HOLE SIZE, LOCATION OPTIMIZATION IN A PLATE AND CYLINDRICAL SHELL FOR MINIMUM STRESS POINTS INTERFACING ANSYS AND MATLAB

By

Soundararaj Thangavel

Presented to the Faculty of the Graduate School of

The University of Texas at Arlington in

Partial Fulfillment of the

Requirements

for the Degree of

MASTER OF SCIENCE IN

MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

December 2016

Acknowledgements

I am expressing my sincere gratitude to my advisor Dr. Bo P Wang for his continuous guidance, sharing knowledge and patience and encouraging all these times till the completion of this thesis work. I learnt many good things from him personally and professionally which would help me to excel in future endeavors. I am thankful to committee members Dr. Kent L Lawrence and Dr. Wen S Chan for their valuable inputs and for sharing their time

I am really grateful for the assistant ship offered by The University of Texas at Arlington. I would like to thank my friends for their support and counseling to cheer me up in hard times.

I dedicate this work to my parents and my brother for their sacrifice and with out them nothing would have been possible. I want to thank almighty god for giving me strength to complete my graduate studies successfully

November 17, 2016

Abstract

HOLE SIZE, LOCATION OPTIMIZATION IN A PLATE

AND CYLINDRICAL SHELL FOR MINIMUM

STRESS POINTS INTERFACING

ANSYS AND MATLAB

Soundararaj Thangavel, M.S

The University of Texas at Arlington, 2016

Supervising Professor: Dr. Bo P Wang

Discontinuities in Structures are inevitable. One such discontinuity in a plate and cylindrical shell is presence of a hole / holes. In Plates they are used for mounting bolts where as in Cylinder / Pressure Vessel, they provide provision for mounting Nozzles / Instruments. Location of these holes plays a primary role in minimizing the stress acting with out any external reinforcement. In this Thesis work, Location Parameters are optimized for the presence of one or more holes in a plate and cylindrical shell interfacing ANSYS and MATLAB with boundary constraints based on the geometry. Contour plots are generated for understanding stress distribution and analytical solutions are also discussed for some of the classical problems.

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

Mathematical optimization is the process of maximizing and/or minimizing one or more objectives without violating specified design constraints, by regulating a set of variable parameters that influence both the objectives and the design constraints. It is important to realize that in order to apply mathematical optimization; you need to express the objective(s) and the design constraint(s) as quantitative functions of the variable parameters. These variable parameters are also known as design variables or decision variables.

A general optimization problem can be formulated using the following set of equations

$$min_x \, f(x)$$

Subjected to

$$g(x) \leq 0$$

$$h(x) = 0$$

The function f(x) represents the objective function, which we would like to minimize. The idea being that as we minimize the objective function, the system or design will behave better (e.g., cheaper, stronger, lighter, or faster). The function g(x) represents a vector of inequality constraints and the function h(x) represents a vector of equality constraints. These constraints make the design feasible (i.e., not unacceptable). For example, they might ensure that the mass is not negative or that a process is not prohibitively expensive. They are called behavioral constraints. The vector x represents the vector of design variables. These are the quantities that we can change in the design to improve its behavior. The constraints on the design variables are called side constraints. A set of design variables that fully satisfies all the constraints is called a feasible solution (even if it does not minimize the objective function).

Broadly speaking, optimization problems can be classified along seven major categories. Knowledge of, and appreciation for, these categories is important as they help us understand whether the problem at hand is simple or involved, and whether a given algorithm or software applies to our problem.

The categories are as follows:

1. Linear vs. Nonlinear, 2. Constrained vs. Unconstrained, 3. Discrete vs. Continuous, 4. Single vs. Multiobjective, 5. Single vs. Multiple Minima, 6. Deterministic vs. Nondeterministic, 7. Simple vs. Complex

In the above paragraph, we discussed the various categories, or classes, of optimization problems... In this section, we discuss the various approaches that are available to us to solve optimization problems. Generally speaking, we can identify four broad **solution approaches: analytical, algorithmic, experimental, and graphical.**

In this paragraph, we discuss the various options available for optimizing problems using computer software. As presented in Table 1 below, we define three main classes of optimization software. **The first** class involves stand-*alone* optimization software, where the primary focus is to solve various types of prescribed optimization problems. **The second** class involves design and/or analyses *integration frameworks*, where analyses codes from different engineering disciplines can be conveniently integrated and designs can be optimized. **The third** class of optimization software involves large scale *analyses codes that have optimization capabilities* as one of their offered features – typically added in recent years with the growing popularity of optimization. Details of the above options are discussed next. For convenience, we respectively refer to these classes as (i) Software for Optimization as Stand-Alone (SO-SA), (ii) Software for Optimization within Design Framework (SO-WDF), and (iii) Software for Optimization within Analysis Package (SO-WAP).

Software for Optimization (SO)			
Stand-Alone (SO-SA)	**Within Design Framework (SO-WDF)**	**Within Analysis Package (SO-WAP)**	**Discrete Integer or Mixed**
MATLAB Toolbox	iSIGHT	GENESIS	XPRESS
NEOS Server	PHX ModelCenter	NASTRAN	CPLEX
DOT-VisualDOC	modeFRONTIER	ABAQUS	Excel and Quattro
NAG	XPRESS	Altair	NEOS Server
NPSOL	LINDO/LINGO	ANSYS	MINLP
GRG2	GAMS	COMSOL	GAMS WORLD
LSSOL	Boss Quattro	MS Excel	
CPLEX	What'sBest! RISKOptimizer Busi. Spreadsh.		
BTN			
PhysPro			

Table 1: Optimization softwares

Within this section, we first provide a table (Table 1) that lists many of the popular software for each of these classes in the first three columns. The last column lists that software that performs optimization for problems with discrete, integer, or mixed design variables. We also note that it may be useful to classify optimization software as being (i) small scale or large scale,(ii)easy or difficult to use, or (iii) particularly effective or less so. The choice of particular optimization software will generally depend on pertinent user experience and on the problem under consideration.

1.1 Literature Review

This section deals with the related work done on hole size & location optimization in a plate and pressure vessel cylinder.

Praveen Mirji [1] studied stress distribution for the rectangular plate with two holes under in-plane load and analysis are done numerically with the help of Finite Element Method. The material used for the plate is isotropic in nature. In this paper a method is attempted to reduce the intensity of stress in the vicinity of holes by relocating the position of one of the holes. A rectangular plate with holes having negligible thickness is analyzed. From this analysis author has found the variation in the intensity of stress in the plate for different W/CD ratios. The W/CD ratio considered are1.67, 2.0 and 2.5. Variation in stresses with respect to different hole locations is studied and plotted by graph. The results of reduction in intensity of stresses for different W/CD ratios are tabulated. The finite element formulation is carried out by using the software ANSYS

M. Javed Hyder & M. Asif [2] optimized the location and size of opening (hole) in a pressure vessel cylinder using ANSYS. Analysis is performed for three thick-walled cylinders with internal diameters 20, 25 and 30 cm having 30 cm height and wall thickness of 20 mm. It is observed that as the internal diameter of cylinder increases , the Von Misses stress increases. Optimization of hole size is carried out by making holes having diameter of 4, 8, 10, 12, 14, 16 and 20 mm located at center in each of the three cylinders, and it is observed that initially Von Misses stress decreases and then become constant with hole size. The optimum size of hole is found to be 8 mm for cylinder having internal diameter of 20 cm whereas a hole of size 10 mm for cylinder having internal diameter of 25 cm and 30 cm on the basis of lowest Von Mises stress value. Lastly, optimization of location of hole is carried out by making a 12 mm hole located at 1/16, 1/8, 2/8, 3/8 and 4/8 of cylinder height from top in all the three cylinders. The Von Misses stress is maximum at the center i.e., 4/8 location and decreases in the direction away from center and then stress increases as the location is changed from 1/8 to 1/16 from cylinder top due to the end effects. The optimum location of the hole is found to be at 1/8 of cylinder height.

Ramesh et al [3] carried out optimization of the location and size of opening hole in a pressure vessel cylinder. For the purpose of analysis and optimization, three thick-walled cylinders with different internal diameters and having same height and thickness are chosen. Further hole diameter is varied and positioned at the center of each cylinder for hole size optimization, also hole of particular size is placed at different pressure vessels surface locations. It is found that with the increase in diameter of hole, Von Misses initially decreases and then becomes constant with hole size. Finally optimum locations of the hole were found.

Shantkumar [4] emphasized on effect of stress concentration. It will be shown that an appropriate location and size of the opening in a pressure vessel results in minimizing the stresses induced due to the stress concentration resulting from the end flanges and other attachments. In this work the main aim is to design and optimize the spherical and elliptical head profile with hole on the head, also Analysis the above profiles for various stress parameter. Software’s used are Pro E & ANSYS

Viraj et al [5] research work is on cylindrical pressure vessels or vertical reactant column that is commonly used in industry to carry both liquids and gases under high pressure & temperature. Pressure vessels are used in a variety of applications in both industry and the private sector. Examples of pressure vessels are diving cylinder, distillation towers, autoclaves, and many other vessels in oil refineries and petrochemical plants. The design problem in the pressure vessel is that the holes at the bottom of pressure vessel which controls the mass flow rate are the weak regions structurally & thermally. This project focuses on optimizing the location of these holes for maximum structural safety. Software’s used are ProE & ANSYS

Ukadgaonker et al [6] presented a closed form analytical solution to the problem of an infinite plate containing collinear unequal elliptical holes subjected to in-plane loadings at an angle β with respect to x or y-axis on infinite boundary of the plate. The problem is formulated in the complex pane using the Kolo-soff-Muskhelishvili’s complex stress functions and further the Schwartz’s alternating method is used to solve the problem of doubly connected region. The stress concentration factor at all crack tips for varying sizes and centre to centre distances are evaluated. Some displacement formulation and the checked by Finite Element Method using displacement formulation and the two solutions are in good agreement. The Present results are compared with reported ones obtained by other methods. An analytical method for locating point in the vicinity of ellipses where the local and global strain energy density is equal is also presented

Shyam et al [7] focused on Pressure vessels that are leak proof containers. These are having wide range of applications in several fabrication industries like steel plants in addition to the main equipment like blast furnace, Nozzles or openings are necessary in the pressure vessels to satisfy certain requirements such as inlet or outlet connections, manholes, vents and drains etc. To incorporate a nozzle on the vessel wall it is supposed to remove some amount of material from the vessel. Then the stress distribution is not uniform. Distribution of stress in the juncture area and the rest will differ as nozzles cause a geometric discontinuity of the vessel wall. So a stress concent ration is created around the opening. The junction may fail due to these high stresses. In this work, the importance of the effect of the discontinuity is mentioned, codes related to design of vessels and its components is discussed; nozzle and vessel parameters are calculated using ASME code formulae. PV Elite is used to determine the design of pressure vessel like the thickness of shell and nozzle data. Different nozzle location with and without reinforcement of nozzle for offset of 0 ,8 16, 24 and 32 inch from vertical centre line at central cross section with different inclination angles like 0°, 15°, 30° and 45° are modeled with Creo Parametric 2.0. ANSYS workbench is used to analyze the model made in Creo Parametric 2.0 by importing it in the workbench environment, generating proper meshing, and applying boundary condition. Von-mises stress and deformation are plotted for all options under study. At initial stage all options are evaluated without reinforcement and then for properly calculated nozzle reinforcement. Summary of all Von-Mises plot are presented in tabulated as well as graphical form. Conclusions are made by discussing from available results.

University of Alberta.[8] ANSYS tutorials are very much helpful in parametrizing your model based on ANSYS Parametric Design Language (APDL). Lots of examples are given to under ANSYS environment effectively

Apart from above referenced website, Prof. Kent L Lawrence ‘s [9] own course website on Advanced Finite Element Methods using ANSYS is equipped with many worked out problems for students to become proficient in utilizing ANSYS for Finite Element Analysis

From all of the above research works, it can be seen clearly that no commercial optimization software is used for the analysis. Each time, the hole location / size is varied interactively in CAD file and then fed to ANSYS for Finite Element Analysis by trial & error method. It is a tedious and time consuming process without automation. This thesis focus on size, location optimization of a hole /holes in simple geometries like plate and cylindrical shell interfacing MATLAB (Optimization) and ANSYS (Finite Element Analysis).

Chapter 2

Problem Definition

2. 1 Objective

The main objective of this thesis work is to minimize the stress concentration by optimizing location, size of discontinuities like holes in a plate and cylindrical shell for the applied load conditions

2. 2 Cases Solved

1) **A Plate with Uniformly Distributed Load**
   - 1.1) Presence of a single hole
   - 1.2) Presence of Two holes
   - 1.3) Presence of Three holes
2) **A Plate with Linearly Varying Load**
   - 2.1) Presence of a single hole
   - 2.2) Presence of Two holes
   - 2.3) Presence of Three holes
3) **A Cylinder with an Internal Pressure**
   - 3.1) Presence of a single hole
   - 3.2) Presence of Two holes

Chapter 3

Solution approach

In this thesis work, two commercial software's MATLAB and ANSYS are interfaced to achieve the objective. Automated Optimization loop works as shown in Figure 1. In this approach the general purpose finite element package ANSYS is used for static analysis and the MATLAB function, 'FMINCON' is used as the optimizer

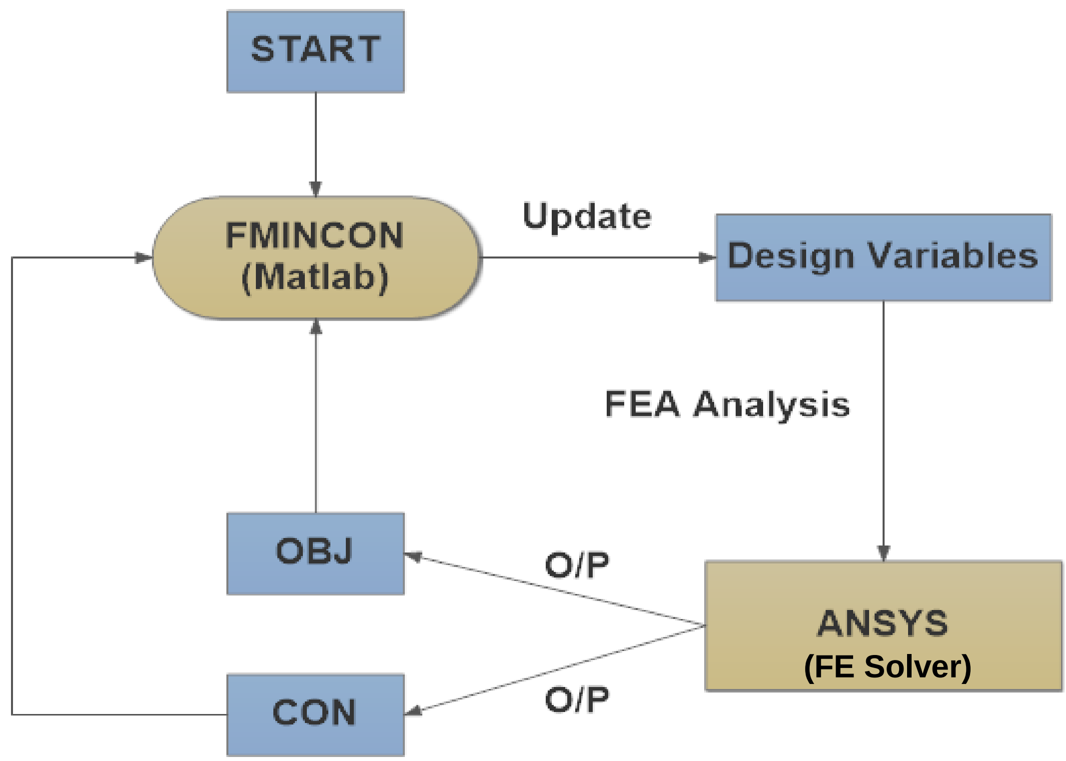


Figure 1: Optimization Loop

Whenever 'FMINCON' is called for optimization, it updates the design variables in the ANSYS parametric input file. Then the finite element analysis is being done in ANSYS in batch mode. The outputs like Stress, Displacement and Volume etc…are taken out from ANSYS and is being fed in to objective and constraint function which in turn called by FMINCON for optimization. The optimization loop continues until the local minimum is attained. MATLAB is the master software and ANSYS is the slave software.

3.1 MATLAB

The aforementioned optimization problem in the introduction (1) section can be implemented using the function '**FMINCON**' in MATLAB. The syntax is shown below

x = FMINCON(fun,x0,A,b,Aeq,beq,lb,ub,nonlcon,options)

Output arguments

Input arguments

A brief description is as follows

x- Optimized output values of the design variables

fun- Objective Function of the problem

x0- Initial design points

lb- lower bound, a row vector based on the number of design variables

ub- upper bound, a row vector based on the number of design variables

nonlcon- Equality & Inequality Constraints , a column vector based on the number of Constraints

options- Using this, we can do the following things

1. Can change the Algorithm used
2. Can define the stopping criteria for the optimization
3. Can plot the iteration values
4. Can define minimum step size for the design variables etc…..

Further In-depth details about FMINCON can be found using the below link:

https://www.mathworks.com/help/optim/ug/FMINCON.html#busow0u-1_1

3.2 ANSYS

ANSYS is a well known and widely used Finite Element Software for Static Structural Analysis. The beauty of this software is, it can be interfaced with other platforms like CAD, Hypermesh, MATLAB etc….for each individual problem based on the requirement.

In this Thesis, MATLAB is interfaced with ANSYS to perform the optimization. Before interfacing, the geometry / Structure used for analysis should be parameterized in a text format in terms of design variables. The input file for ANSYS is generated using ANSYS Parametric Design Language (APDL).It is mandatory to aware of relevant commands in APDL to parameterize your model. Those details are found using the below link.

https://www.sharcnet.ca/Software/ANSYS/16.2.3/en-us/help/ans_cmd/Hlp_C_CmdTOC.html

After Interfacing, ANSYS is being run in batch mode (i.e. in background) by MATLAB. As described in Figure 1, Whenever Optimization begins; MATLAB (FMINCON) updates the value of design variables in I/P file, run it in ANSYS and gets the outputs like Vonmises Stress, Volume, and Displacement etc. Those outputs are fed in to objective and constraint function files which is being used by FMINCON for optimization. Then the loop continues until the desired objective (Maximum/ Minimum) is achieved.

3.2.1 MESHING

In this thesis, Element types used in ANSYS are **PLANE 183** for meshing Plates and **SOLID 95** for meshing Cylinder. Free meshing is done on these geometries using size control option to create fine mesh of size 1 (minimal) and to obtain better result. A short description of the above elements are given below

3.2.2 PLANE 183

PLANE183 is a higher order 2-D, 8-node or 6-node element. It has quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

This element is defined by 8 nodes or 6 nodes having two degrees of freedom at each node: translations in the nodal x and y directions.

