UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH

Practice Assignment no. 3

Author: Xavier Corbella & Waleed Ahmad Mirza

Polytechnic University of Catalonia

April 21, 2016

Contents

1. Clamped plate with uniform load

1.1 Introduction

In structural mechanics, flat plates are considered as thin flat structures with thickness much smaller than the other two dimensions and are subjected to transverse loading. The loads are resisted mainly by bending deflections. Such structures have wide range of applications such as in roofs and ceilings.

1.2 Preprocessing

The given square plate as shown in fig. 1.1 fixed on all sides. The sketch of the plate is built suing dimensions from figure 1.1

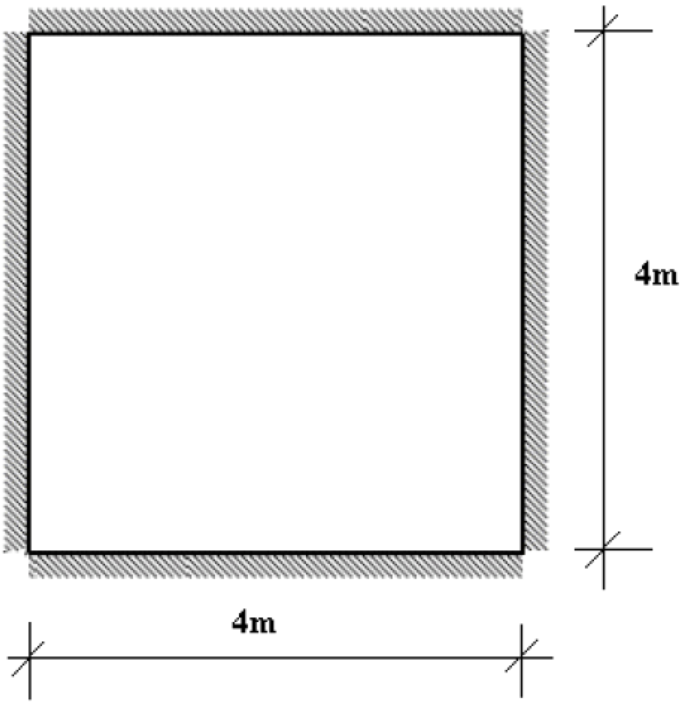

Figure 1.0.1 Problem description of plate analysis

A fixed supported condition means that the rotation vector tangent to the segment is also zero. The plate is subjected to a uniform transverse pressure load of $1e4\,{}^{N}/_{m}$.

The analysis is carried out using two types of elements i.e.

1. Reduced integration 6 node Reissner-Mindlin element
2. Triangular plate DKT element
3. 4 node quadrilateral CLLL element

The study is done at varying mesh size to find the converged mesh size for each element. Linear isotropic material model with Young modulus $E = 3e10\,{}^{N}/_{m^2}$ and poison ratio $v = 0.2$ is assigned to the plate. Static analysis involving non-linear geometric deformation is carried out.

Linear Static analysis is defined in solver definition. Results of this study is illustrated in the next section.

1.3 Post processing

Results of this study are in terms of vertical displacement w and von-misses stress S_{von}. Mesh convergence study is carried out by reducing the element size until convergence tolerance of $1e-6$ m is reached. Results for each element are described in figure 1.2.

