noting that the distance between the centers of the circles is constant and equal to l=0.4 m.

3) Now subtracting the circle from the rectangle to get a plate with hole.

4) Specifying the element type as structural solid and choosing the element Quad 8node 82 and considering plane stress analysis.

5) Selecting linear elastic isotropic material such as Aluminum with (EX= 73 GPa, PRXY= 0.35).

6) A load of 1000 N/m^2 is applied on the right side of model.

7) Meshing the whole area.

8) Defining Static analysis type.

9) Applying constrain of the left side of the plate in horizontal direction (i.e. U_x=0).

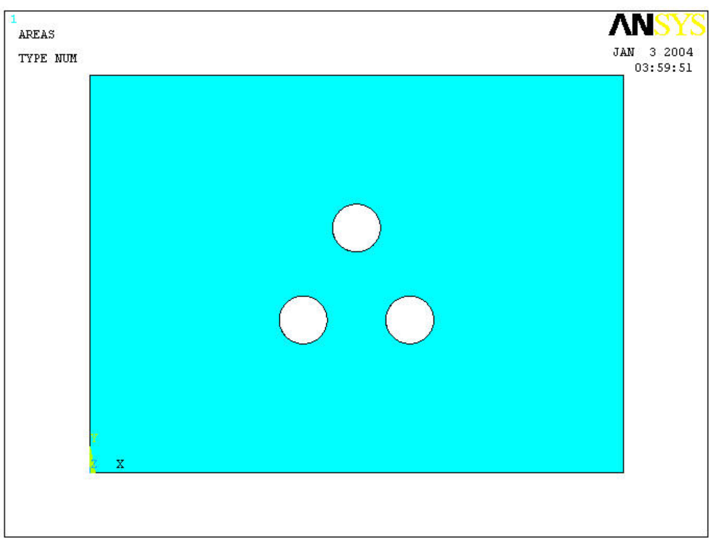
10) At the side also at the middle we put also U_y=0 to avoid moving in the vertical direction.

11) Solving now for nodal and stress distribution in the x-direction.

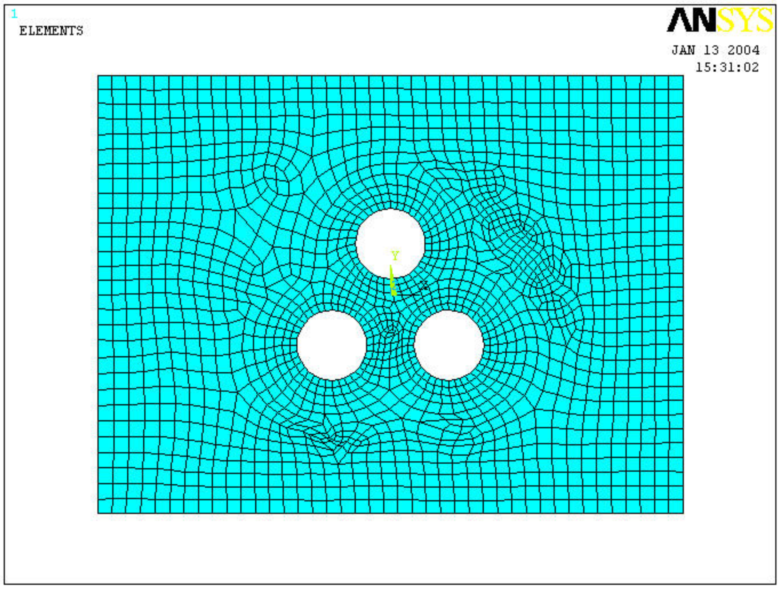
Since the stress concentration factor is dependent purely on the geometry, I am going o change the radius of the hole every time and keep the rest unchanged. After that, finding the maximum stress in the plate which is usually around the holes. Then, the stress concentration factor is calculated from:

$$K = \frac{\sigma_{\mathbf{max}}}{\sigma_{\mathrm{nom}}}$$

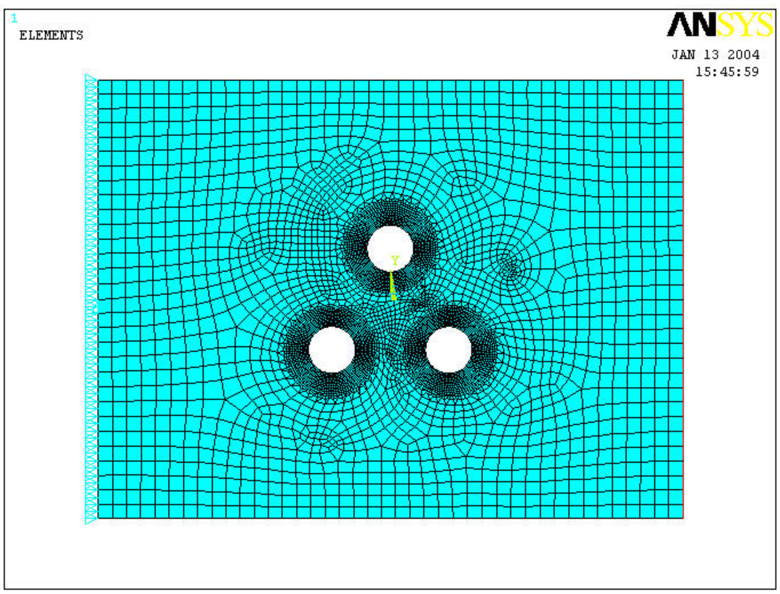
Example of Geometrical model



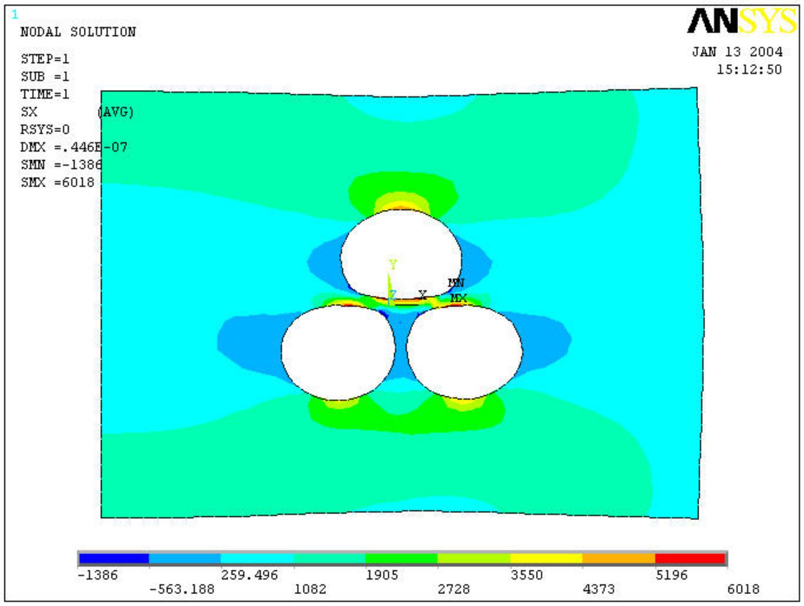
Example of FE mesh



Example of FE mesh after refining



Examples of Stress in the x-direction:



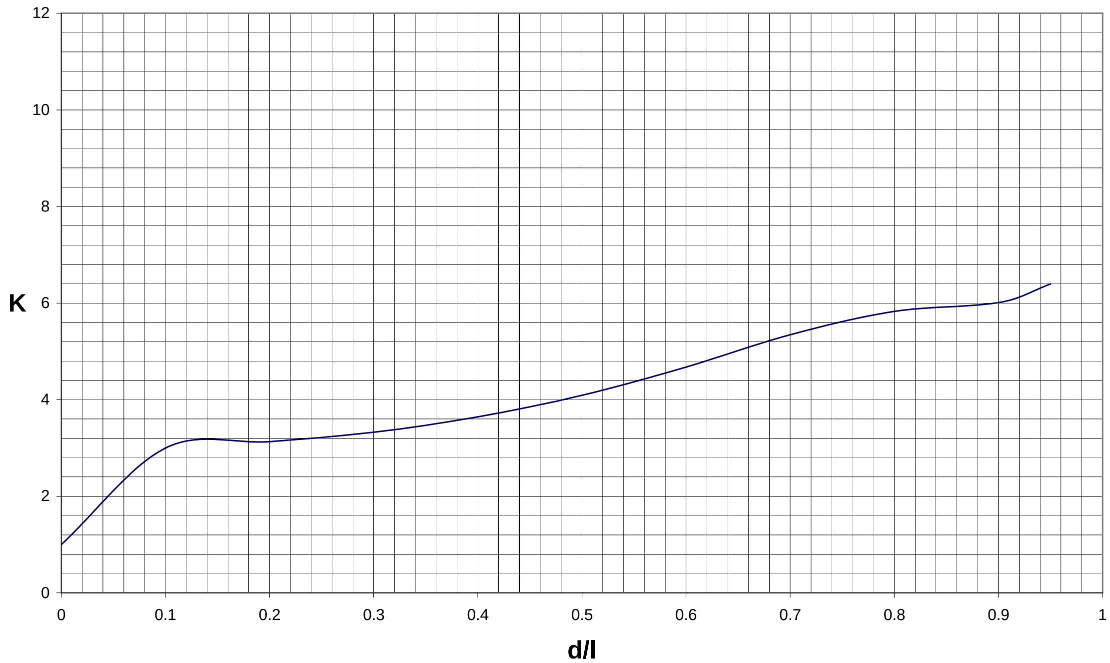
<u>Stress Concentration Factor Results</u>