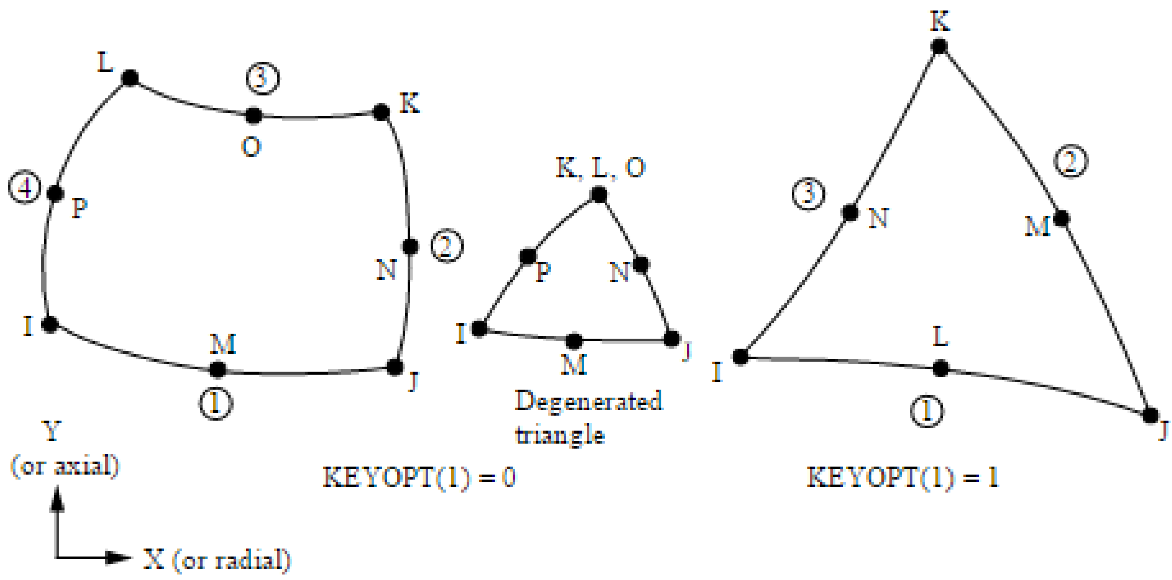


Figure 2: PLANE 183 ELEMENTS

A full description of this element is found in the below link

https://www.sharcnet.ca/Software/ANSYS/16.2.3/en-us/help/ans_elem/Hlp_E_PLANE183.html

3.2.3 SOLID 95

SOLID95 is a higher order version of the 3-D 8-node solid element (SOLID45). It can tolerate irregular shapes without as much loss of accuracy. These elements have compatible displacement shapes and are well suited to model curved boundaries.

The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions.

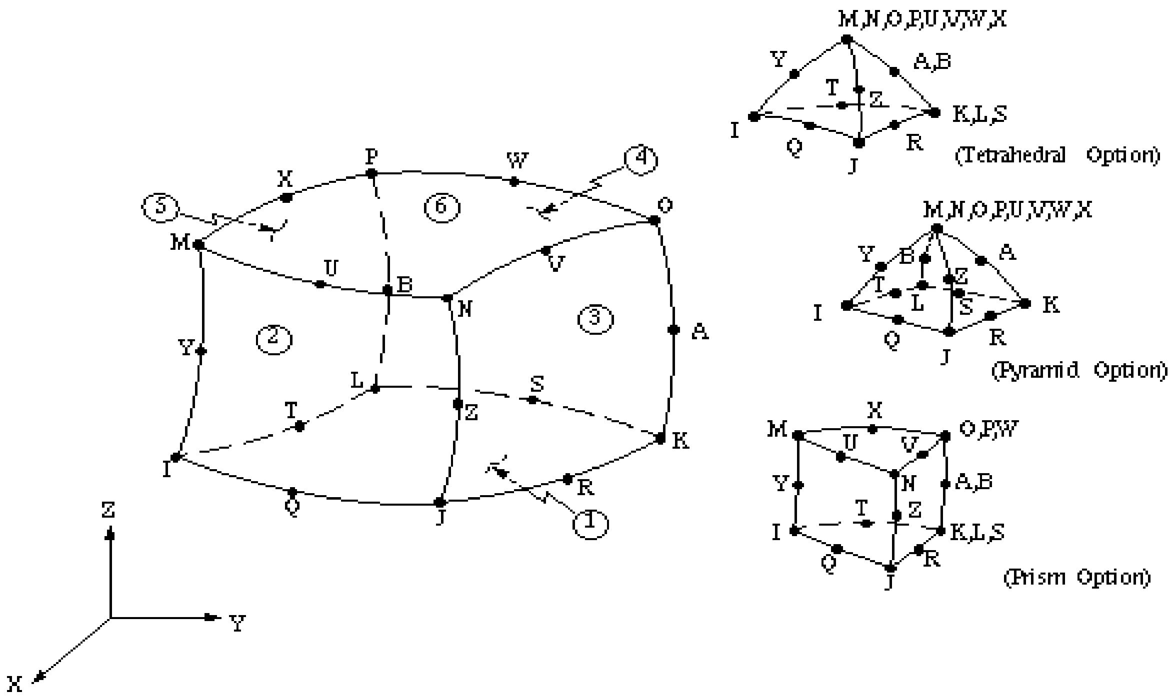


Figure 3: SOLID95 ELEMENTS

A full description of this element is found in the below link

http://www.ANSYS.stuba.sk/html/elem_55/chapter4/ES4-95.htm

3.3 Sample Implementation

A sample implementation of the proposed approach of using ANSYS-FMINCON to solve design optimization problem is described in this section. Specifically, consider the case, a plate with two holes subjected to UDL (4.1.2). For this case, the pertinent files are listed below

'runfmincon.m' - Main program for design optimization, it contains the following 2 sub-functions

(i) **'PVCH_obj'** - objective function

(ii) **'Outfun'** - function to extract optimization/ iteration history

The above functions within themselves utilize the following files to run optimization

(iii) **'Plate_opt.m' -** script file to update design variables in ANSYS input file for each iteration

(iv) **'IP_UDL.txt' –** ANSYS parameterized input file for the model chosen

See '**Appendix A'** for more program listing.

Chapter 4

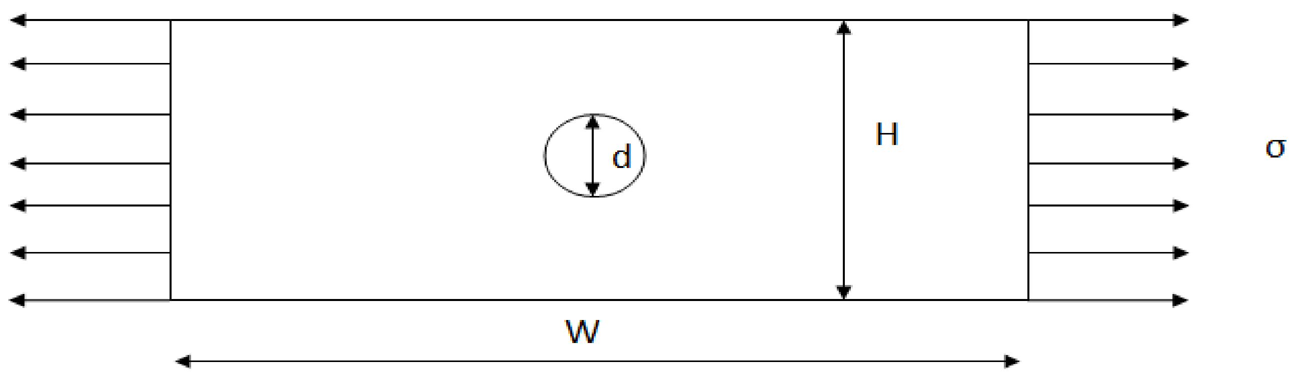
Results and Discussions

4.1 A PLATE WITH UNIFORMLY DISTRIBUTED LOAD

4.**1.1 Presence of a Single Hole**

(1) Actual Model

The pictorial representation of the plate with a hole subjected to Uniformly Distributed Load (UDL) is given below



d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress

Figure 4: UDL_PWSH

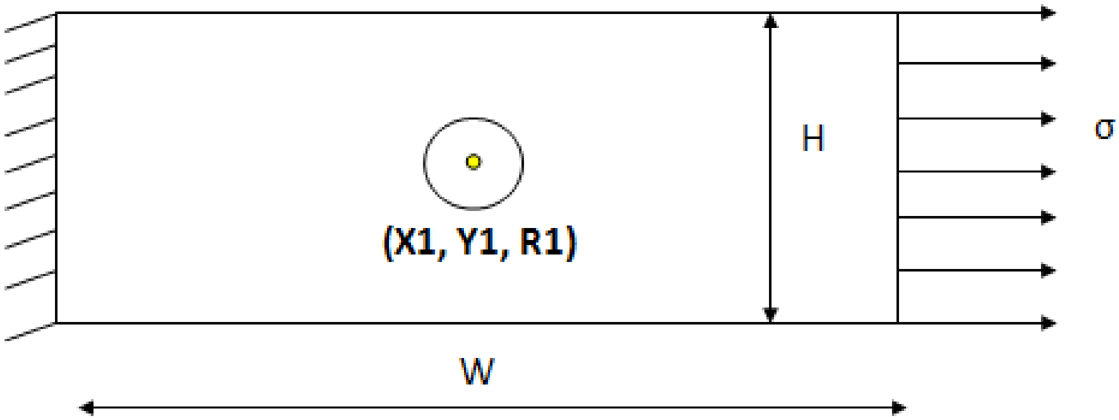
*(*2) Geometry Details

The geometry details of the plate with a hole subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ= 5MPa

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables

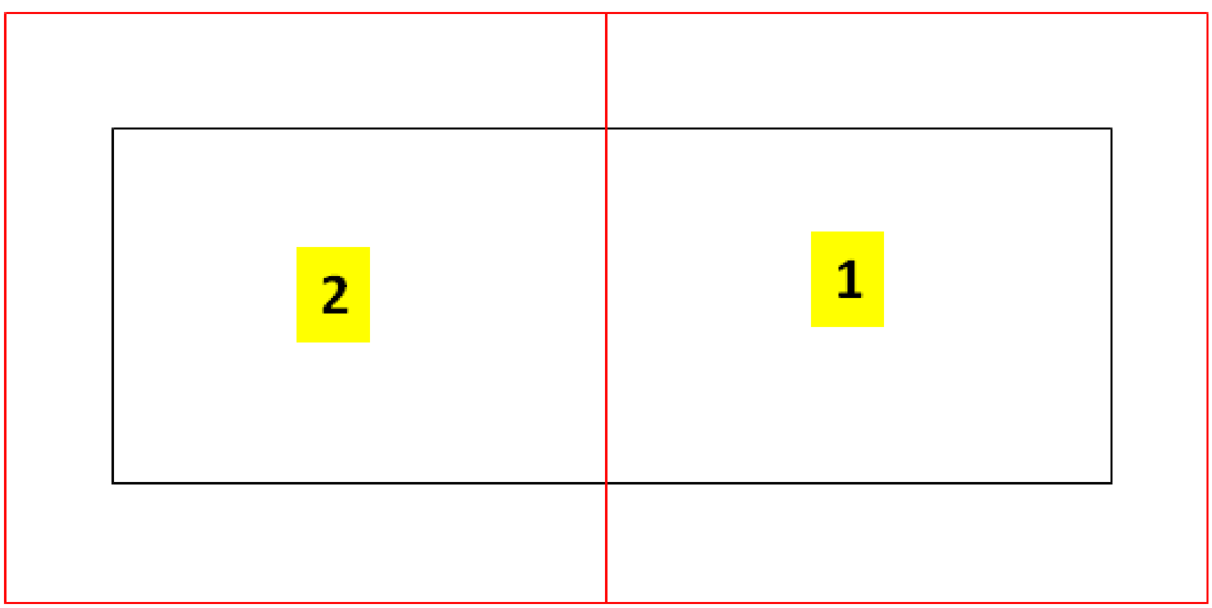


X1, Y1- Centre Coordinates of the hole

R1- Radius of the hole

(4) Contour plot of Maximum Von-Mises Stresses (S_{max})

For generating the contour plot of Maximum Von-Mises Stresses (S_{max}), centre of the plate is at origin (0, 0) and also plate is divided in to two sections as below.



The radius ‘R1’ of the hole is taken as 5 mm. The location of the hole (X1, Y1) is varied in Section 1 and Section 2 to generate the plot. S_{max} is plotted by having X1 along x-axis and Y1 along y-axis as in the figures below

(i) Section 1: (X1: 0 to 39, Y1: -15 to 15)

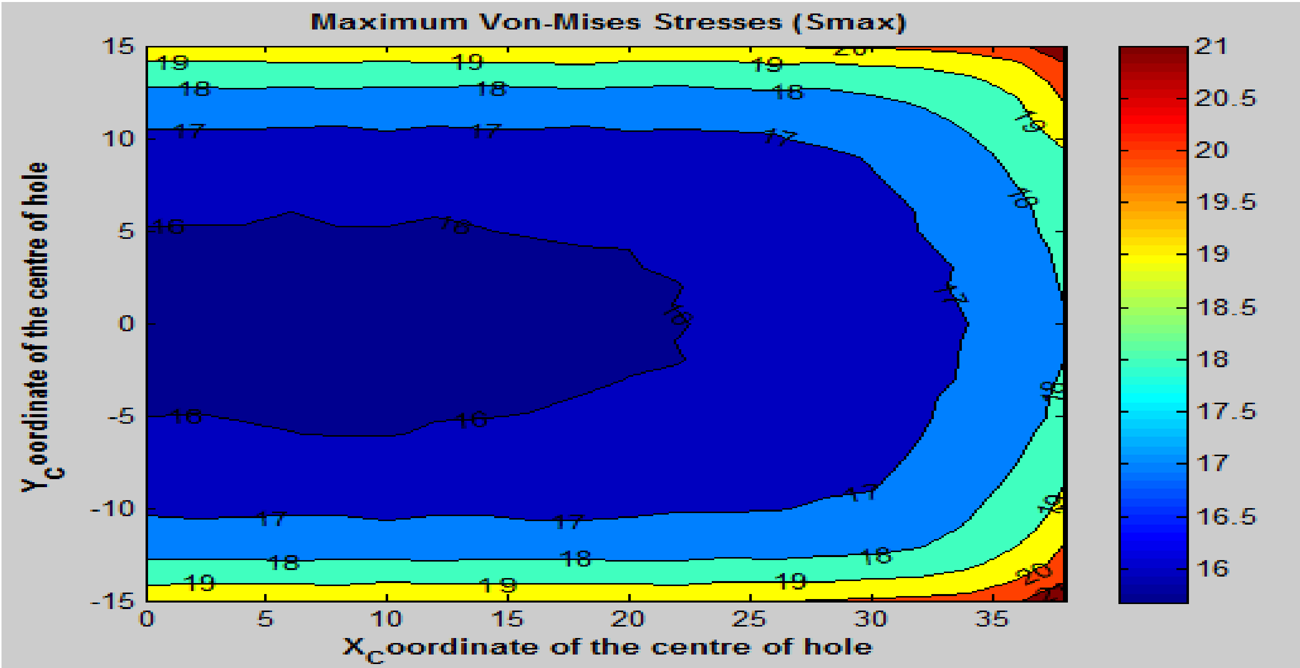


Figure 5: UDL_CPSH_S1

(ii) Section 2: (X1: -39 to 0, Y1: -15 to 15)

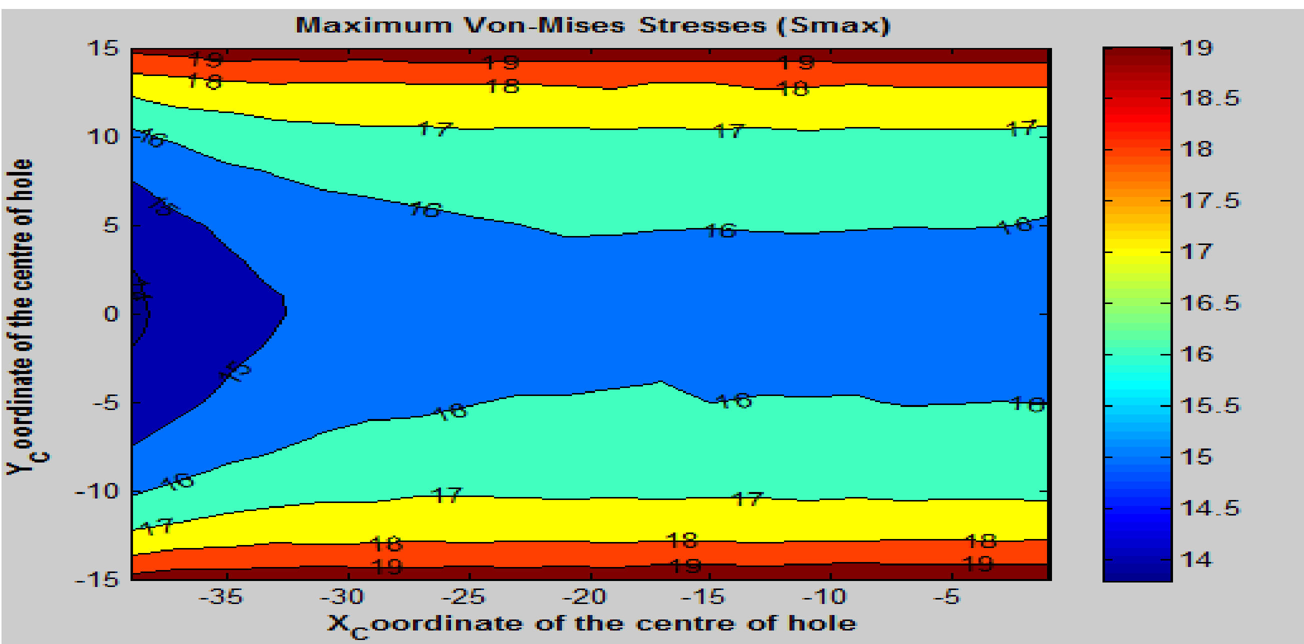


Figure 6: UDL_CPSH_S2

It is understood from the above plots that Section 1 & Section 2 are symmetric to each other. So, Optimization is carried out in Section 1 to find the best location at which stress acting is minimum which is being covered in the following pages

(5) Optimization

For performing optimization, radius 'R1' of the hole is taken as 5 mm. Co-ordinates (X1, Y1) is varied in Section 1 with the limits, X1: 0 to 39, Y1: -15 to 15

Design Variables: X1, Y1- Centre Co-ordinates of the hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints, Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[5 -5]	[0 -15]	[39 15]
2	[3 -5]	[0 -5]	[7 5]
3	[1 -0.5]	[0 -1]	[2 1]

Table 2: UDL_1H_2D_Boundary Limits

SETS	X1	Y1	SMAX
1	7.1867	0.0465	15.6349
2	4.9992	-0.5647	15.631
3	1.5	-0.2975	15.6257

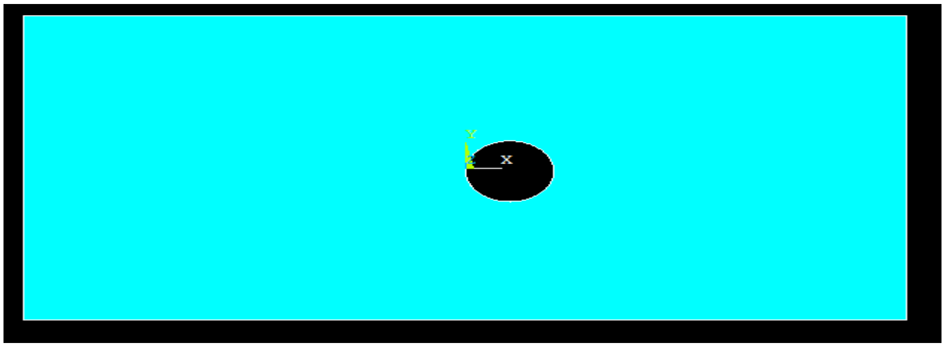
Table 3: UDL_1H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as hole approaches origin (0, 0). Hence the best location for a Plate with Single hole subjected to tensile load is at the centre of the plate. Also from figures 5 & 6, stress values are less near the origin.