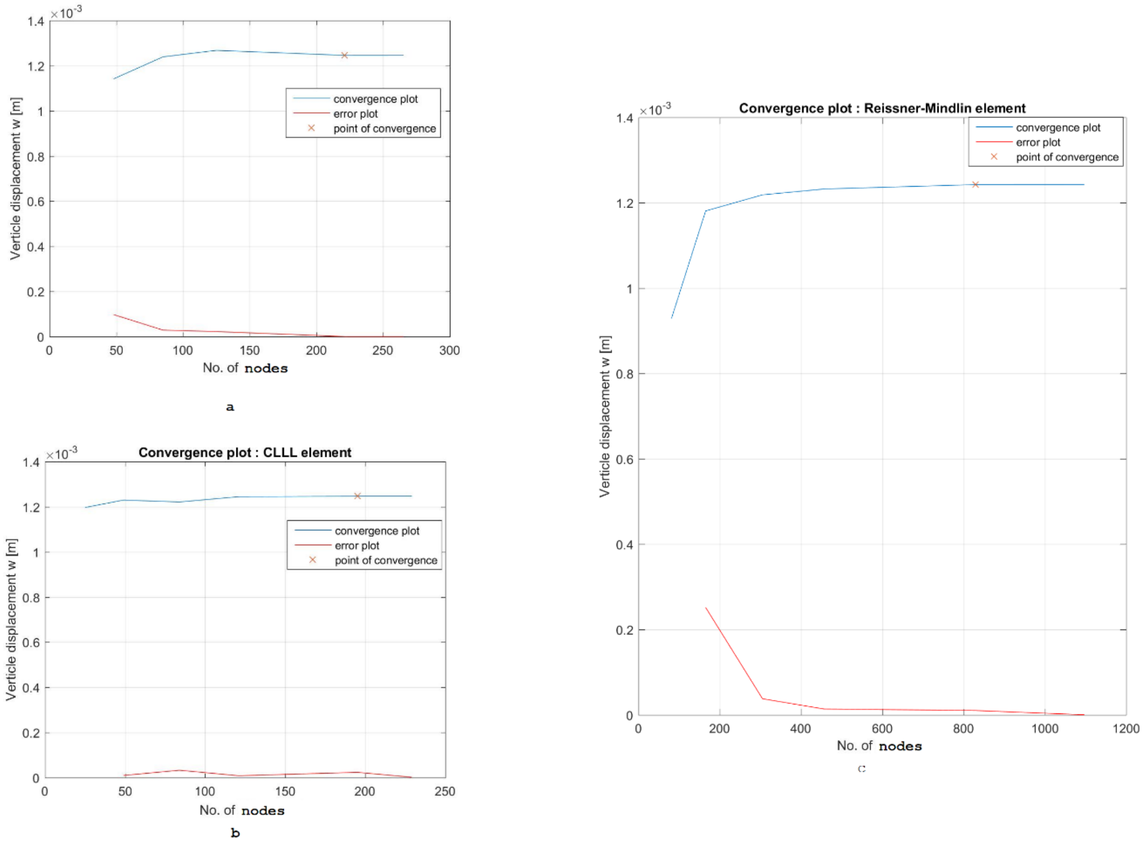

Figure 1.2 Convergence test a) DKT element b) CLLL element c) RM element

It can be seen that CLLL element, DKT element and RM element converge at 191, 225 and 825 nodes respectively. Hence the convergence rate of CLLL element is the fastest.

From here onward the results are displayed for converged mesh size of CLLL element. Vertical deflection contours for CLLL element is shown in figure 1.3. It serves to verify that the restraints were properly applied.

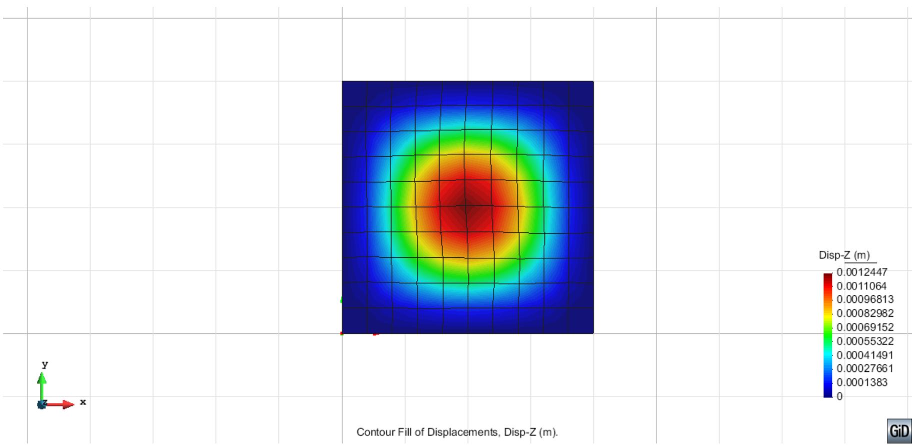

Figure 1.3 Vertical deflection of CLLL element

The center point deflection can also be compared to analytic estimates [1]. The differantial equation that govern the behavior of plate is as follow.

$$\nabla^4\left(D.W\ \ x,y\ \right) = p(x,y) \tag{1}$$

Where,

$$D = \frac{Eh^2}{12(1-v^2)}$$

Solution of equation 1 is subjected to the following boundary condition along all the edges.

$$W = 0\ ,\frac{\partial W}{\partial n} = 0$$

As a result following solution is obtained for the center point.

$$W_{\substack{x=0\\y=0}}\ \alpha\frac{pb^4}{D}$$

For a clamped plate the value of $\alpha = 0.00126$. The analytical result turn up to be $0.001238630m$. The comparison of analytical solution with numerical solution obtained from three elements for converged mesh size is illustrated in table 1.