This is the tabulated data and plot of d/l versus K_x:

Where: d/l is the diameter over the distance of the centers of the holes (i.e. d/l=d/0.4).

K_x is stress concentration factor.

d/l	r	σmax	Kx
0	0	1000	1
0.1	0.02	2999	2.999
0.2	0.04	3130	3.13
0.3	0.06	3327	3.327
0.4	0.08	3642	3.642
0.5	0.1	4089	4.089
0.6	0.12	4675	4.675
0.7	0.14	5343	5.343
0.8	0.16	5824	5.824
0.9	0.18	6009	6.009
0.95	0.19	6397	6.397


12
10
8
6
4
2
0
K
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1
d/l

Biaxial Load

Geometrical Modeling for biaxial Load on Rectangular Plate

1) Creating rectangular area with dimensions 2 m (width) and 2 m (height).

2) Creating three circular area with different radius ranging from 0 m 0.2 m, and noting that the distance between the centers of the circles is constant and equal to l=0.4 m.

3) Now subtracting the circle from the rectangle to get a plate with hole.

4) Specifying the element type as structural solid and choosing the element Quad 8node 82 and considering plane stress analysis.

5) Selecting linear elastic isotropic material such as Aluminum with (EX= 73 GPa, PRXY= 0.35).

6) A load of 1000 N/m^2 is applied on the right side of model.

7) A load of 500 N/m^2 is applied on the top and bottom side of model.

8) Meshing the whole area.

9) Defining Static analysis type.

10) Applying constrain of the left side of the plate in horizontal direction (i.e. U_x=0).

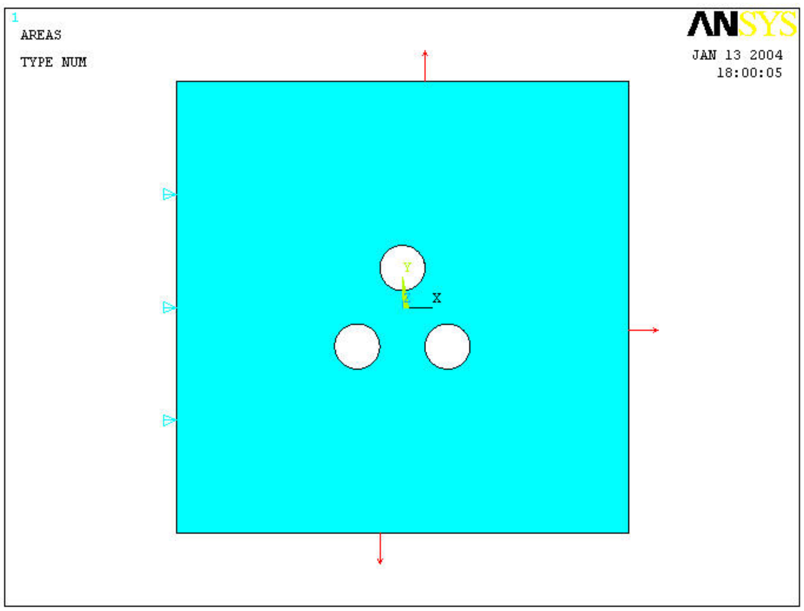
11) At the side also at the middle we put also U_y=0 to avoid moving in the vertical direction.

12) Solving now for nodal and stress distribution in the x-direction.