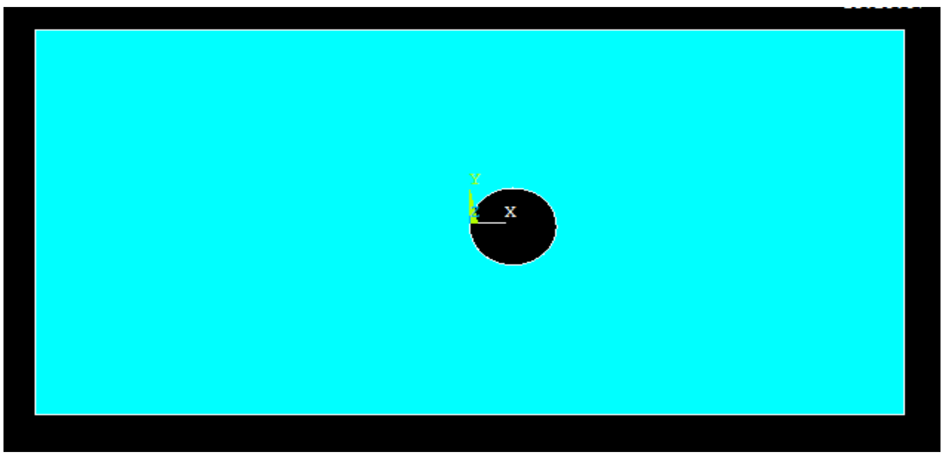
(ii) Plots

The below plots are generated based on values of (X1, Y1) from table 2

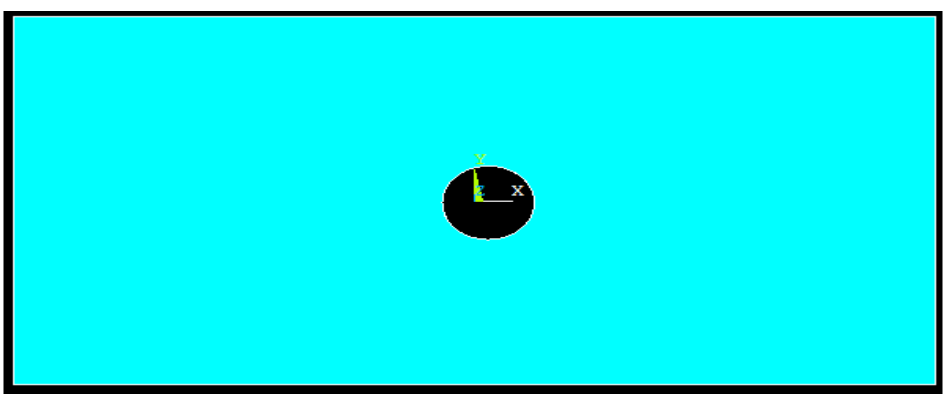
1) SET 1



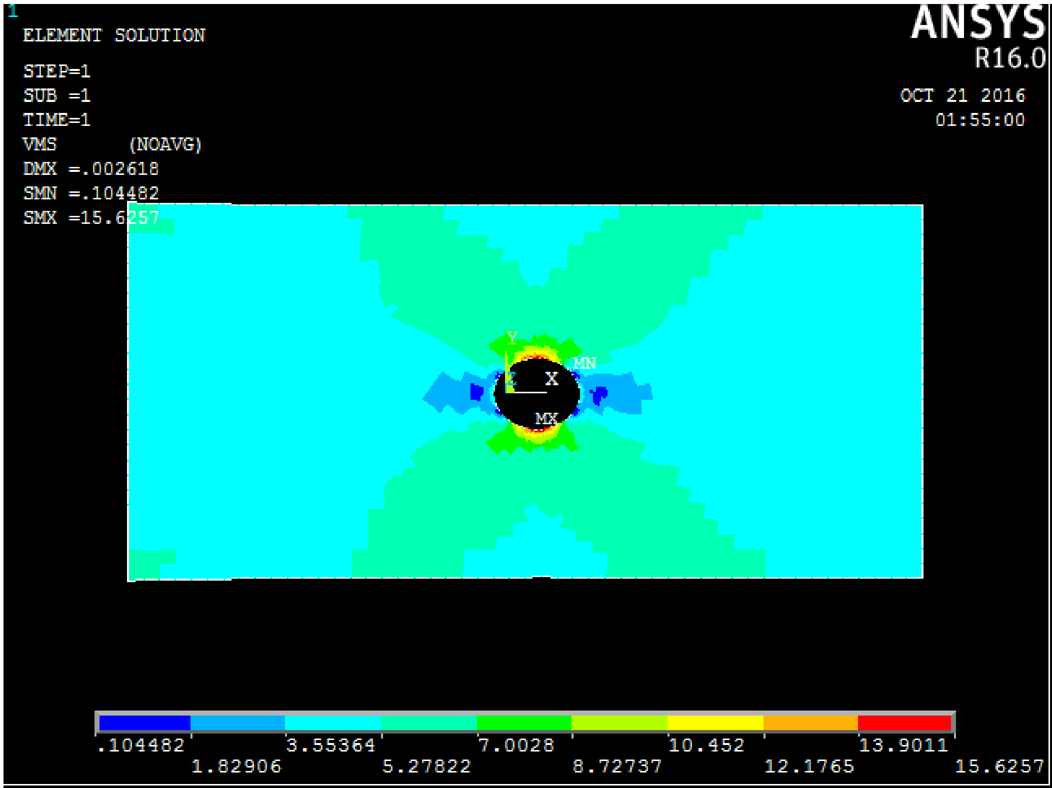
2) SET 2



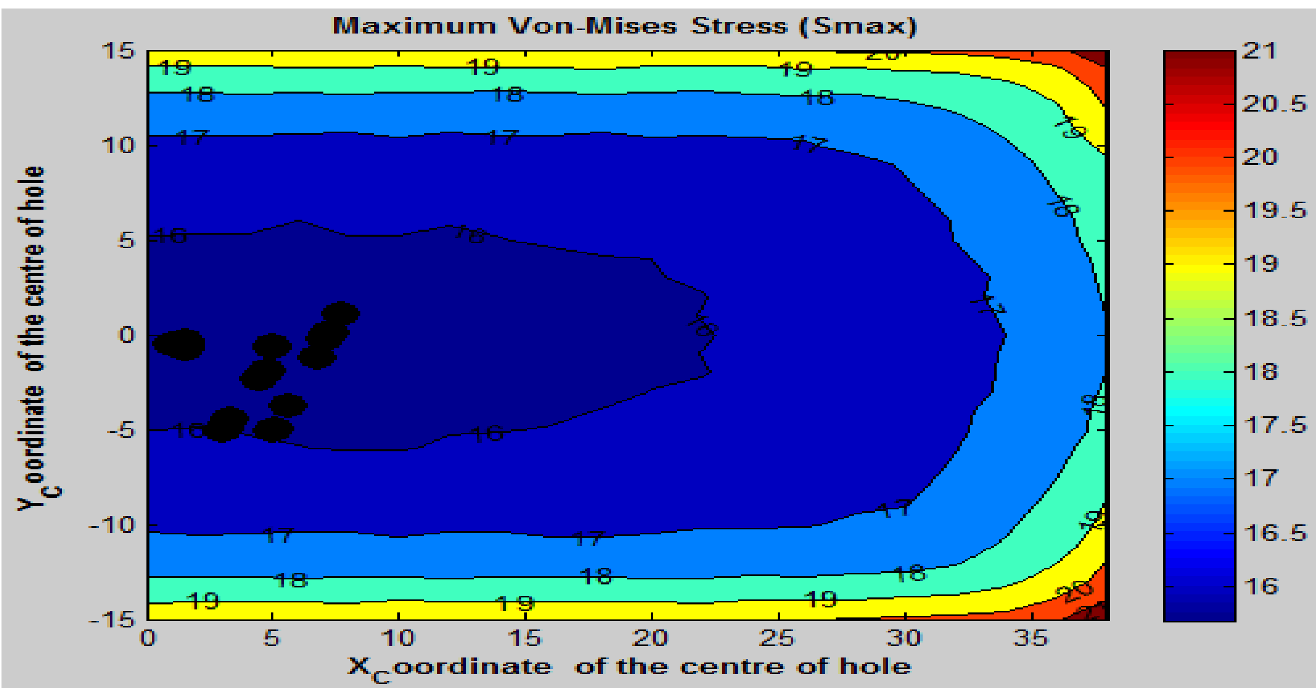
3) SET 3



(iv) Contour plot of maximum stress concentration



(iii) Stress Plot with Design Points



(6) Maximum Von-mises Stress Vs Radius of the Hole

For generating below plot, hole is fixed at origin (i.e. X1=Y1=0). Only radius of the hole 'R1' is varied to get below plot

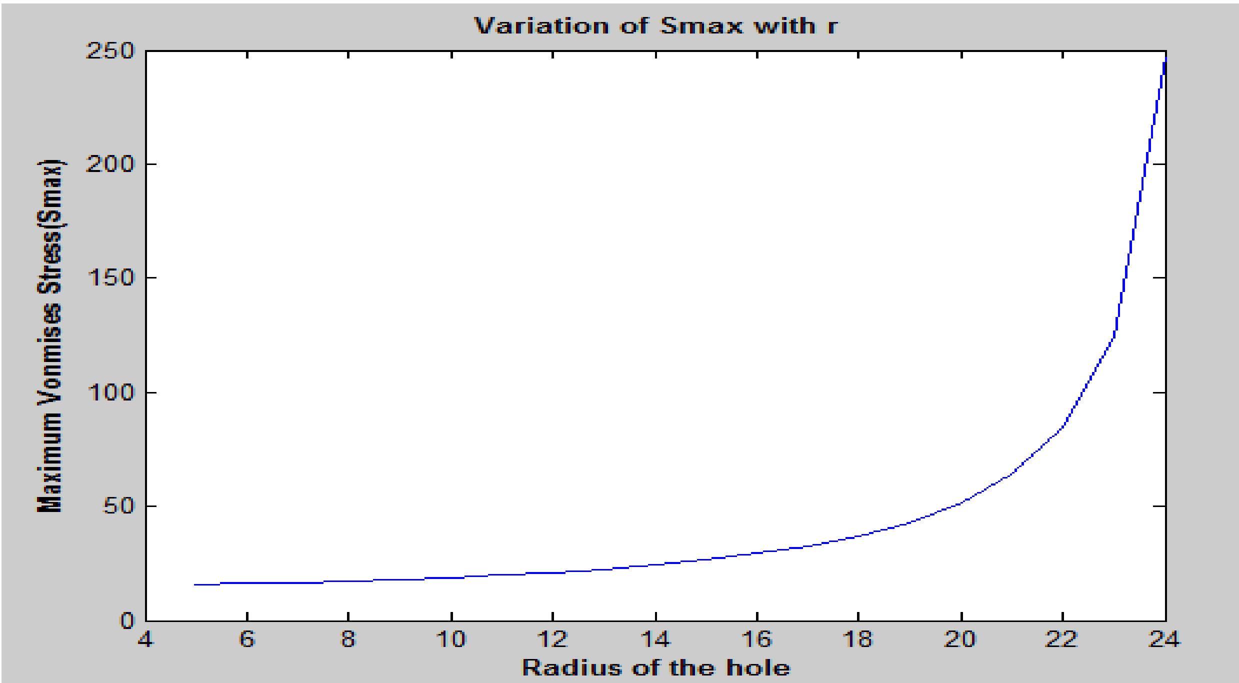


Figure 7: UDL_Smax Vs R1

From figure 7, it is understood that S_{max} is getting increased whenever hole size increases

(7) Analytical Solution

The approximate analytical solution for a finite width plate with hole at the centre is taken from the book '***Stress concentration factor by Walter D. Pilkey & Deborah F. Pilkey'.*** The below formula is derived by *Heywood (1952)*

(i) Stress Concentration Factor

$$K_{tn} = 2 + (1 - \frac{d}{H})^3$$

d- diameter of the hole, H- Width of the plate

(ii) Maximum Stress

$$\sigma_n = \frac{\sigma}{(1-\frac{d}{H})}, \quad \sigma_{max} = \sigma_n * K_{tn}$$

σ_{max} – Maximum Stress developed due to discontinuity in Structure

σ_n – Nominal Stress, σ – Applied Stress

Substituting: d=10 mm, H=50 mm, σ= 5 MPa

We get: **K_{tn}=2.512, σ_n= 6.25 MPa, σ_{max}= 15.7 MPa**

(iii) Comparison

ANALYTICAL	ANSYS	% ERROR
15.7	15.622	0.5

From the above result, analytical values are pretty much agreed with ANSYS results with the error of 0.5 percentages.

(iv) Notes

(d/H)	K_t	σ_n	σ_{max}
0.2	2.512	6.25	15.7
0.96	2	125	250

(d/H)	K_t	σ_n	σ_{max}
Decreases	Increases	Decreases	Decreases
Increases	Decreases	Increases	Increases

From the above tables, we can come to conclusion that

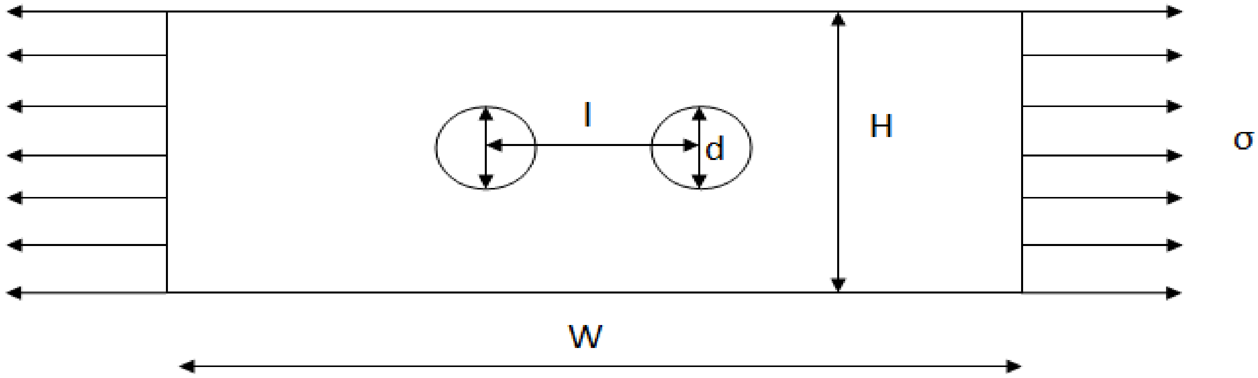
1) Whenever d/H is of lower value, K_t is higher, nominal stress (σ_n) is lower and finally Maximum stress (σ_{max}) is also decreased. So, σ_{max} is proportional to σ_n

2) Whenever d/H is of higher value, K_t is lower, nominal stress (σ_n) is higher and finally Maximum stress (σ_{max}) is also increased. So, σ_{max} is proportional to σ_n

4.1.2 Presence of Two Holes

(1) Actual Model

The pictorial representation of the plate with two holes subjected to Uniformly Distributed Load (UDL) is given below



d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress, l- centre to centre distance

Figure 8: UDL_PW2H

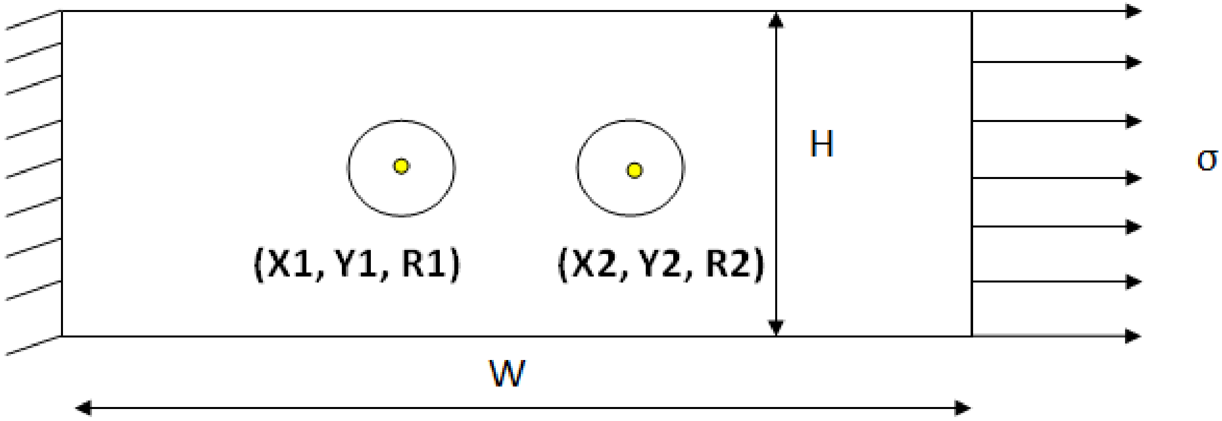
(2) Geometry Details

The geometry details of the plate with two holes subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ= 5MPa

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables

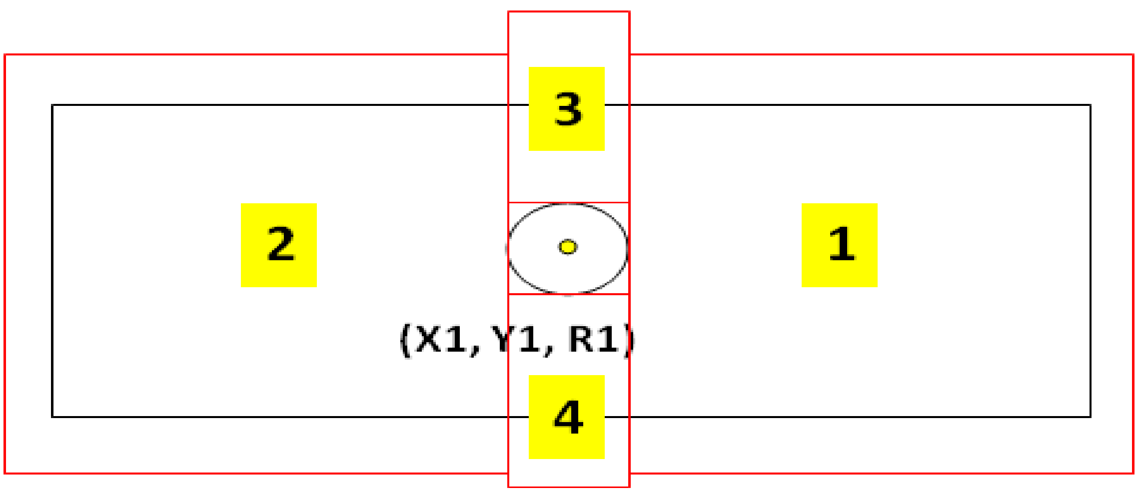


X1, Y1, R1- Center Coordinates & Radius of 1st hole

X2, Y2, R2- Center Coordinates & Radius of 2nd hole

(4) Contour plot of Maximum Von-Mises Stresses (S_{max})

For generating the contour plot of Maximum Von-Mises Stresses (S_{max}). 1st hole is fixed at centre of the plate (i.e. at origin) and also plate is divided in to four sections as below.