Element type	**Numerical result (m)**	**Percentage relative error**
CLLL element	0.0012447	0.4%
RM element	0.0012429	0.3%
DKT element	0.0012460	0.6%

Table 1 Comparison of analytical and numerical solution

Moreover, here the maximum computed deflection is less than half the thickness of the plate, therefore use of linear elastic analysis is justified. For stresses the results are shown in terms of von Mises stress. Von Mises effective stress is proportional to the square root of the sum of the squares of the differences in the principal stresses, so it is always positive as shown in fig 1.4 (**Note that the displayed contour is simulated in ABAQUS using shell elements as plate elements are not absent in ABAQUS element library. Moreover shell element give quite comparable results with plate element under bending loads.**).

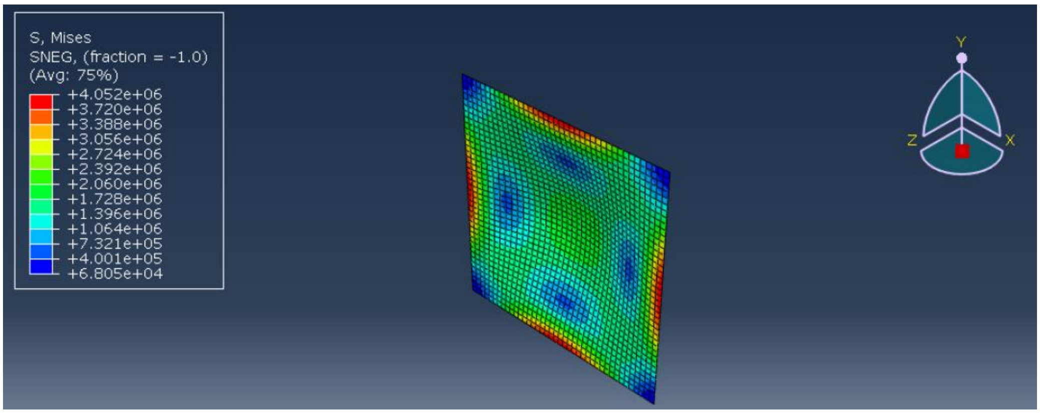

Figure 1.4 Von Mises stress countour plot

In the shear contour and bending moment diagram in figure 1.5, it can be noticed that the edges are the critical regions.