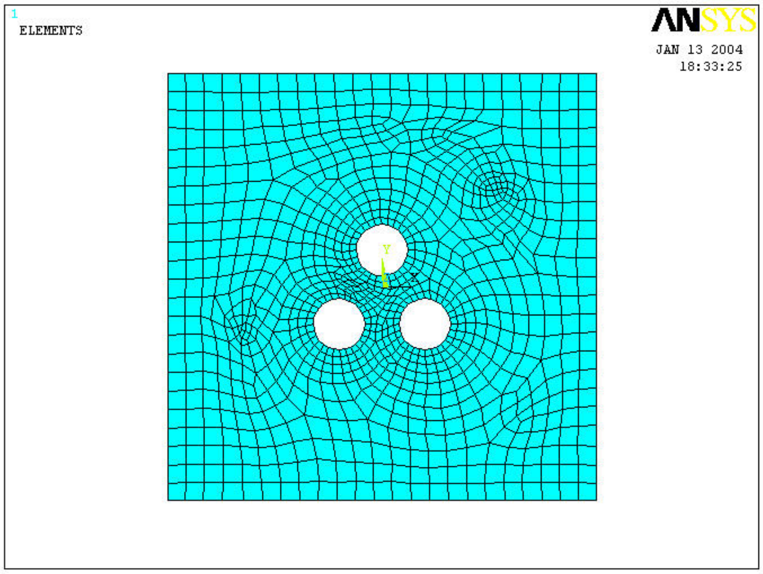
Since the stress concentration factor is dependent purely on the geometry, I am going o change the radius of the hole every time and keep the rest unchanged. After that, finding the maximum stress in the plate which is usually around the holes. Then, the stress concentration factor is calculated from:

$$K = \frac{\sigma_{\mathbf{max}}}{\sigma_{\mathrm{nom}}}$$

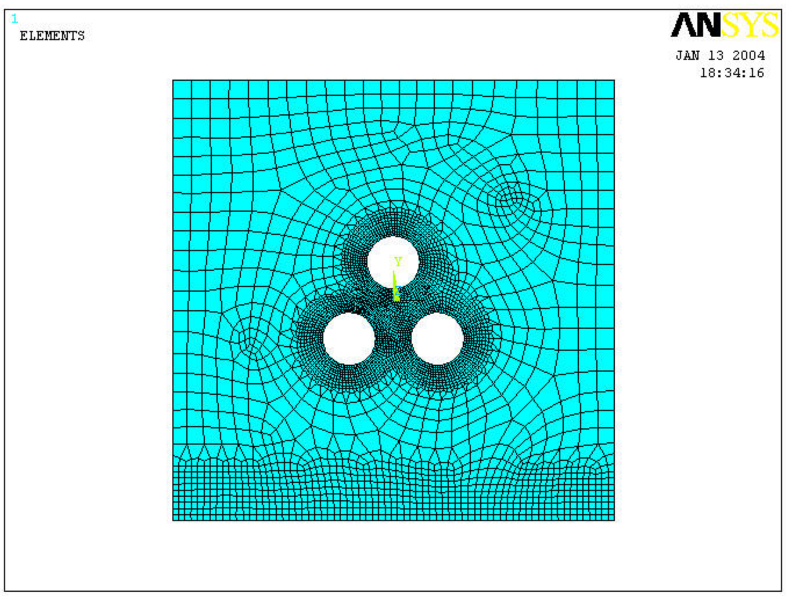
Example of Geometrical model



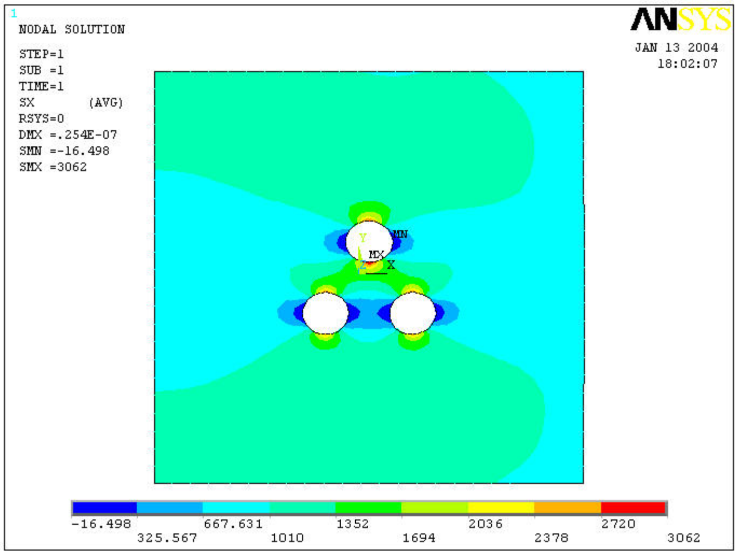
Example of FE mesh



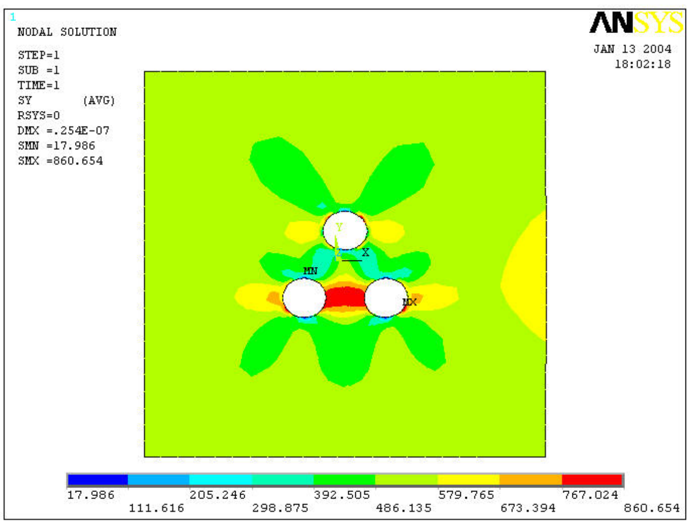
Example of FE mesh after refining



Examples of Stress in the x-direction:



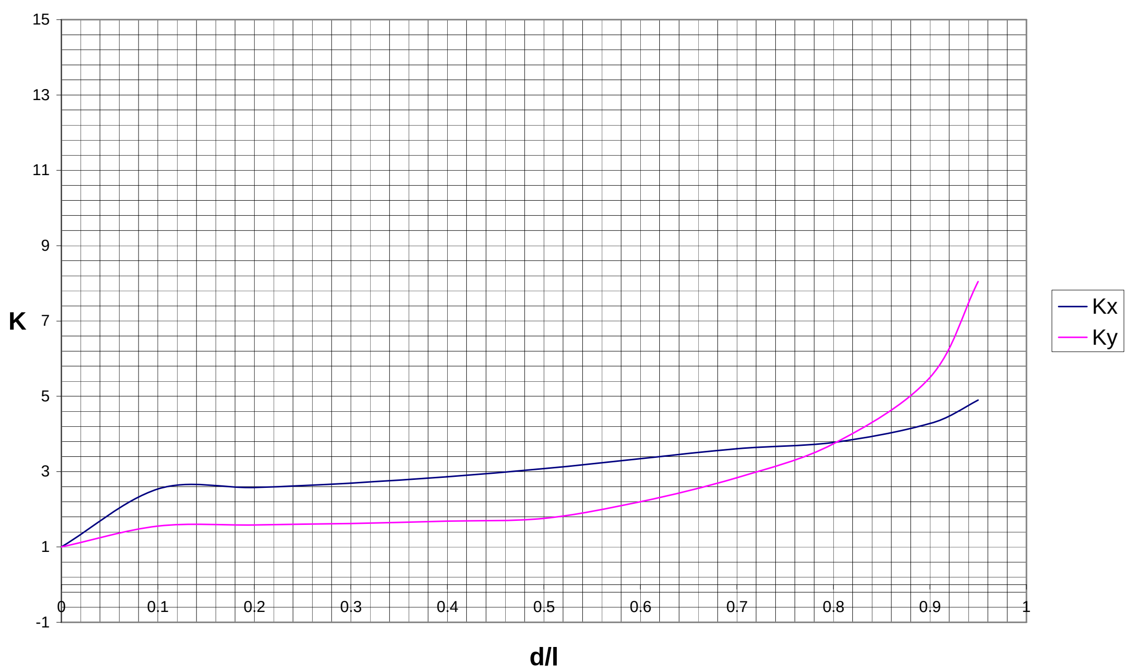
Examples of Stress in the y-direction:



Stress Concentration Factor Results

This is the tabulated data and plot of d/l versus K:

d/l	r	σx	Kx	σy	Ky
0	0	1000	1	500	1
0.1	0.02	2541	2.541	777	1.554
0.2	0.04	2581	2.581	791	1.582
0.3	0.06	2696	2.696	811	1.622
0.4	0.08	2864	2.864	841	1.682
0.5	0.1	3082	3.082	881	1.762
0.6	0.12	3344	3.344	1100	2.2
0.7	0.14	3608	3.608	1419	2.838
0.8	0.16	3776	3.776	1870	3.74
0.9	0.18	4276	4.276	2749	5.498
0.95	0.19	4902	4.902	4023	8.046



Radial Load

Geometrical Modeling for Radial Load on Circular Plate

1) Creating circular area with dimensions 1 m (radius).

2) Creating three circular area with different radius ranging from 0 m 0.2 m, and noting that the distance between the centers of the circles is constant and equal to l=0.4 m.

3) Now subtracting the small circles from the big one to get a plate with holes.

4) Specifying the element type as structural solid and choosing the element Quad 8node 82 and considering plane stress analysis.

5) Selecting linear elastic isotropic material such as Aluminum with (EX= 73 GPa, PRXY= 0.35).

6) A load of 1000 N/m^2 is applied radially on the surface of model.

7) Meshing the whole area.

8) Defining Static analysis type.

9) Applying constrain of the left side of the plate in horizontal direction (i.e. U_x=0).

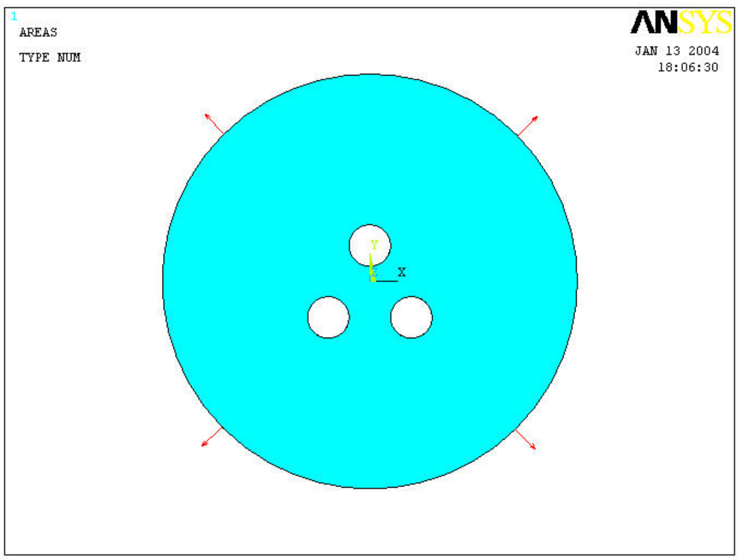
10) At the side also at the middle we put also U_y=0 to avoid moving in the vertical direction.

11) Solving now for nodal and stress distribution in the x-direction.

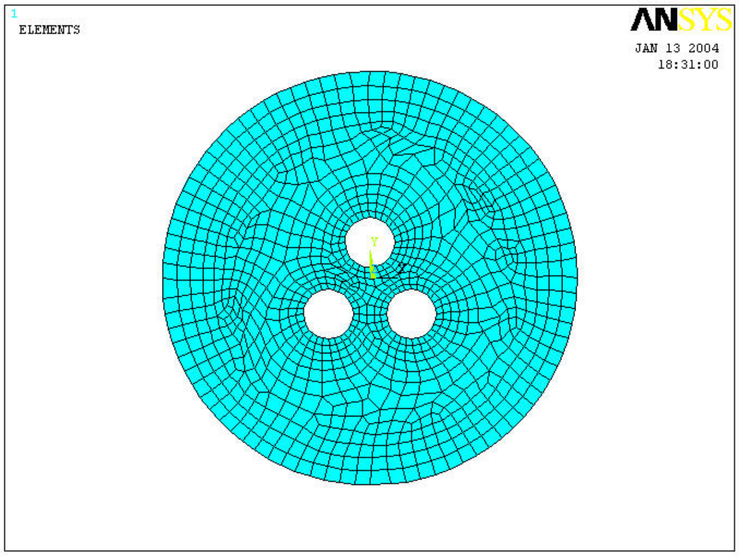
Since the stress concentration factor is dependent purely on the geometry, I am going o change the radius of the hole every time and keep the rest unchanged. After that, finding the maximum stress in the plate which is usually around the holes. Then, the stress concentration factor is calculated from:

$$K = \frac{\sigma_{\mathbf{max}}}{\sigma_{\mathrm{nom}}}$$

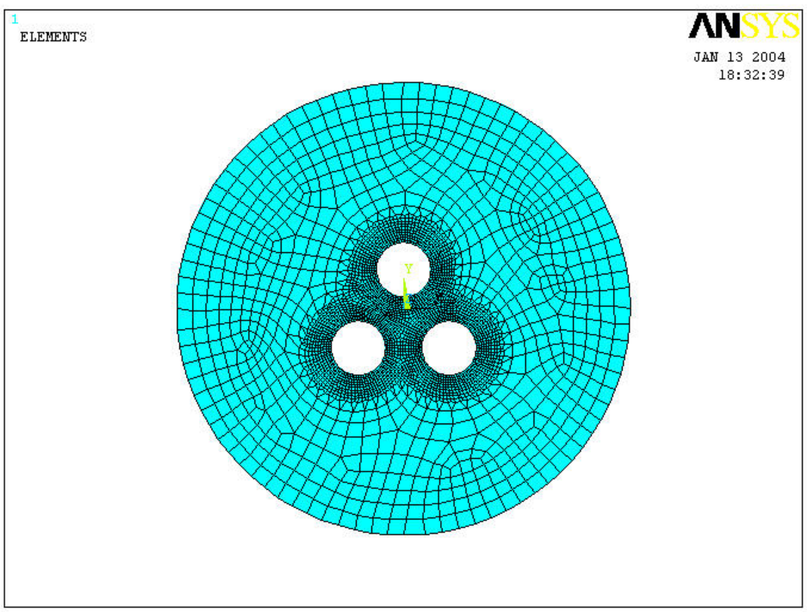
Example of Geometrical model



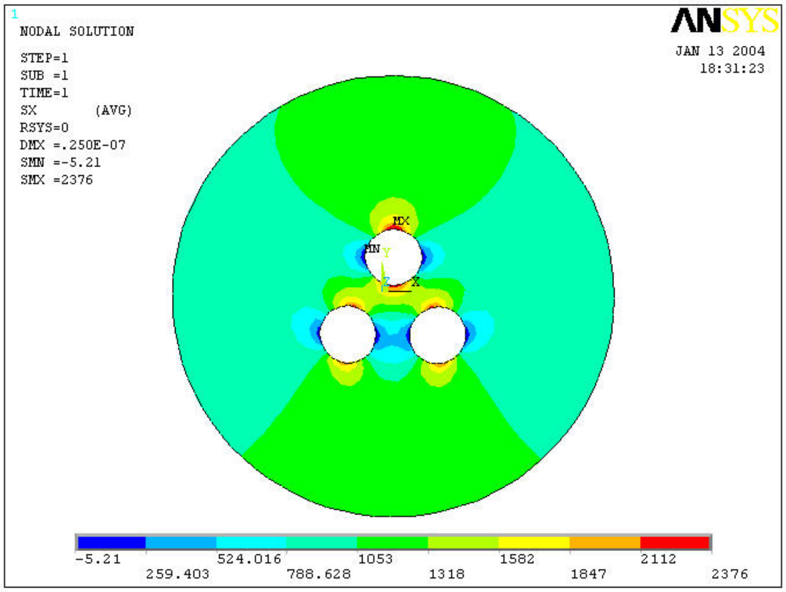
Example of FE mesh



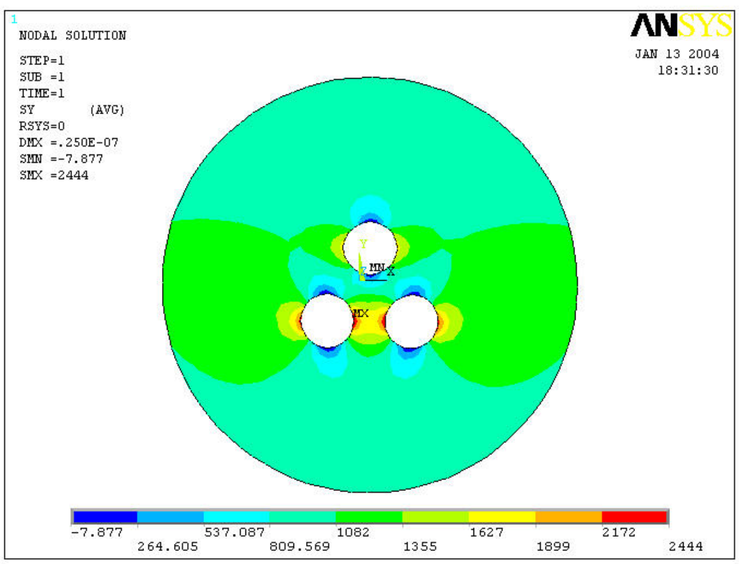
Example of FE mesh after refining



Examples of Stress in the x-direction:



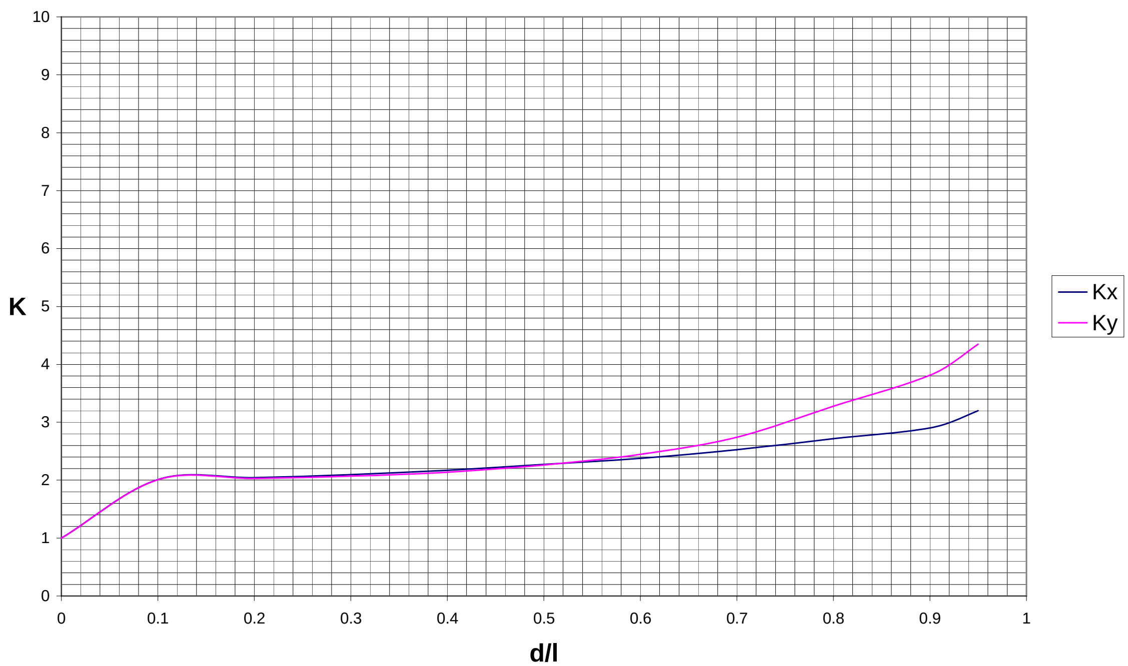
Examples of Stress in the y-direction:



Stress Concentration Factor Results

This is the tabulated data and plot of d/l versus K:

d/l	r	σx	Kx	σy	Ky
0	0	1000	1	1000	1
0.1	0.02	2011	2.011	2007	2.007
0.2	0.04	2045	2.045	2030	2.03
0.3	0.06	2095	2.095	2071	2.071
0.4	0.08	2172	2.172	2139	2.139
0.5	0.1	2271	2.271	2261	2.261
0.6	0.12	2376	2.376	2444	2.444
0.7	0.14	2527	2.527	2741	2.741
0.8	0.16	2715	2.715	3277	3.277
0.9	0.18	2903	2.903	3813	3.813
0.95	0.19	3200	3.2	4349	4.349



Conclusion

It is clear that from the graphs that that stress concentration factor is inversely proportional to cross sectional area at which the load is applied. And from these graphs we can determine the maximum stress in any general plate provided that given the ratio between the diameter of hole in the plate over the distance between the centers of the holes.

References

1) Hibbeler R. C., "Mechanics of Materials," Prentice Hall, 3rd edition, New Jersey, 1997.
2) Instructor's handouts.
3) Ansys manuals.