The radius ‘R1’ & ‘R2’ of the holes are taken as 5 mm. The location of 2nd hole (X2, Y2) is varied in the above sections to generate the plot. S_{max} is plotted by having X2 along x-axis and Y2 along y-axis as in the figures below

(i) Section 1: (X1: 11 to 39, Y1: -15 to 15)

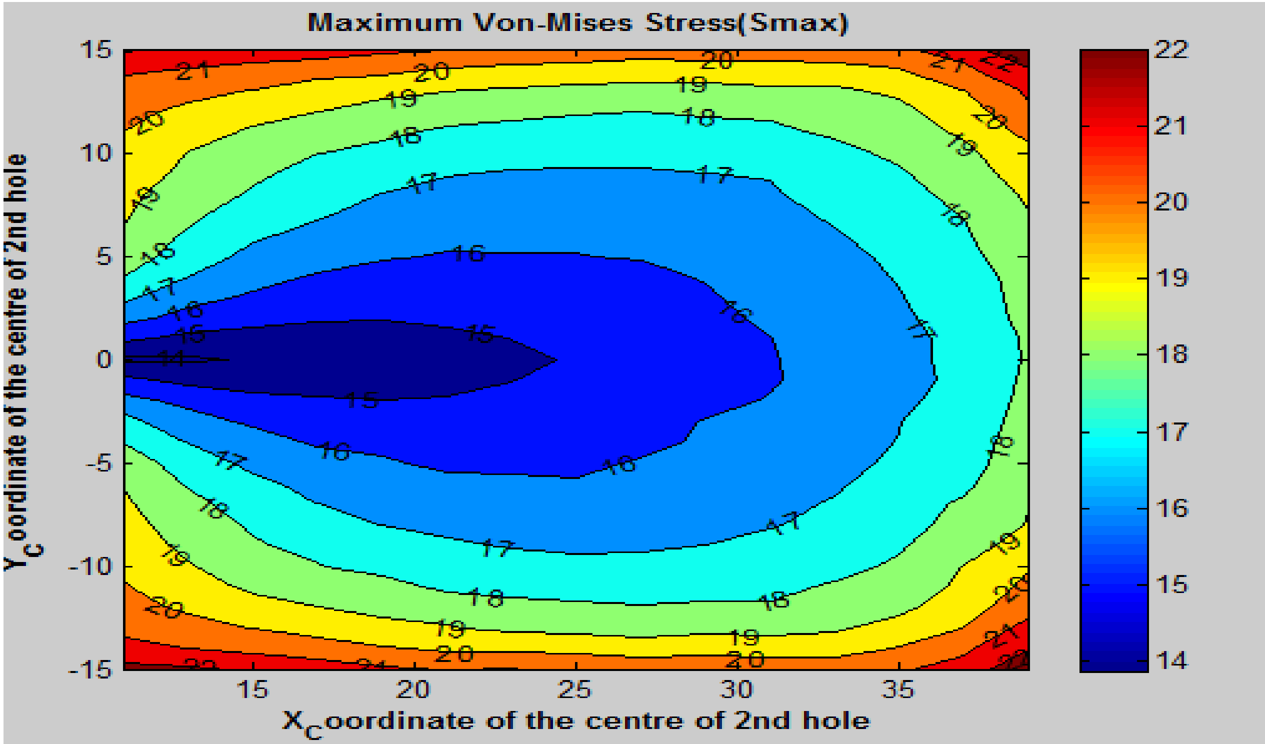


Figure 9: UDL_CP2H_S1

(ii) Section 2: (X1: -39 to -11, Y1: -15 to 15)

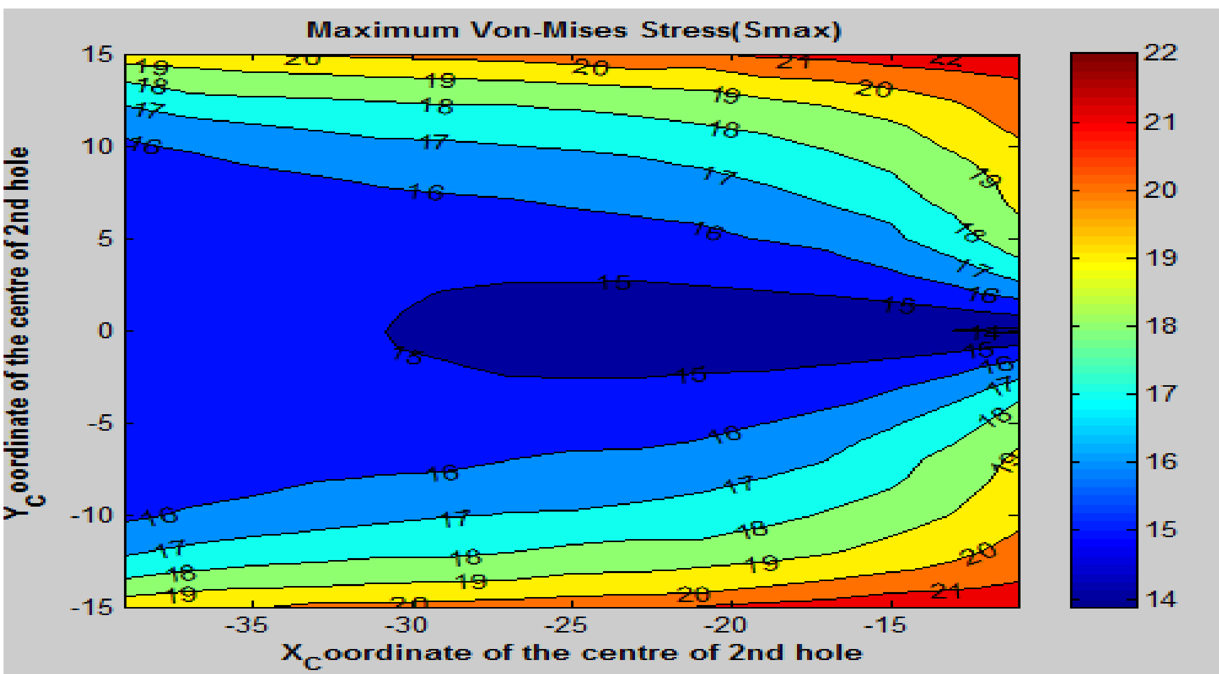


Figure 10: UDL_CP2H_S2

(iii) Section 3: (X1: -10 to 10, Y1: 11 to 15)

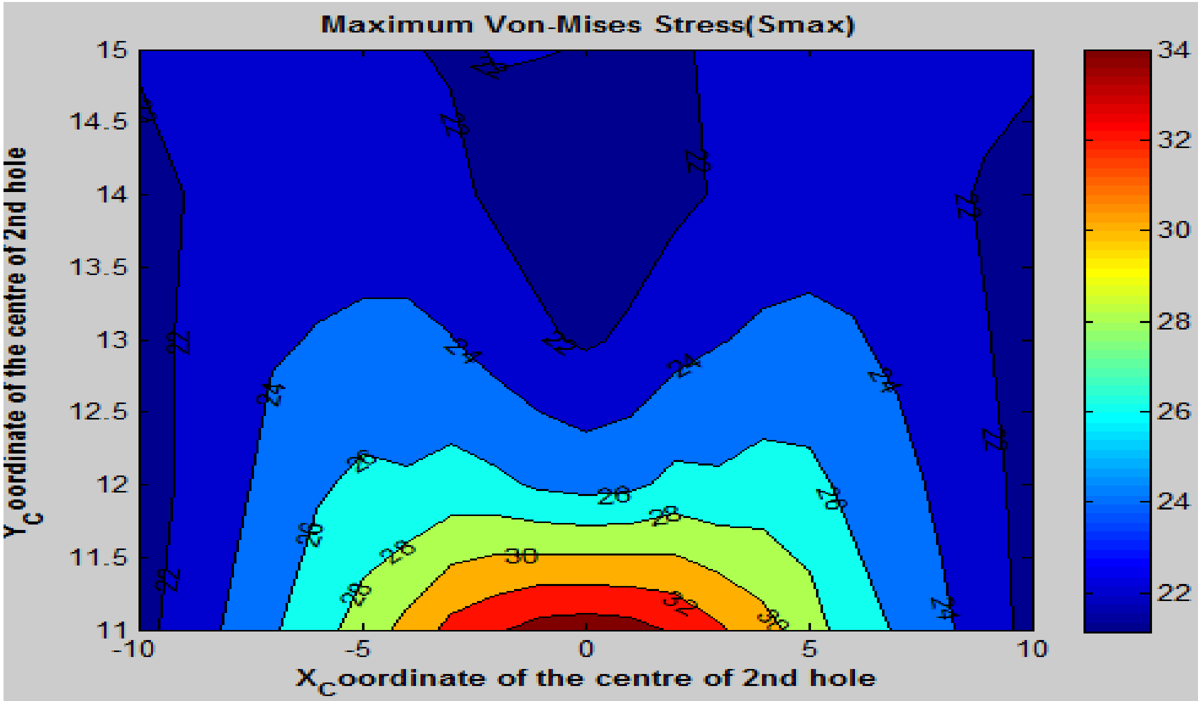


Figure 11: UDL_CP2H_S3

(iv) Section 4: (X1: -10 to 10, Y1: -15 to -11)

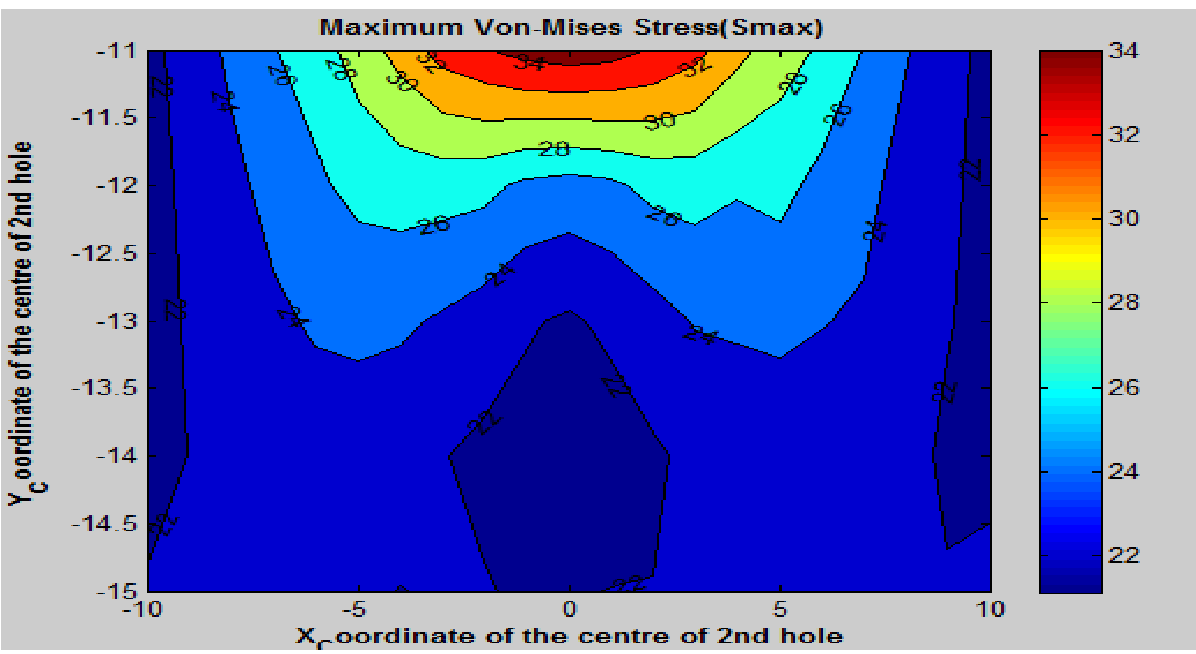


Figure 12: UDL_CP2H_S4

It is understood from the above plots that Sections 1 & 2, Sections 3 & 4 are symmetric to each other. Stress is less in Sections 1 & 2 comparatively. So, Optimization is carried out in Section 1 to find the best location at which stress acting is minimum which is being covered in the following pages

(5) Optimization

For Performing optimization, 1st hole is fixed at origin (0, 0) and its radius 'R1' is taken as 5 mm. Co-ordinates (X2, Y2) is varied in Section 1 with the limits, X2: 11 to 39, Y2: -15 to 15

(5.1) Two Design Variables

For this case, radius of 2nd hole 'R2' is taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of the hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints, Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[13 -5]	[11 -15]	[39 15]
2	[12 0]	[11 -5]	[15 5]
3	[11.3 0]	[11 -1]	[11.5 1]

Table 4: UDL_2H_2D_Boundary Limits

SETS	X2	Y2	SMAX
1	12.3382	-0.0502	13.8913
2	11.9968	-0.0191	13.8432
3	11.3	0	13.8409

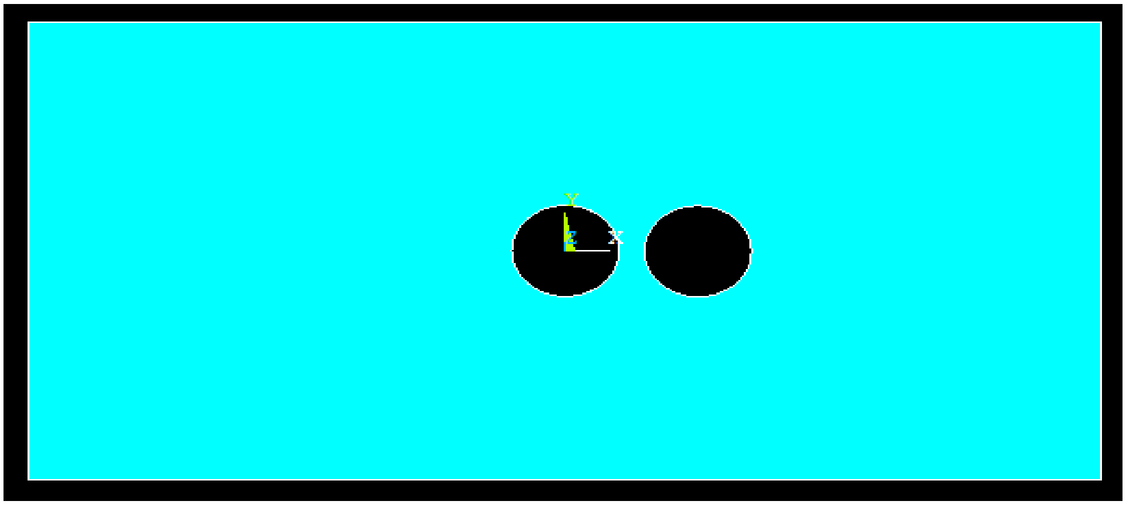
Table 5: UDL_2H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd hole goes closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of two holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure.