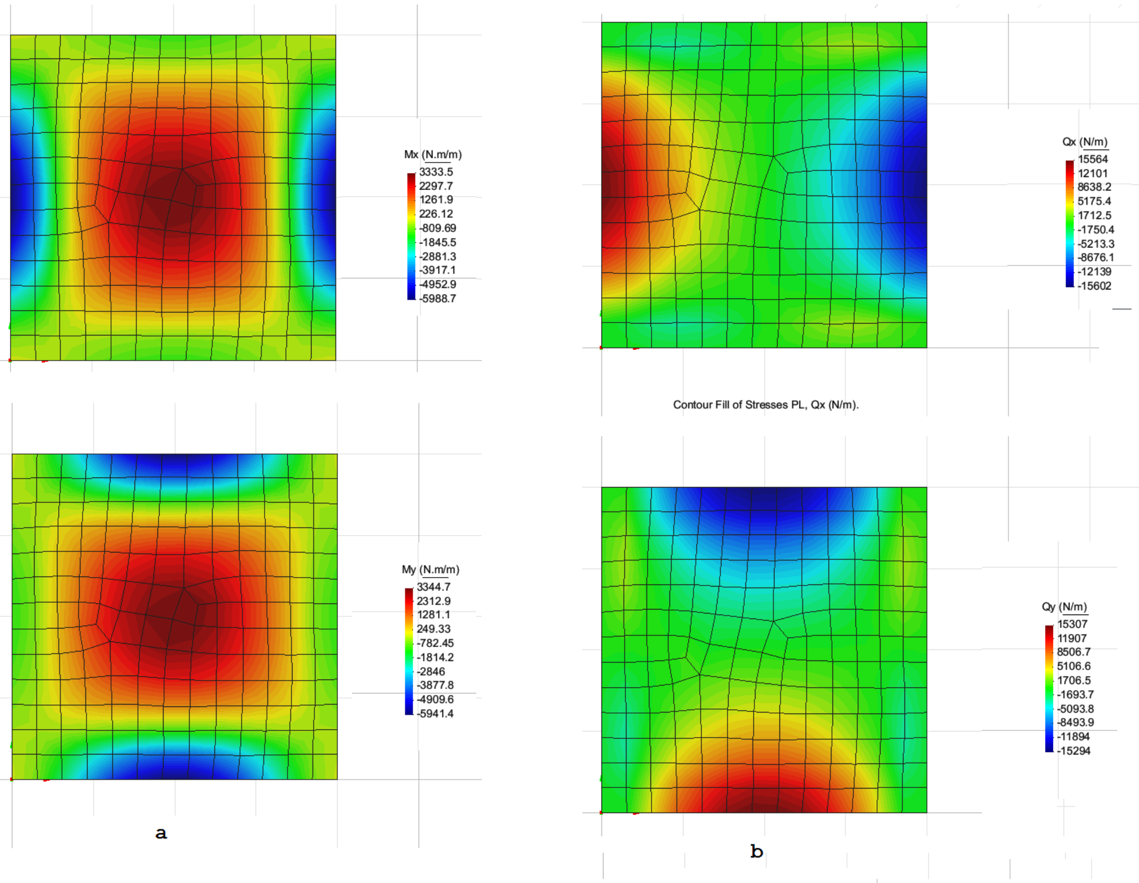

Figure 1.5 a) Bending moment diagram b) shear moment diagram

1.4 Design insight

It can be seen in stress contour plot that critical stress region lie along the plate edge and it should be made sure that peak stress values should not exceed the yield strength of the material on this region. In case of concrete the the tensile strength is usally between 3-5MPa and the estimated peak von Mises stress is 4.02MPa. Hence it can be concluded that the given plate structure is not within safe limit.

2 Thin plate with internal hole

Given structure is an octahedral thin plate with an internal hole under self weight and unifrom pressure load. Max length to thickness $({}^{L}/_{t})$ ratio of the structure is *40* and thus can be modelle using plate elements.

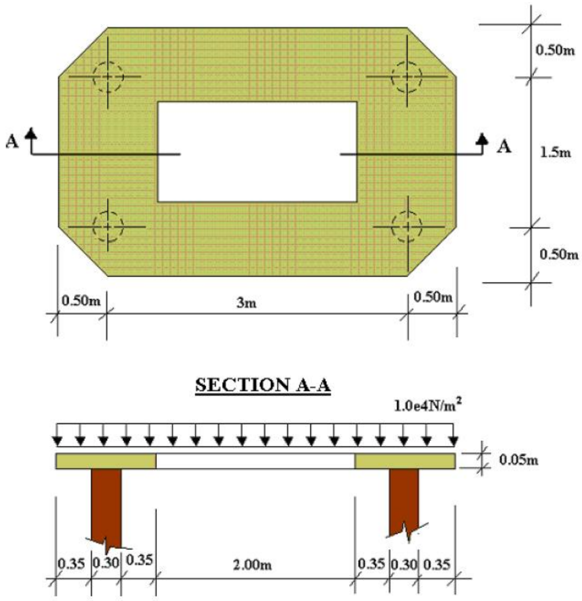

Figure 2.1

2.1 Preprocessing

The CAD model of the given problem is modelled with the geometric dimension given in figure 2.1. Four partitions in form of holes shown in the figure 2.1 are made on the plate surface. Material properties of an isotropic concrete model with Youngs modulus $E = 2.1e^{11}\, {}^{N}/_{m^2}$s, poison ratio $v = 0.30$ and specific gravity $\gamma = 7.80e^{4}\, {}^{N}/_{m^3}$ is assigned to the plate.

A uniform pressure of $1e^{04}\, {}^{N}/_{m^2}$ is assigned to the plate. Moreover body weight is also considered in the analysis. Drichelet boundary condition is applied by assigning $U_y = 0$ displacement to the four partitions made in the CAD model. This BC reflects that that plate is supported with four columns that constrain its vertical movement.

The structure is meshed using triangular DKT element. The study has been conducted for varying mesh size as will be explained in the next section. In solver definition a linear static analysis is assigned for the study.

2.2 Preprocessing

Results of this study are in terms of vertical displacement *w* and von-misses stress S_{von}. Mesh convergence study is carried out by reducing the element size until convergence tolerance of *1e-06 m is reached.* Results of mesh convergence study is shown in figure 2.2. A converged result of vertical displacement *w* is obtained at 997 nodes.

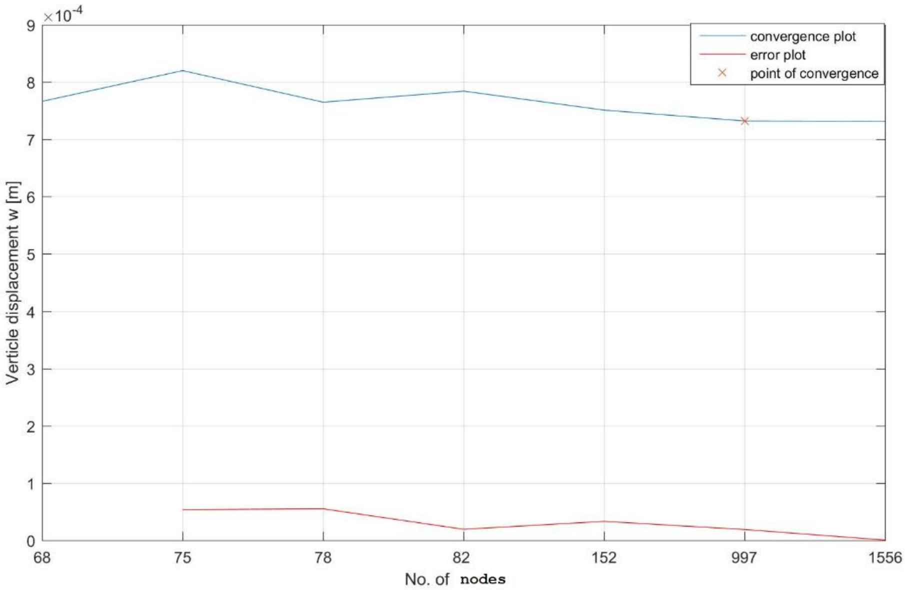

Figure 2.2

Vertical displacement w for converged element size is shown in figure 2.3. Vertical deflection contour plot show that peak deflections are less than half of the thickness. Hence small scale deflection holds valid for this analysis.

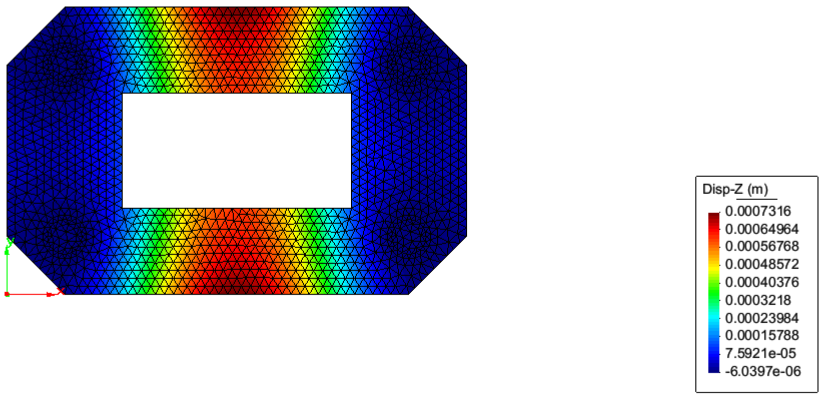

Figure 2.3 Verticle deflection countour plot

In bending moment and shear force contour plot it can be seen that peak bending stress and shear stress exist in the region where Dirichlet BCs are prescribed.