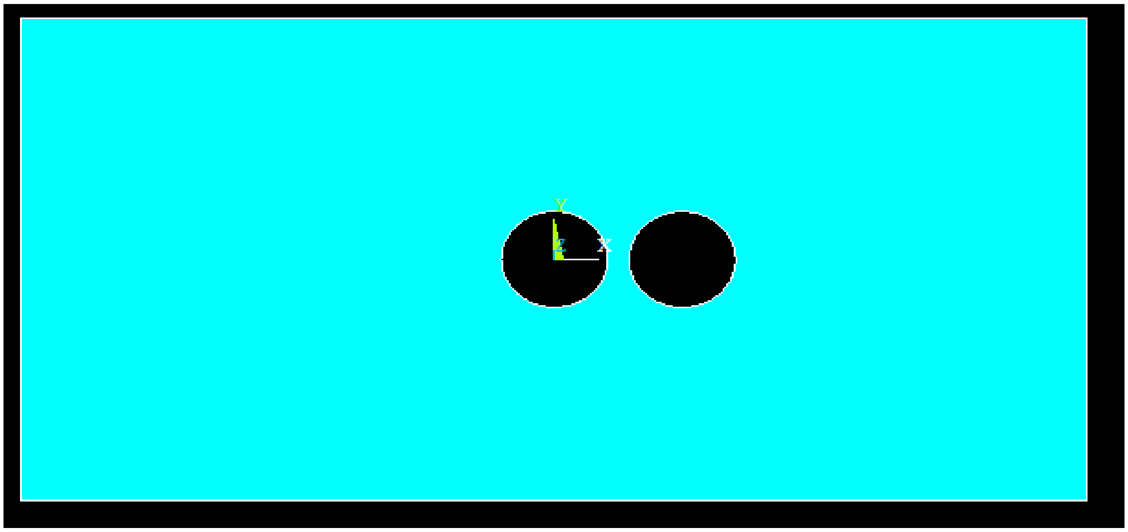
(ii) Plots

The below plots are generated based on values of (X2, Y2) from table 5

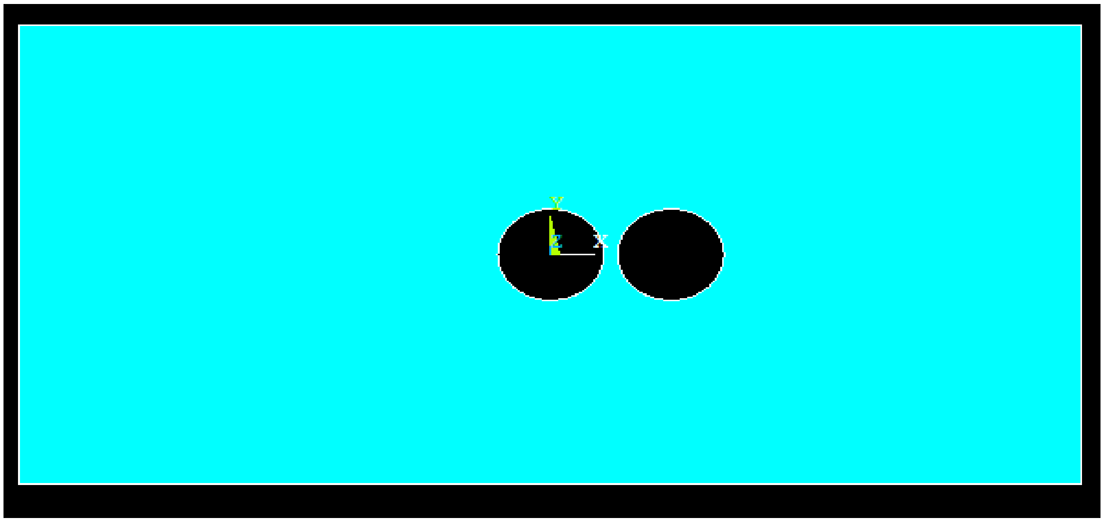
1) SET 1



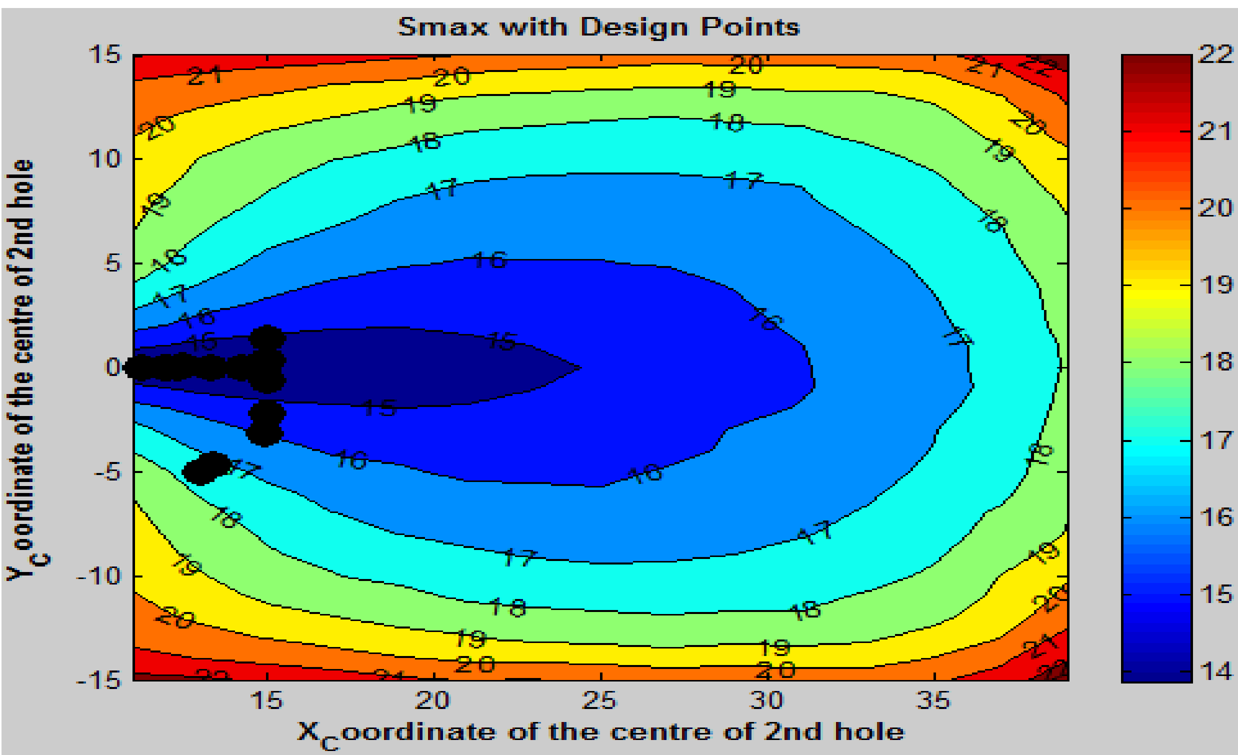
2) SET 2



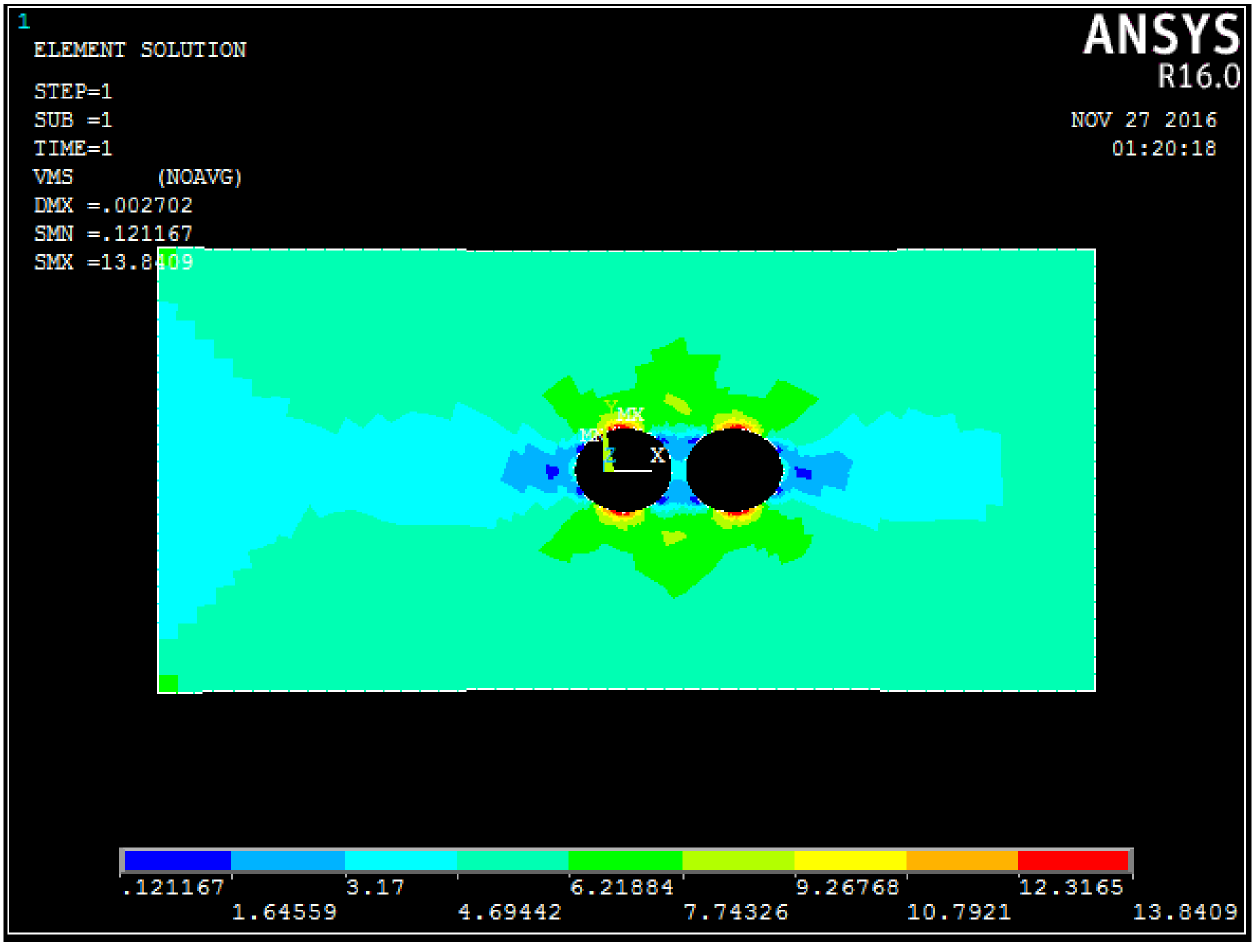
3) SET 3



(iii) Stress Plot with Design Points



(iv) Contour plot of maximum stress concentration



(5.2) Three Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[13 -5 3.3]	[11 -15 3]	[15 15 5]

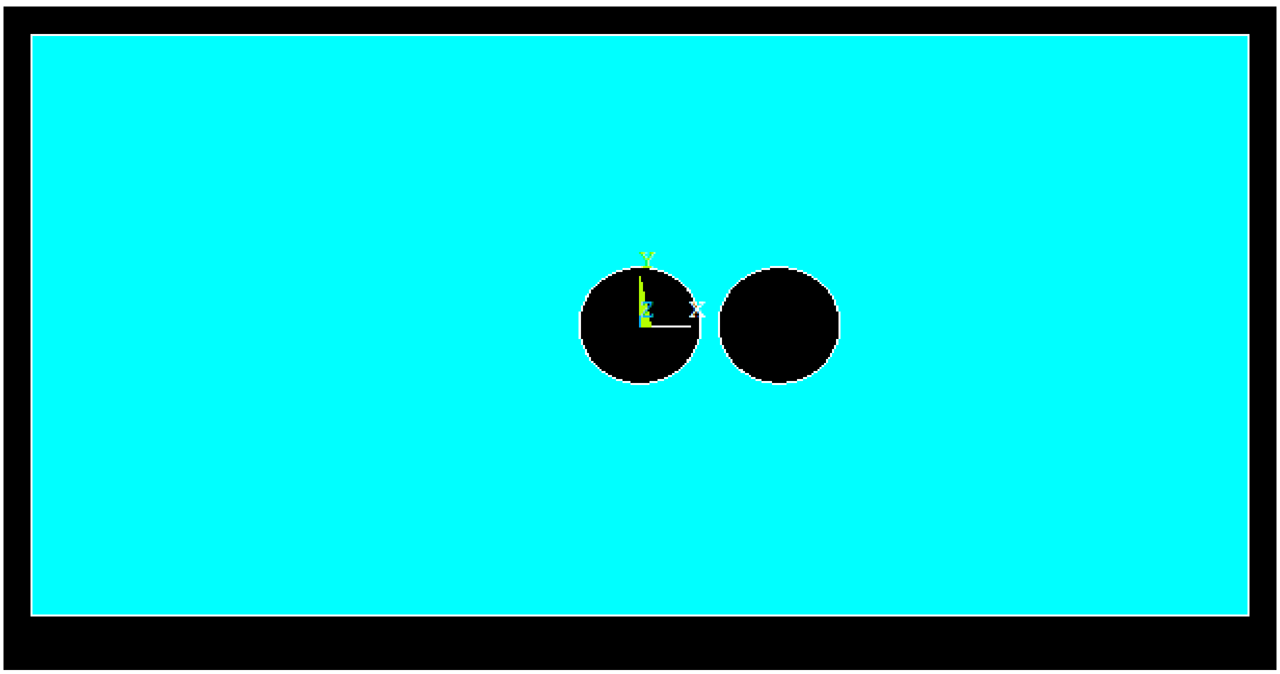
Table 6: UDL_2H_3D_Boundary Limits

SETS	X2	Y2	R2	SMAX
1	11.4435	0.0187	4.9999	13.8228

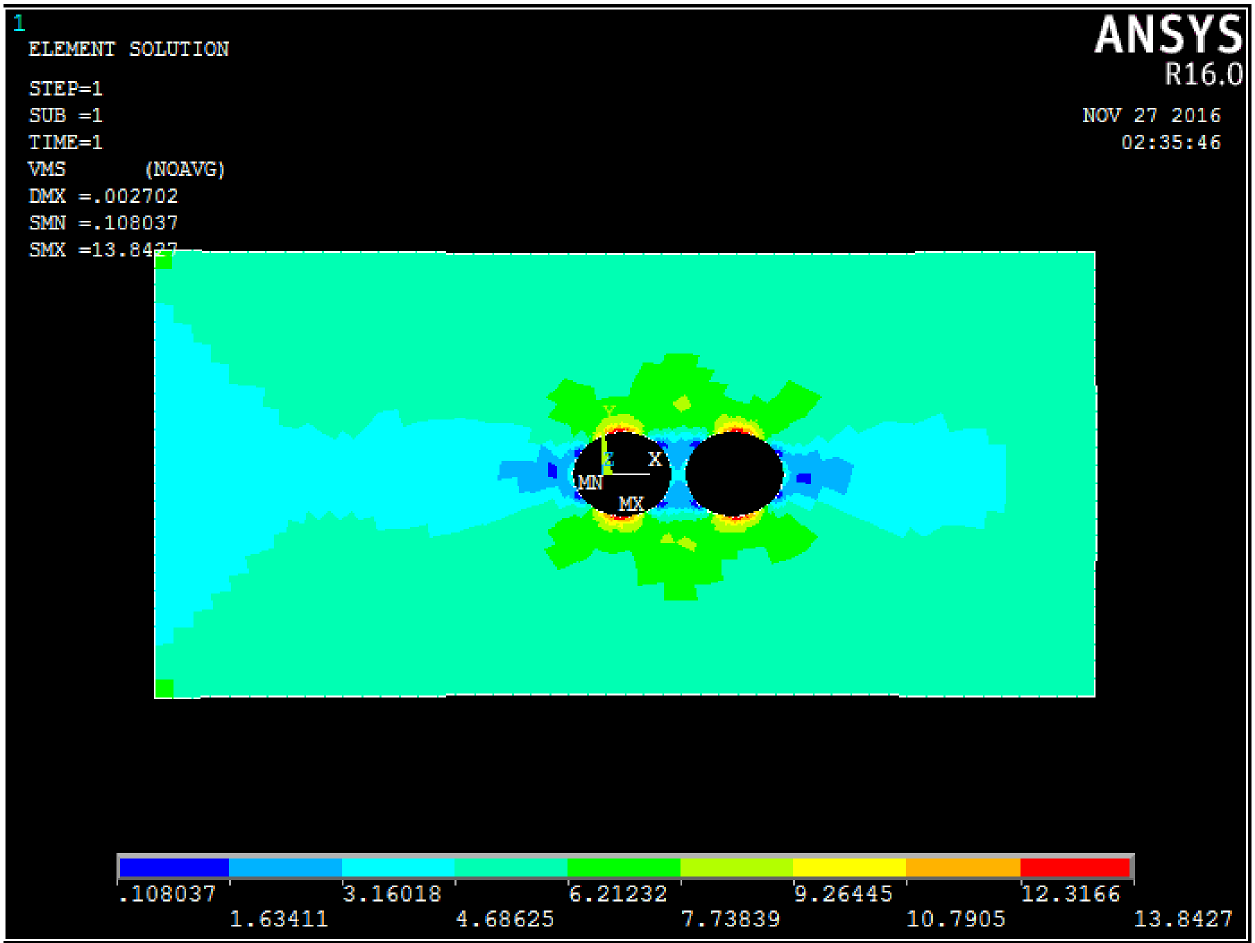
Table 7: UDL_2H_3D_Optres

(ii) Plots

The below plot is generated based on values of (X2, Y2, R2) from table 7



(iii) Contour plot of maximum stress concentration



(6) Maximum Von-mises Stress Vs X- Coordinate of 2^{nd} hole

For generating below plot, 1^{st} Hole is fixed at origin (i.e. X1=Y1=0). Radius of the holes 'R1' & 'R2' are taken as 5 mm, 2^{nd} hole is made collinear with 1^{st} hole (i.e Y2=0 for all values of X2). Only X2 is varied to generate below plot

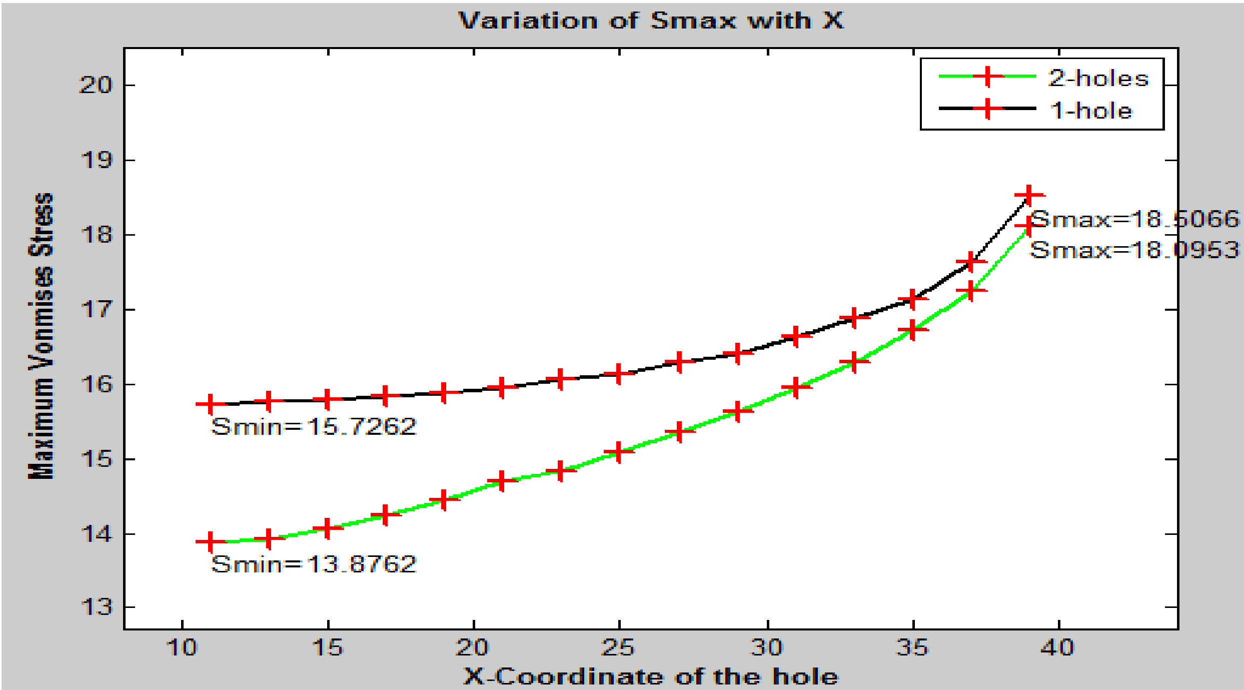


Figure 13: UDL_Smax Vs X2

(7) Maximum Von-mises Stress Vs Radius of 2nd hole

For generating below plot, 1st Hole is fixed at origin (i.e. X1=Y1=0). Radius of 1st hole is taken as 5 mm, 2nd hole is made collinear with 1st by fixing it at X2=11,Y2=0. Only 'R2' is varied to generate below plot

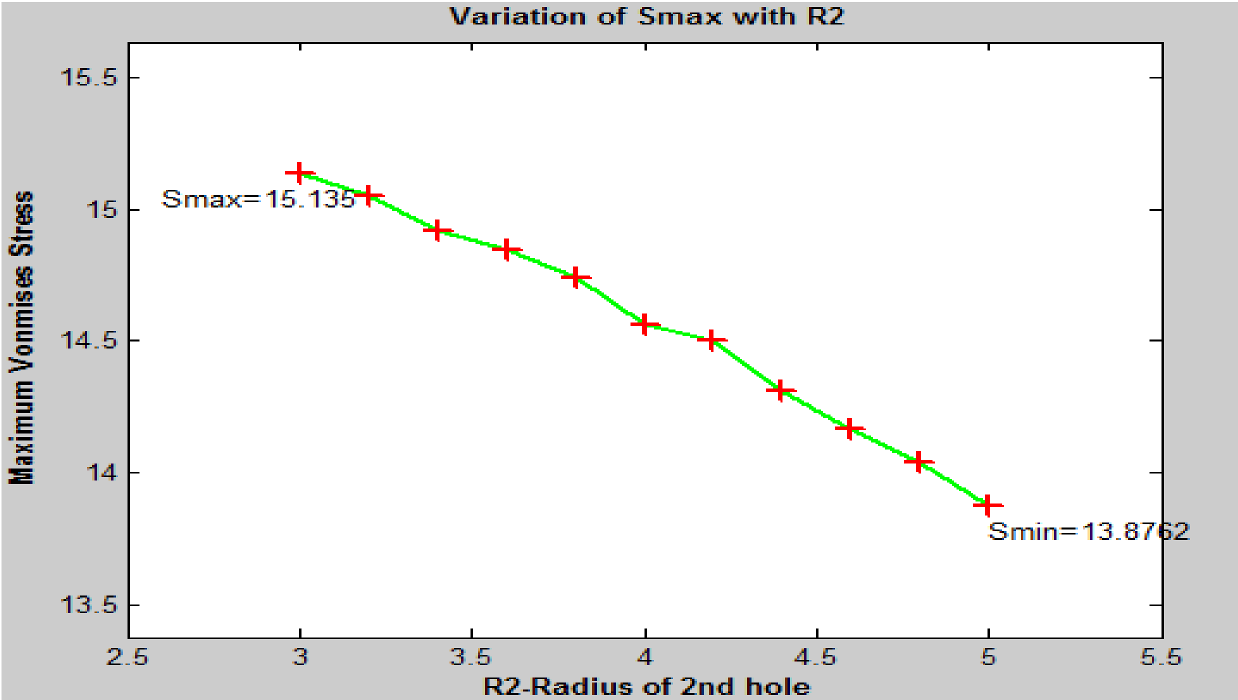


Figure 14: UDL_Smax Vs R2

From figures 13 & 14, it is evident that stress is getting relieved & drops to a lower value only when the holes are of same size and close to each other

(7) Analytical Solution

The approximate analytical solution for an infinite plate with two holes of same size & same distance apart is derived by *Ling (1948) & Haddon (1967)*

(i) Stress Concentration Factor

$$K_{tn} = 3 - \left(0.712 * \frac{d}{l}\right) + \left(0.271 * \left(\frac{d}{l}\right)^2\right)$$

d- Diameter of the holes, l- Centre to Centre distance

(ii) Maximum Stress

$$\sigma_{max} = \sigma * K_{tn}$$

σ_{max} – Maximum Stress developed due to discontinuity in Structure

σ – Applied Stress

Substituting: d=10 mm, l=11.3 mm, σ= 5 MPa

We get: **K_{tn}=2.58, σ_{max}= 12.9 MPa**

(iii) Comparison

ANALYTICAL	ANSYS	% ERROR
12.9	13.8816	6

From the above result, analytical values are pretty much agreed with ANSYS results with the error of 6 percentages.