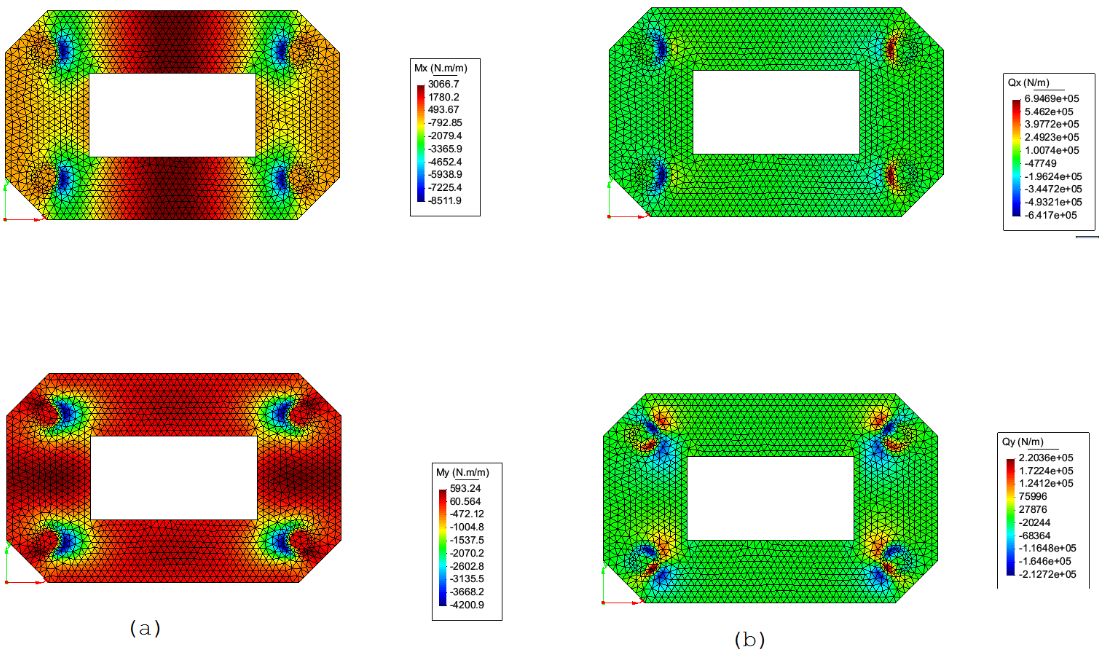

Figure 2.4 a) Bending moment diagram b) Shear force diagram

A similar analysis is performed in ABAQUS to estimate von Mises stress in the circular disk. Results are shown in the figure 2.5. It can be seen that from design point of view that our structure is in safe limit as yield stress of steel (*250e6 MPa*) is much higher than peak von Mises stress.

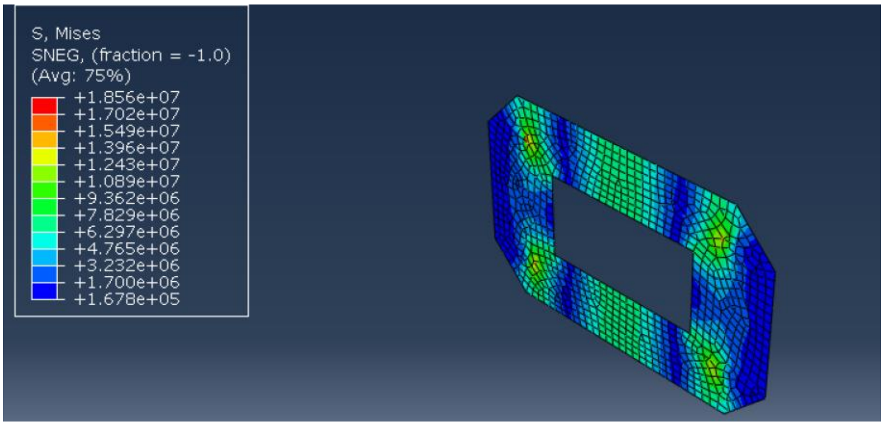

Figure 2.5 Von mises stress contour plot

3 Thick circular plate with internal hole

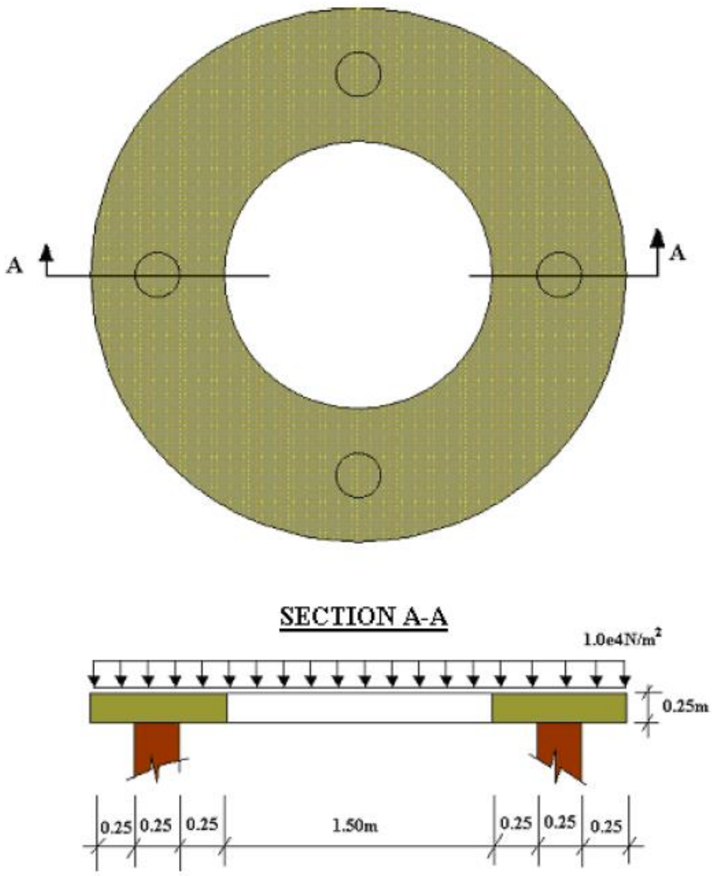

Figure 3.1 Problem description of Question 3

3.1 Preprocessing

The given annulus plate is modelled as a two dimensional structure. The part of sketch with. Four circular partitions are drawn in the model at locations illustrated in figure 3.1.