(iv) Notes

l	(d/l)	K_t	σ_{max}
20	0.5	2.712	13.6
10	1	2.559	12.8

(d/l)	K_t	σ_{max}
Decreases	Increases	Increases
Increases	Decreases	Decreases

From the above tables, we can come to conclusion that

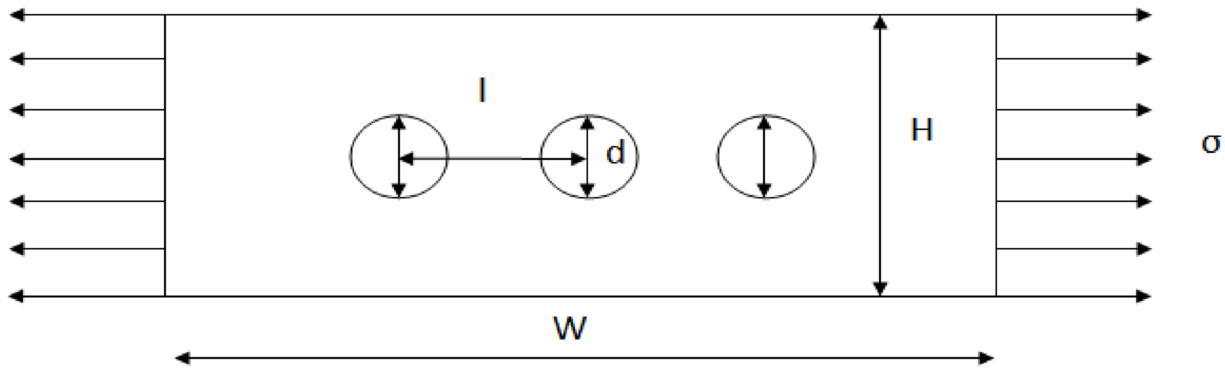
1) Whenever d/l ratio is of lower value, K_t is higher, which in turn increases σ_{max}. So, σ_{max} is proportional to K_t

2) Whenever d/l ratio is of higher value, K_t is lower, which in turn decreases σ_{max}.

4.1.3 Presence of Three Holes

(1) Actual Model

The pictorial representation of the plate with three holes subjected to Uniformly Distributed Load (UDL) is given below



d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress, l- centre to centre distance

Figure 15: UDL_PW3H

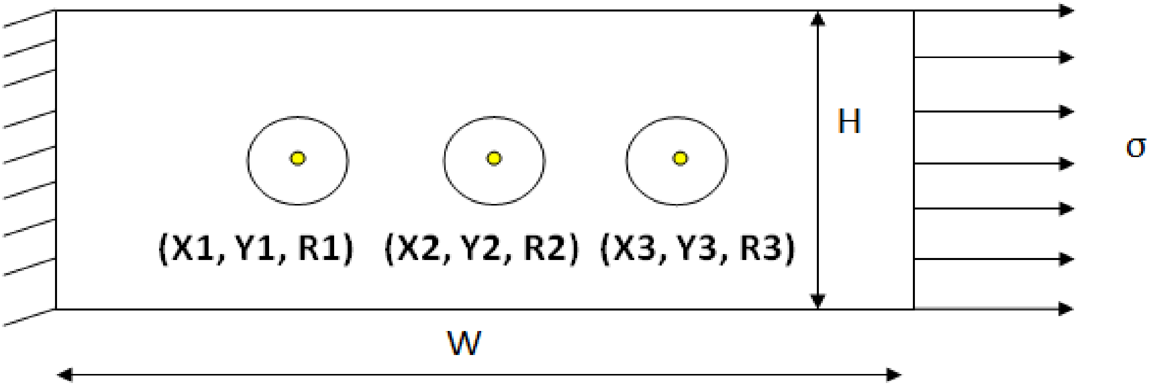
(2) Geometry Details

The geometry details of the plate with three holes subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ= 5MPa

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables



X1, Y1, R1- Center Coordinates & Radius of 1st hole

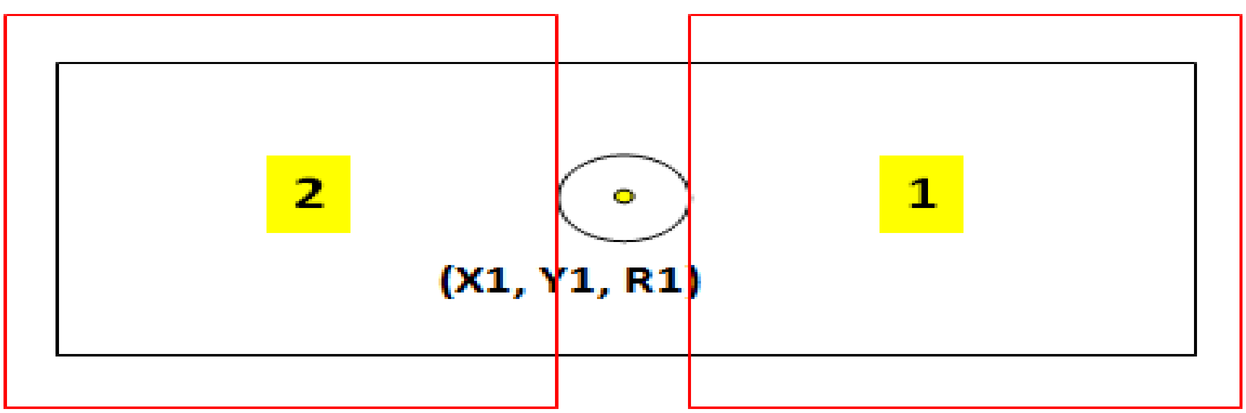
X2, Y2, R2- Center Coordinates & Radius of 2nd hole

X3, Y3, R3- Center Coordinates & Radius of 3rd hole

(4) Optimization

(4.1) Part 1

For performing optimization, 1st hole is fixed at origin (0, 0) and its radius ‘R1’ is taken as 5 mm. Co-ordinates (X2, Y2) of 2nd hole is varied in negative x-axis (Section 2) with the limits, X2: -39 to -11, Y2: -15 to 15. Co-ordinates (X3, Y3) of 3rd hole are varied in positive x-axis (Section 3) with the limits, X3: 11 to 39, Y3: -15 to 15.It is further pictorially represented below



(4.1.1) Four Design Variables

For this case, radius of 2nd & 3rd holes are taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of 2nd hole,
X3, Y3- Centre Co-ordinates of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[-14 0 14 0]	[-39 -15 11 -15]	[-11 15 39 15]
2	[-12 0 12 0]	[-13 -1 11 -1]	[-11 1 13 1]
3	[-11.3 0 11.3 0]	[-11.5 -1 11 -1]	[-11 1 11.5 1]

Table 8: UDL1_3H_4D_Boundary Limits

SETS	X2	Y2	X3	Y3	SMAX
1	-13.6365	-0.0046	14.0442	0.0311	13.3631
2	-11.9858	-0.1004	11.9256	-0.1202	13.3258
3	-11.1479	0.0086	11.3075	-0.0879	13.1555

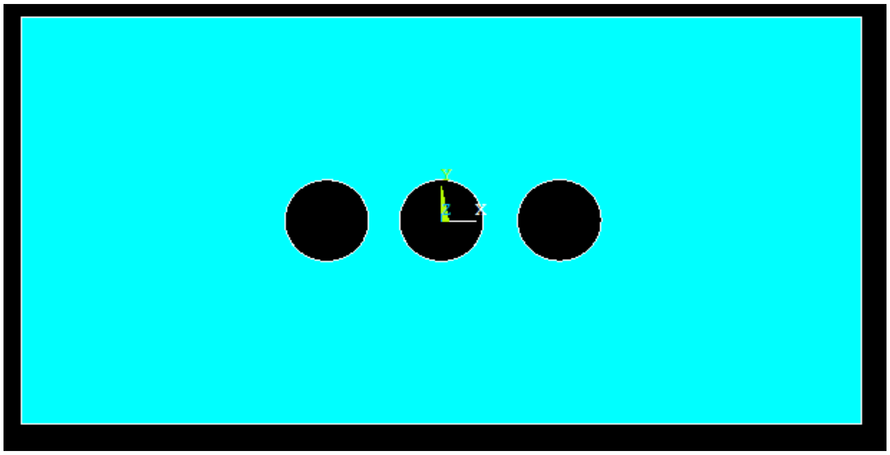
Table 9: UDL1_3H_4D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd & 3rd holes gets closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure

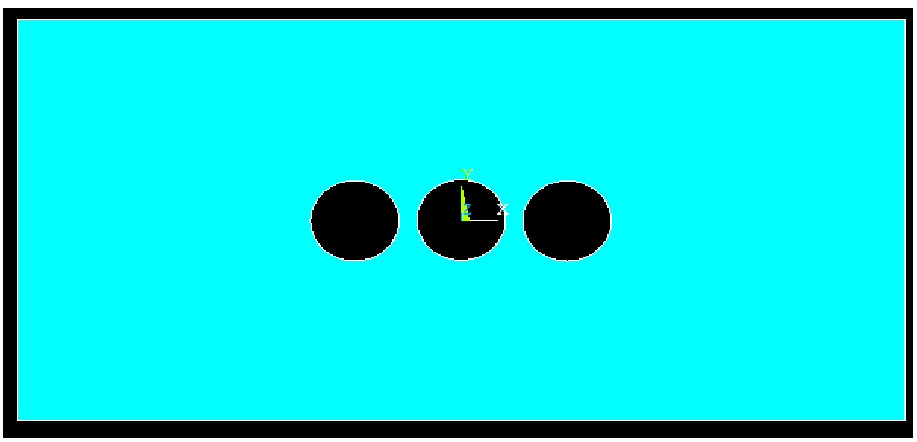
(ii) Plots

The below plots are generated based on values of (X2, Y2) & (X3, Y3) from table 9

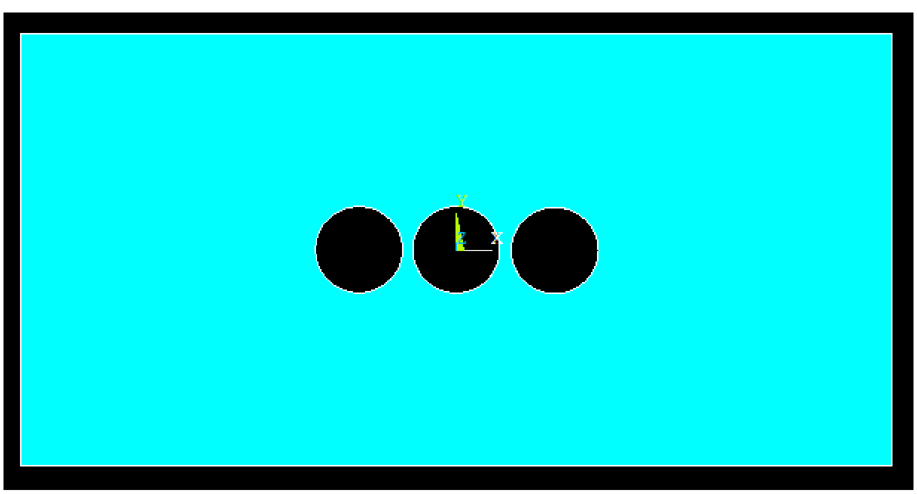
1) SET 1



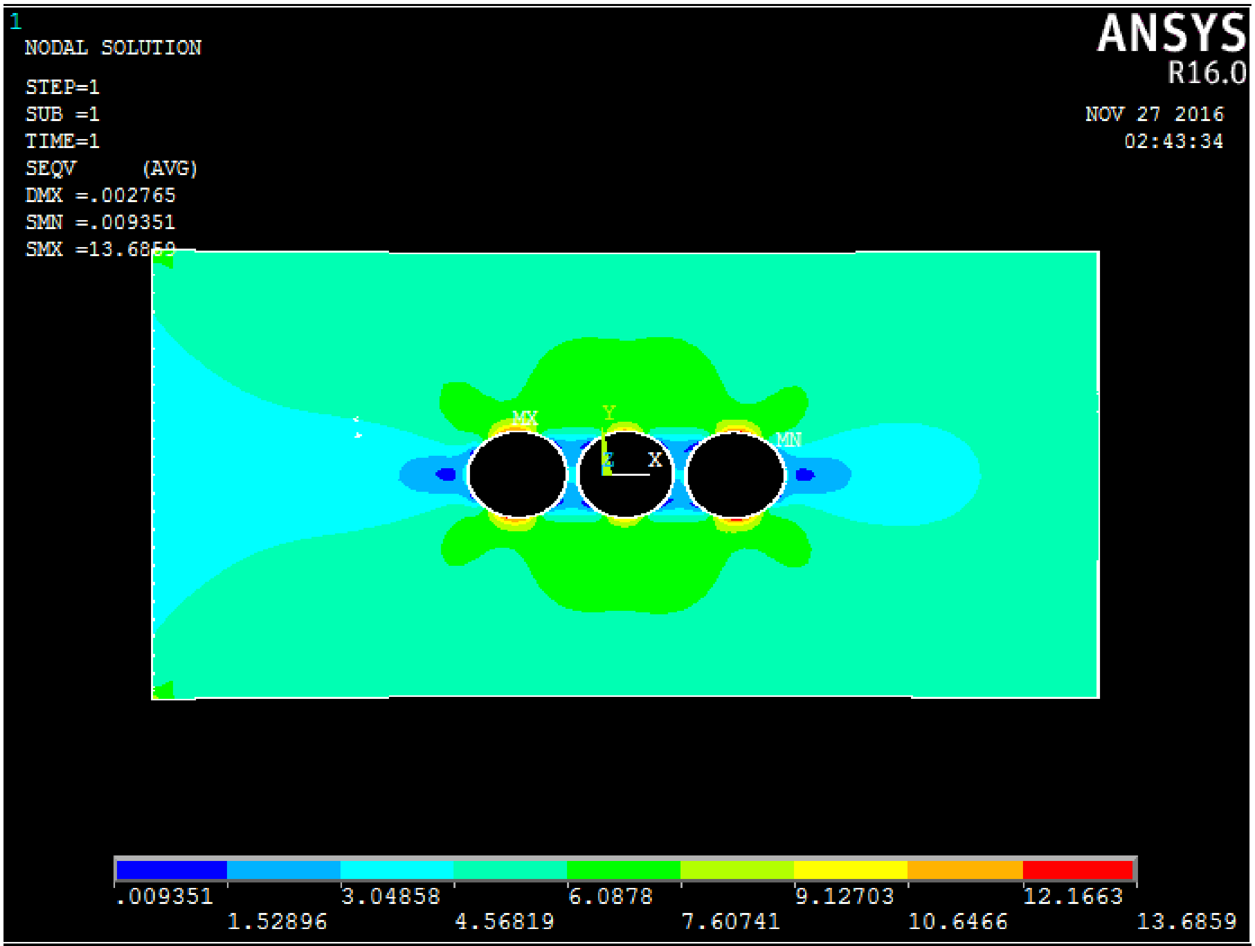
2) SET 2



3) SET 3



(iii) Contour plot of maximum stress concentration



(4.1.2) Six Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

X3, Y3, R3- Centre Co-ordinates & Radius of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[-14 0 4 14 0 4]	[-39 -15 3 11 -15 3]	[-11 15 5 39 15 5]
2	[-12 0 4.5 14 0 4.5]	[-14 -1 4 11 -1 4]	[-11 1 5 14 1 5]
3	[-11.5 0 4.8 11.5 0 4.8]	[-12 -0.5 4.5 11 -0.5 4.5]	[-11 0.5 5 12 0.5 5]

Table 10: UDL1_3H_6D_Boundary Limits

SETS	X2	Y2	R2	X3	Y3	R3	SMAX
1	-14.5065	0.3477	3.854	14.4827	-0.0662	4.3429	12.8525
2	-12.3421	-0.0207	4.4232	13.9665	-0.2796	4.2199	12.7376
3	-11	-0.0096	4.5	11	0.0079	4.5	12.5189

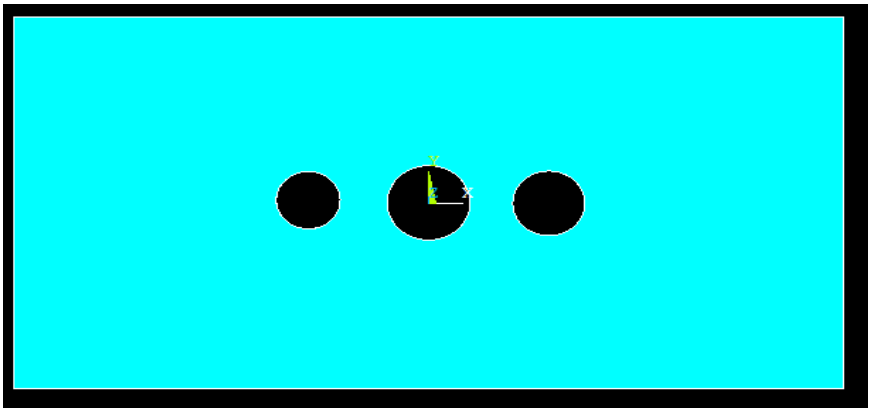
Table 11: UDL1_3H_6D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd & 3rd holes gets closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure. Also, when holes are of different size, when they get closer, they try to attain same size for better stress relieving.

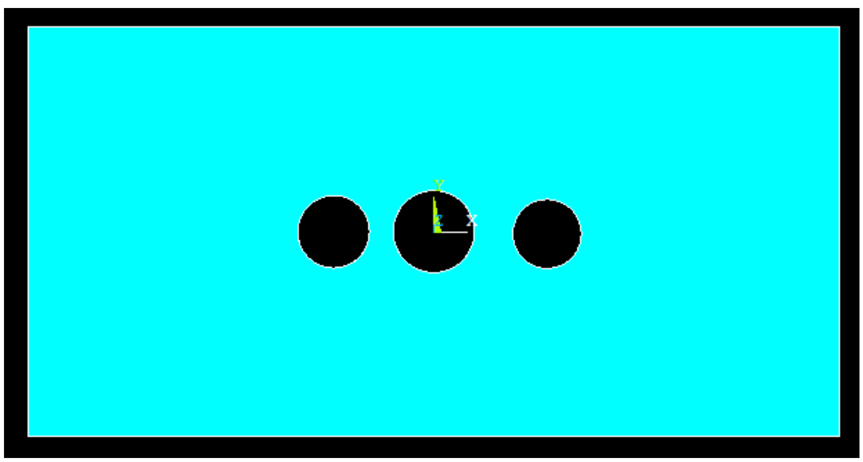
(ii) Plots

The below plots are generated based on values of (X2, Y2, R2) & (X3, Y3, R3) from table 11

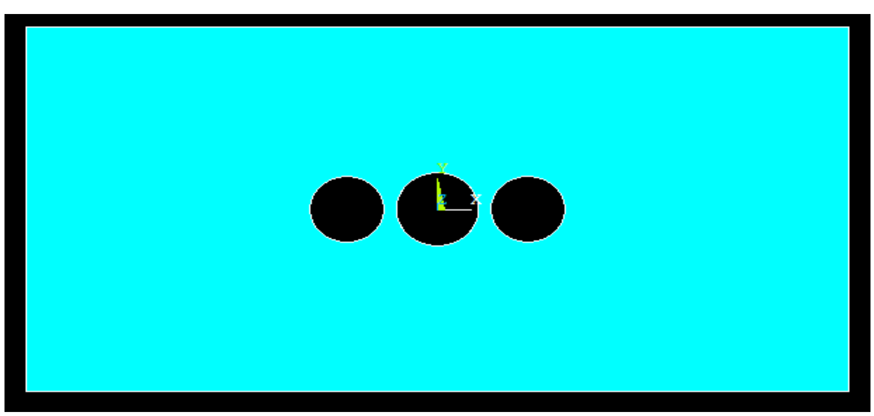
1) SET 1



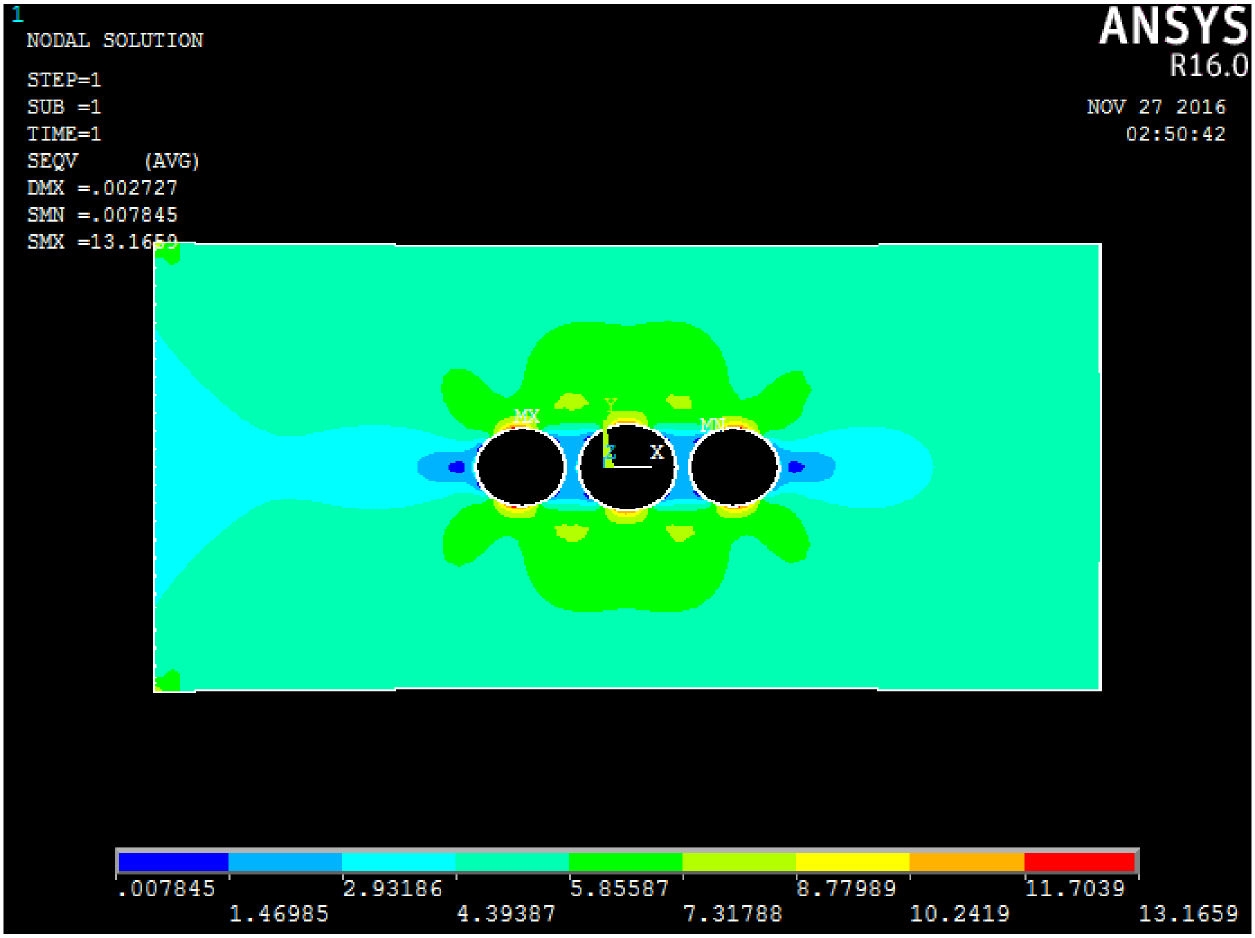
2) SET 2



3) SET 3

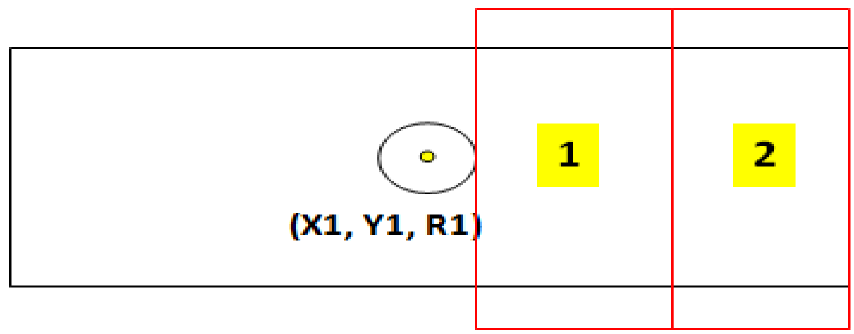


(iii) Contour plot of maximum stress concentration



(4.2) Part 2

For performing optimization, 1st hole is fixed at origin (0, 0) and its radius 'R1' is taken as 5 mm. Co-ordinates of both the holes are varied in positive x-axis. X2: 11 to 20, Y2: -15 to 15 (Section 1), X3: 31 to 40, Y2: -15 to 15 (Section 2) such that only two of these three holes can be closer to each other all the time. It is further pictorially represented below



(4.2.1) Four Design Variables

For this case, radius of 2nd & 3rd holes are taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of 2nd hole,

X3, Y3- Centre Co-ordinates of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[15 0 35 0]	[11 -15 31 -15]	[20 15 40 15]
2	[12 0 32 0]	[11 -15 31 -15]	[20 15 40 15]
3	[19 0 39 0]	[11 -15 31 -15]	[20 15 40 15]
4	[19 0 32 0]	[11 -15 31 -15]	[20 15 40 15]

Table 12: UDL2_3H_4D_Boundary Limits

SETS	X2	Y2	X3	Y3	SMAX
1	16.148	-0.0049	31	-0.0432	14.3992
2	12.4452	-0.0208	31	-0.2896	14.6677
3	18.0855	-0.0052	31	0.0175	14.3952
4	18.9756	-0.3821	31.8761	-0.3013	14.5057

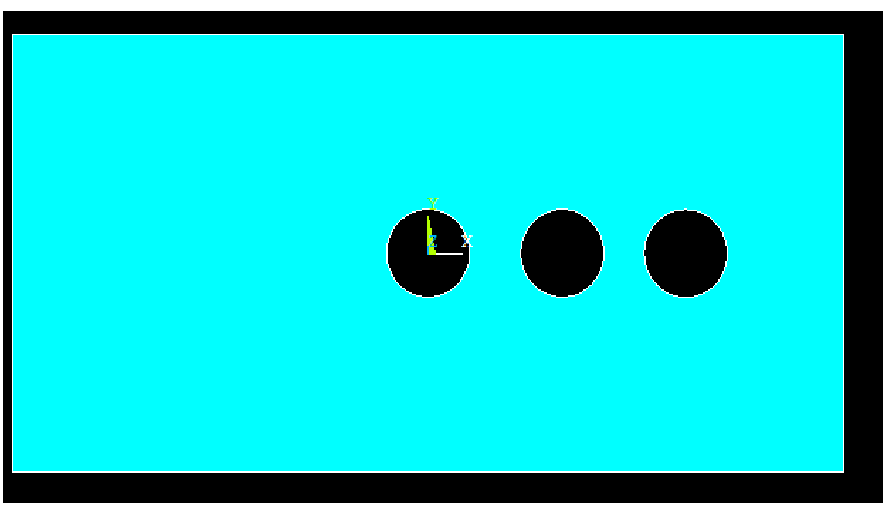
Table 13: UDL2_3H_4D_Optres

From the above optimized values, it is evident that the stress is getting reduced whenever two of the three holes are getting closer. This is because stress relieving is happening when the holes are close to each other. Hence, in case of two holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure.

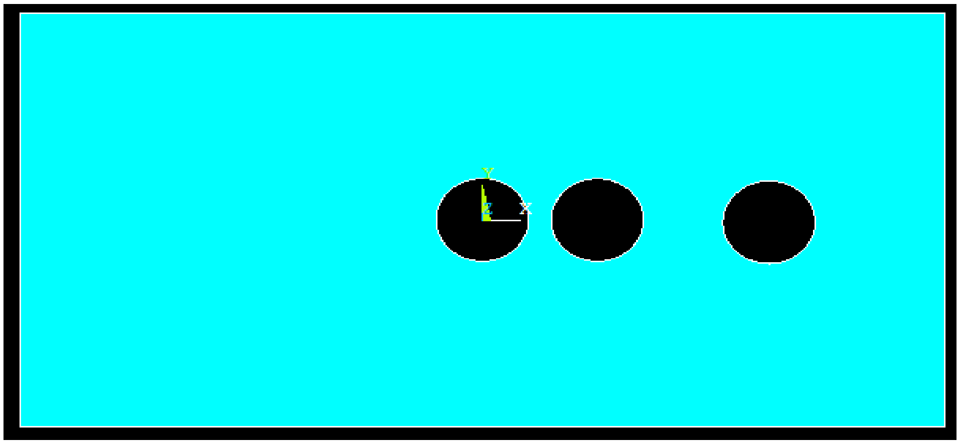
(ii) Plots

The below plots are generated based on values of (X2, Y2) & (X3, Y3) from table 13

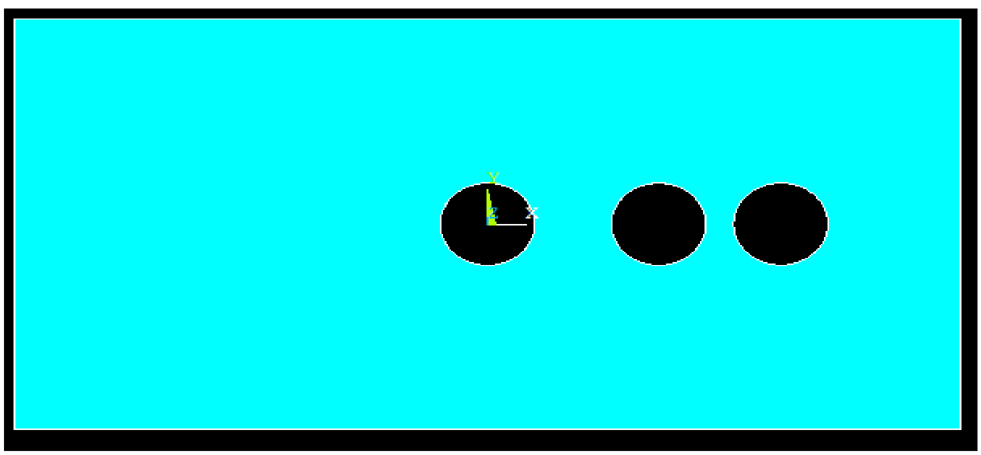
1) SET 1



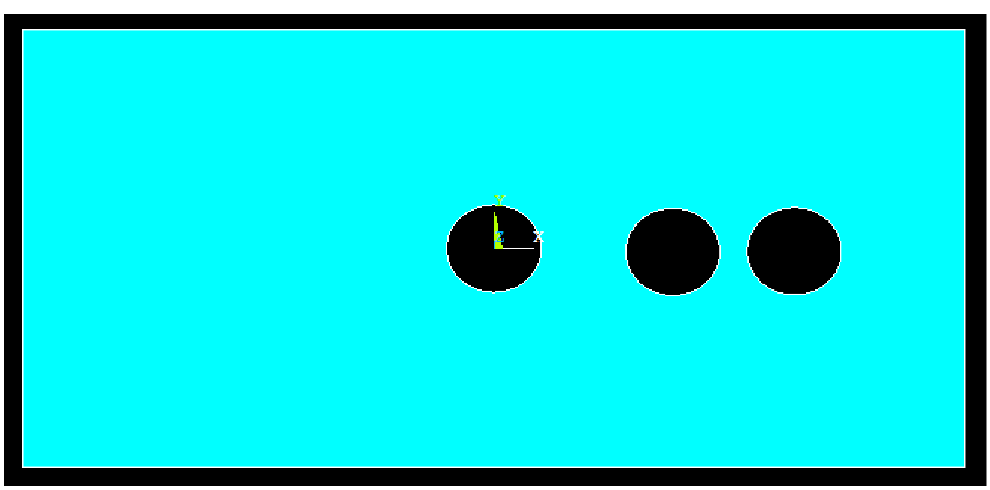
2) SET 2



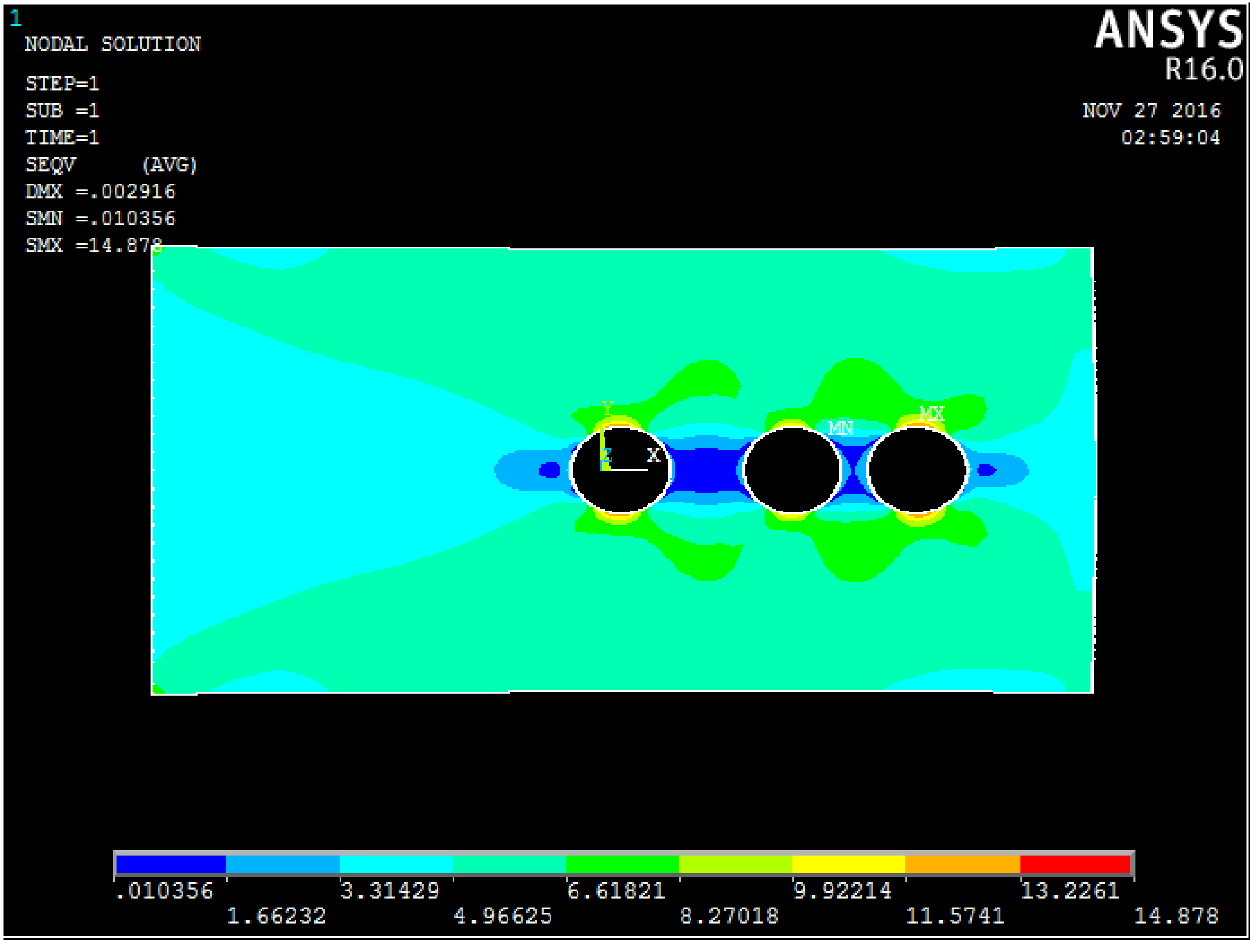
3) SET 3



4) SET 4



(iii) Contour plot of maximum stress concentration



(4.2.2) Six Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

X3, Y3, R3- Centre Co-ordinates & Radius of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[15 0 4 35 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
2	[12 0 4 31 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
3	[19 0 4 39 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
4	[19 0 4 32 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]

Table 14: UDL2_3H_6D_Boundary Limits

SETS	X2	Y2	R2	X3	Y3	R3	SMAX
1	11.4669	0.0295	5	31.4741	0.2974	3	13.3665
2	11.5237	0.0117	5	31.0134	0.0082	3	13.3747
3	18.5013	0.1005	5	36.8309	-0.1162	3	13.7467
4	18.9482	-0.0406	5	31.8511	0.0663	3.2456	13.7803

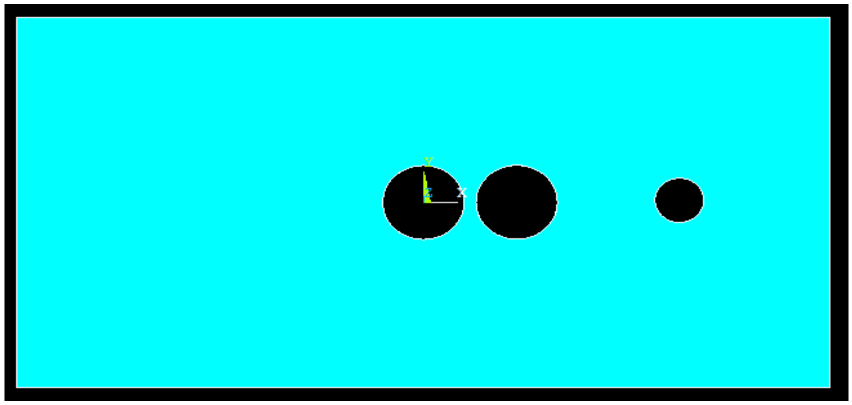
Table 15: UDL2_3H_6D_Optres

From the above optimized values, it is evident that the stress is getting reduced whenever two of the three holes are getting closer. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure. Also, when holes are of different size, when they get closer, they try to attain same size for better stress relieving.