Material properties of an isotropic concrete model with Youngs modulus $E = 3.0\text{e}10\,{}^{N}/_{m^2}$, Poison ratio $v = 0.20$ and specific gravity $\gamma = 2.4e4\,{}^{N}/_{m^3}$ is assigned to the plate.

A uniform pressure of $1e04\,{}^{N}/_{m^2}$ is assigned to the plate. Moreover body weight is also considered in the analysis. Dirichlet boundary condition is applied by assigning $U_y = 0$ displacement to the four partitions made in the CAD model. This BC reflect that that disk is supported with four columns that constrain its vertical movement.

The structure is meshed using reduced integration 6 node triangular Reissner-Mindlin element. The study has been conducted for varying mesh size as will be explained in the next section. In solver definition a static analysis is carried out.

3.2 Results and discussion

Results of this study are in terms of vertical displacement w and von-misses stress S_{von}. Mesh convergence study is carried out for vertical displacement w convergence tolerance of $1e - 08m$. Results of mesh convergence study is shown in figure 3.2. A converged result of vertical displacement is obtained at 2202 elements.

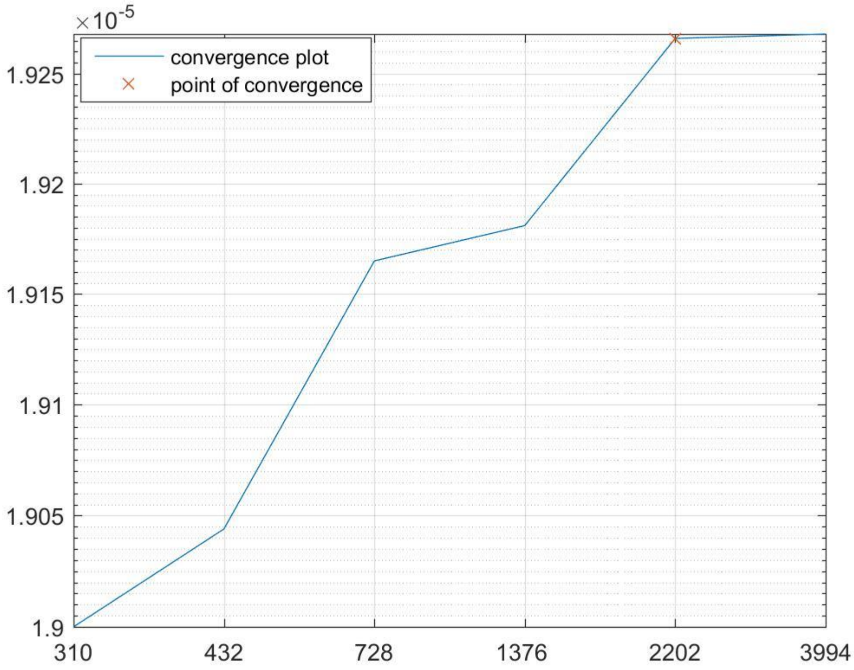

Figure 3.2 Circular plate analysis

A similar analysis is performed in ABAQUS to estimate von Mises stress in the circular disk. Results are shown in the figure 3.3. It can be seen that from design point of view that our structure is in safe limit as tensile yield stress of concrete (3-5 MPa) much is higher than peak von Mises stress.

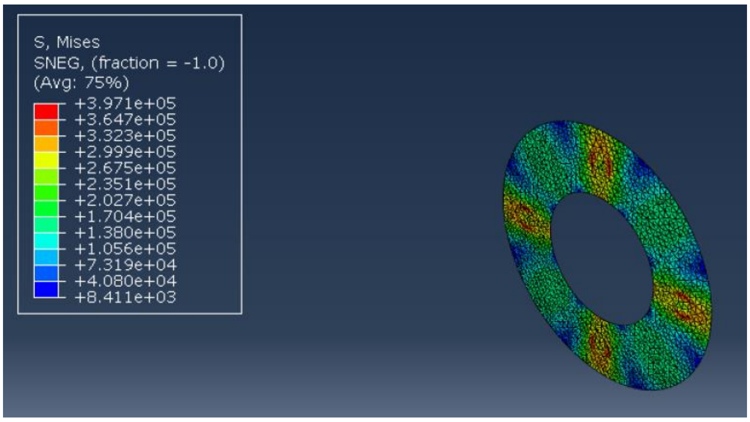

Figure 3.3 von Mises contour

Vertical displacement w, bending moments M and shear force V for converged study is shown in figure 3.3. It can be seen in fig. 3.4 that peak bending stress and shear stress exist in the region where Dirichlet BC are prescribed.

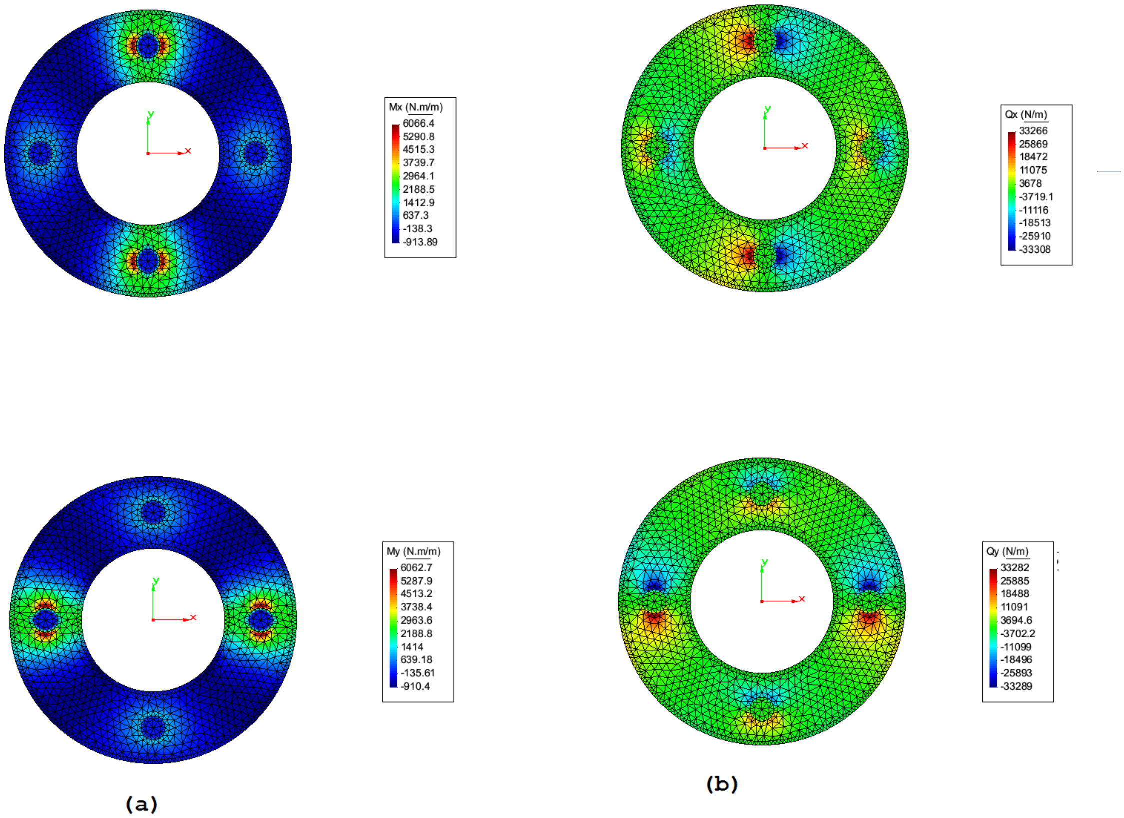

Figure 3.3 a) Bending moment diagram b) shear force diagram

Vertical deflection contour plot show that peak deflections are in order of 10^{-5}. Hence small scale deflection holds valid for this analysis.

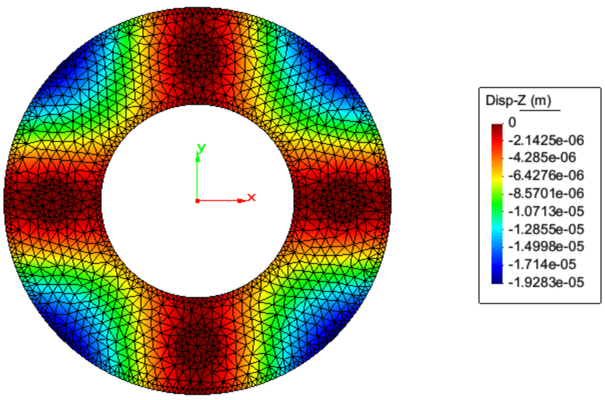

Figure 3.4 Verticle displacement contour plot

References

[1] C.E. İmrak and İ. Gerdemeli (2007), "An exact solution for the deflection of a clamped rectangular plate under uniform load". *Applied Mathematical Sciences,* Vol. 1, 2007, no. 43, 2129 - 2137. Accessed [21-04-2016] at
[https://www.researchgate.net/publication/267078798_An_exact_solution_for_the_deflection_of_a_clamped_rectangular_plate_under_uniform_load]