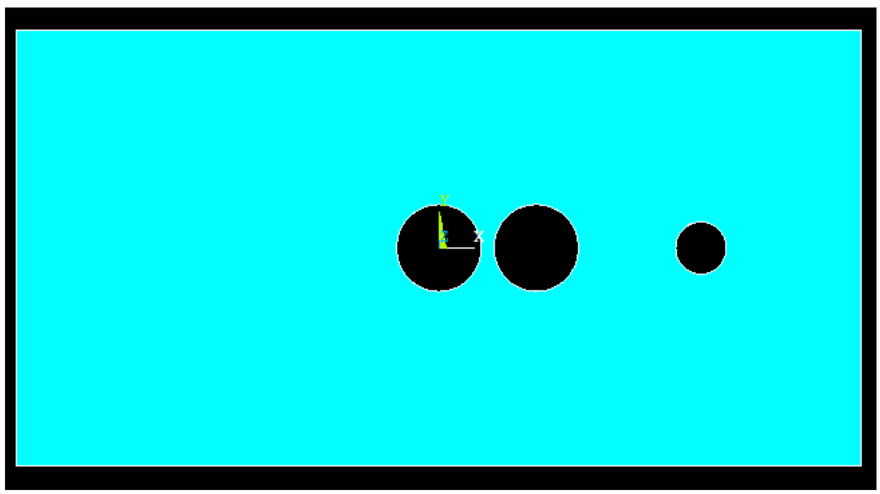
(ii) Plots

The below plots are generated based on values of (X2, Y2, R2) & (X3, Y3, R3) from table 15

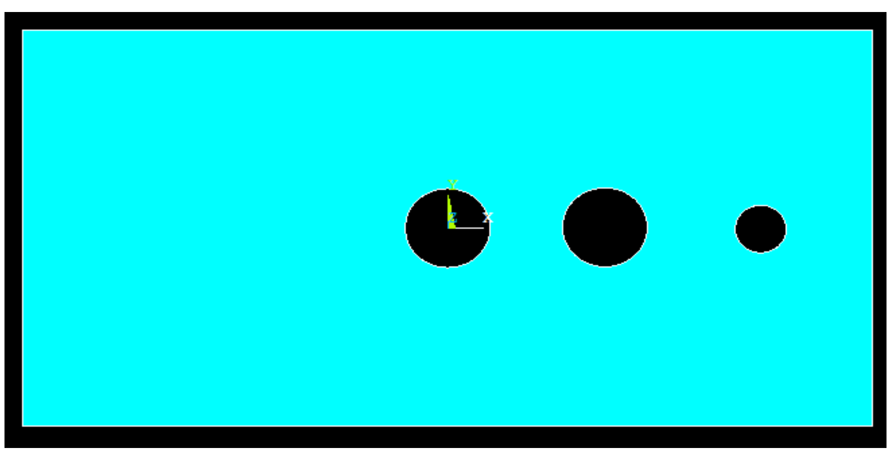
1) SET 1



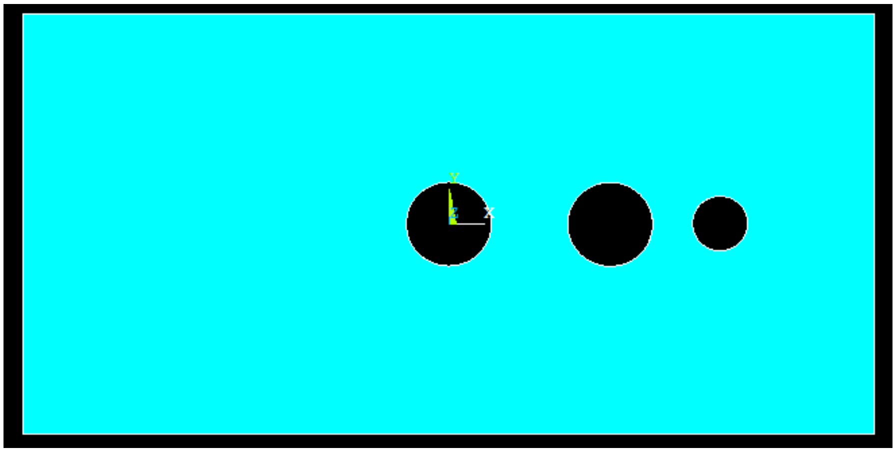
2) SET 2



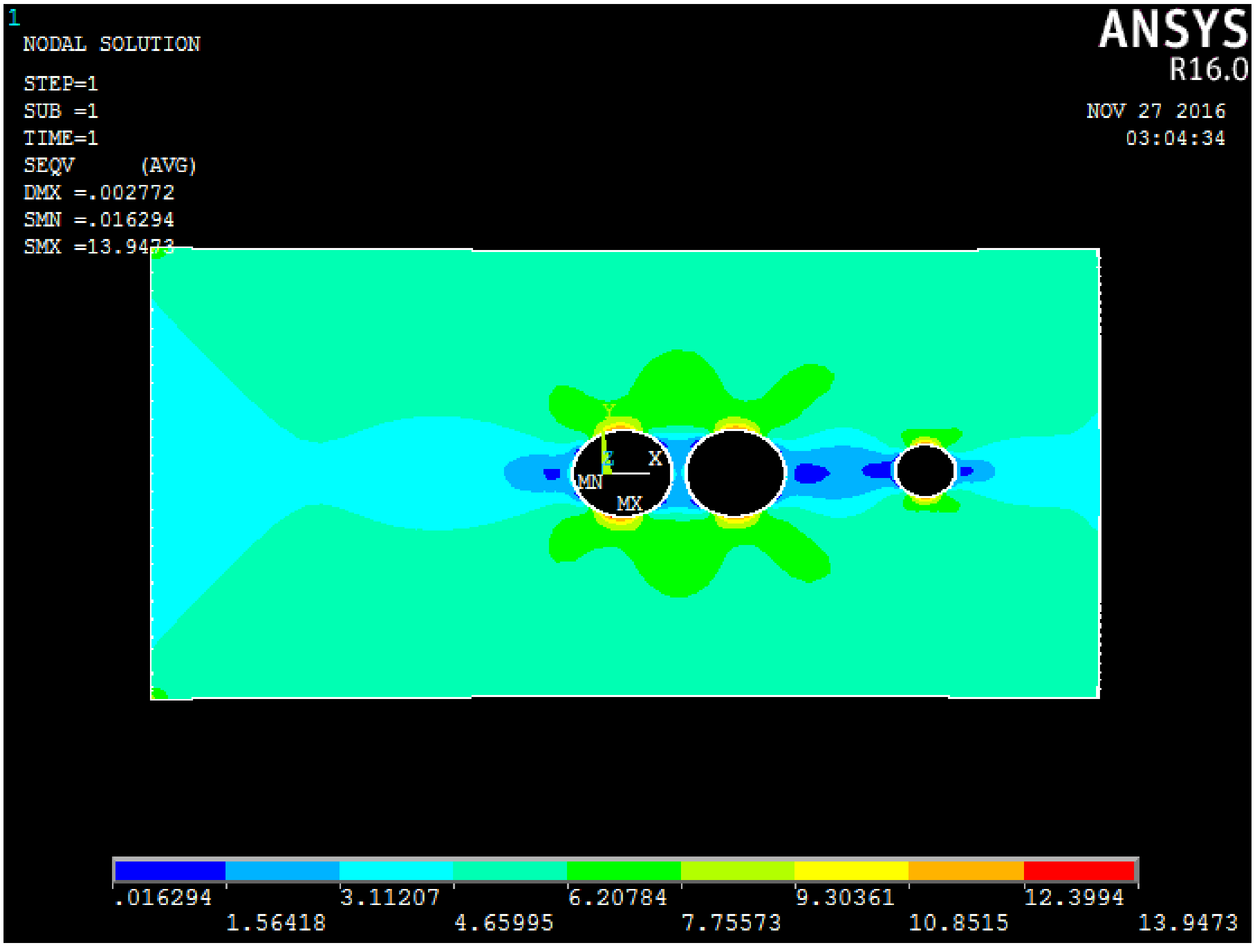
3) SET 3



4) SET 4



(iii) Contour plot of maximum stress concentration



(7) Analytical Solution

Approximate analytical solution for the presence of an infinite number of holes (same size & same distance apart) in a finite width plate is derived by *Schulz (1941)*

(i) Stress Concentration Factor

$$K_{tn} = 3 - \left(0.9916 * \frac{d}{l}\right) - \left(2.5899 * \left(\frac{d}{l}\right)^2\right) + \left(2.2613 * \left(\frac{d}{l}\right)^3\right)$$

d- Diameter of the holes, l- Centre to Centre distance

(ii) Maximum Stress

$$\sigma_n = \frac{\sigma}{(1-\frac{d}{H})} \quad , \quad \sigma_{max} = \sigma * K_{tn}$$

σ_{max} – Maximum Stress developed due to discontinuity in Structure

σ_n – Nominal Stress, σ – Applied Stress

Substituting: d=10 mm, l=11 mm, σ= 5 MPa

We get: **K_{tn}=1.66, σ_n = 6.25 MPa, σ_{max} = 10.4 MPa**

(iii) Comparison

ANALYTICAL	ANSYS	% ERROR
10.4	13.2573	21

From the above result, analytical values are differing from ANSYS results with the error of 21 percentages.

(iv) Notes

l	(d/l)	K_t	σ_{max}
20	0.5	2.14	13.375
11	1	1.68	10.5

(d/l)	K_t	σ_{max}
Decreases	Increases	Increases
Increases	Decreases	Decreases

From the above tables, we can come to conclusion that

1) Whenever d/l ratio is of lower value (i.e. holes are apart), K_t is higher, which in turn increases σ_{max}. So, σ_{max} is directly proportional to K_t

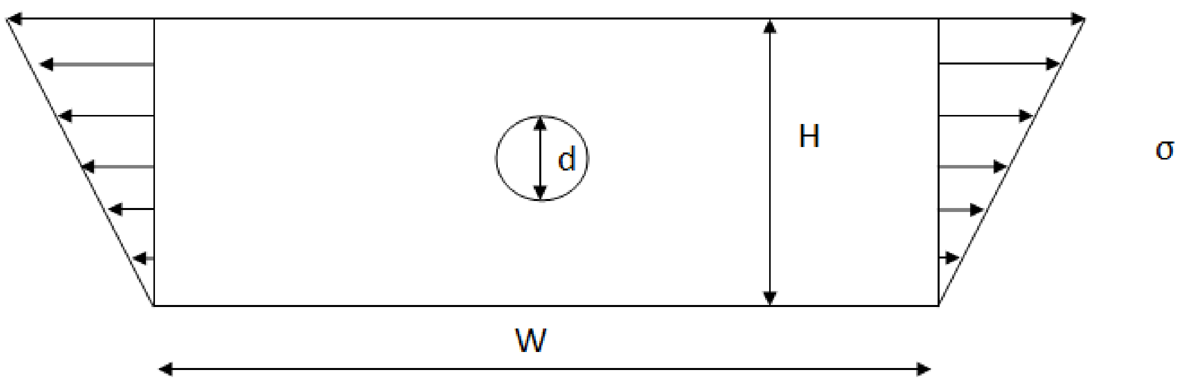
2) Whenever d/l ratio is of higher value (i.e. holes are closer), K_t is lower, which in turn decreases σ_{max}.

4.2 A PLATE WITH LINEARLY VARYING LOAD

4.2.1 Presence of a Single Hole

(1) Actual Model

The pictorial representation of the plate with a hole subjected to Linearly Varying Load (LVL) is given below



.

d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress

Figure 16: LVL _PWSH

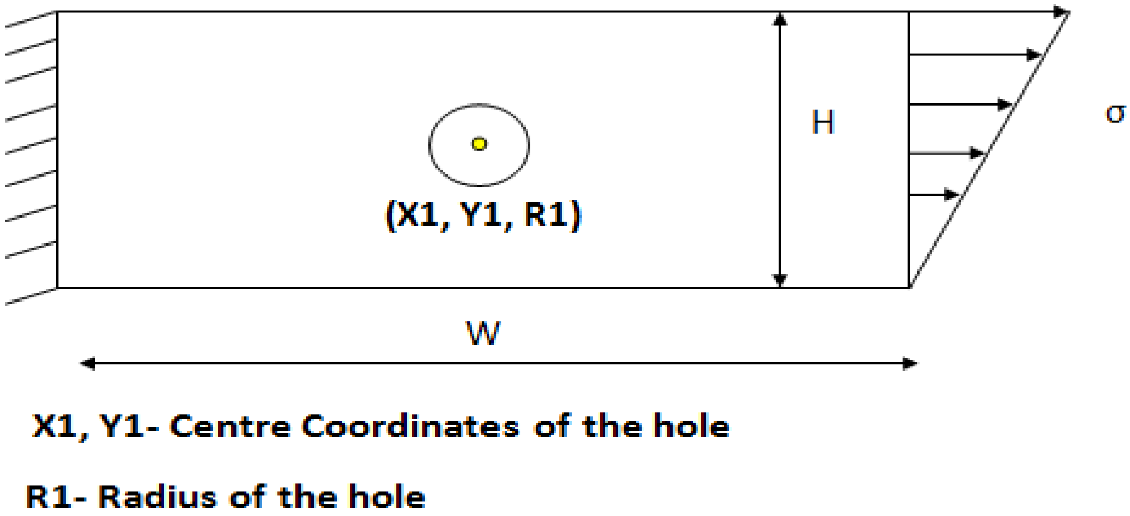
(2) Geometry Details

The geometry details of the plate with a hole subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ: 0 to 10 MPa

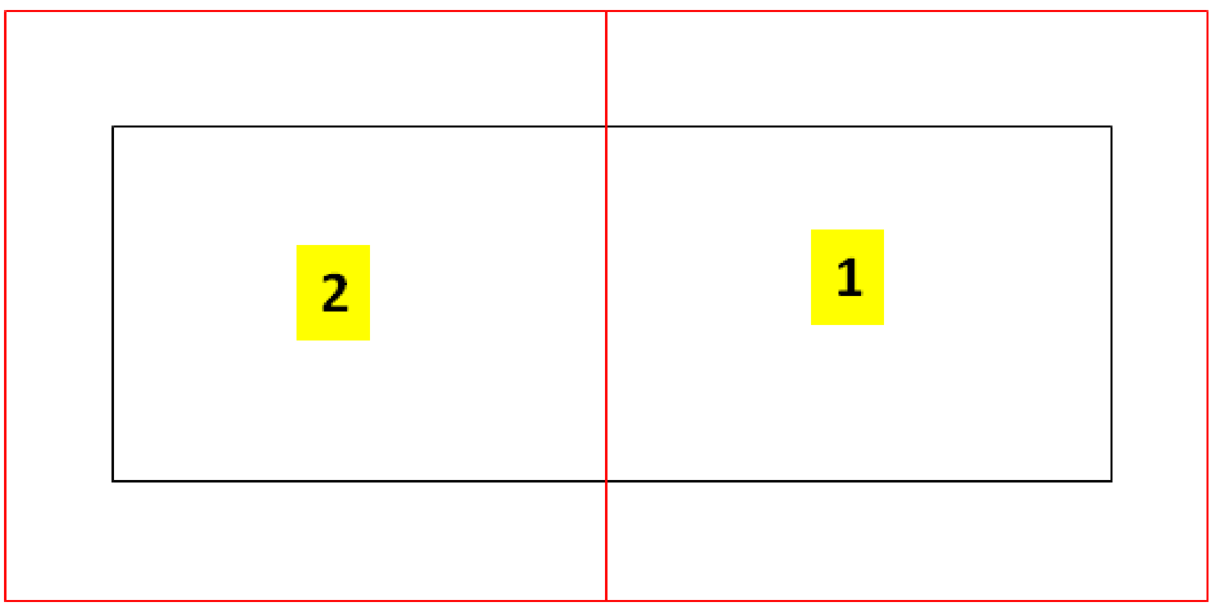
(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables



(4) Contour plot of Maximum Von-Mises Stresses (S_{max})

For generating the contour plot of Maximum Von-Mises Stresses (S_{max}), centre of the plate is at origin (0, 0) and also plate is divided in to two sections as below.



The radius 'R1' of the hole is taken as 5 mm. The location of the hole (X1, Y1) is varied in Section 1 and Section 2 to generate the plot. S_{max} is plotted by having X1 along x-axis and Y1 along y-axis as in the figures below

(i) Section 1: (X1: 0 to 39, Y1: -15 to 15)

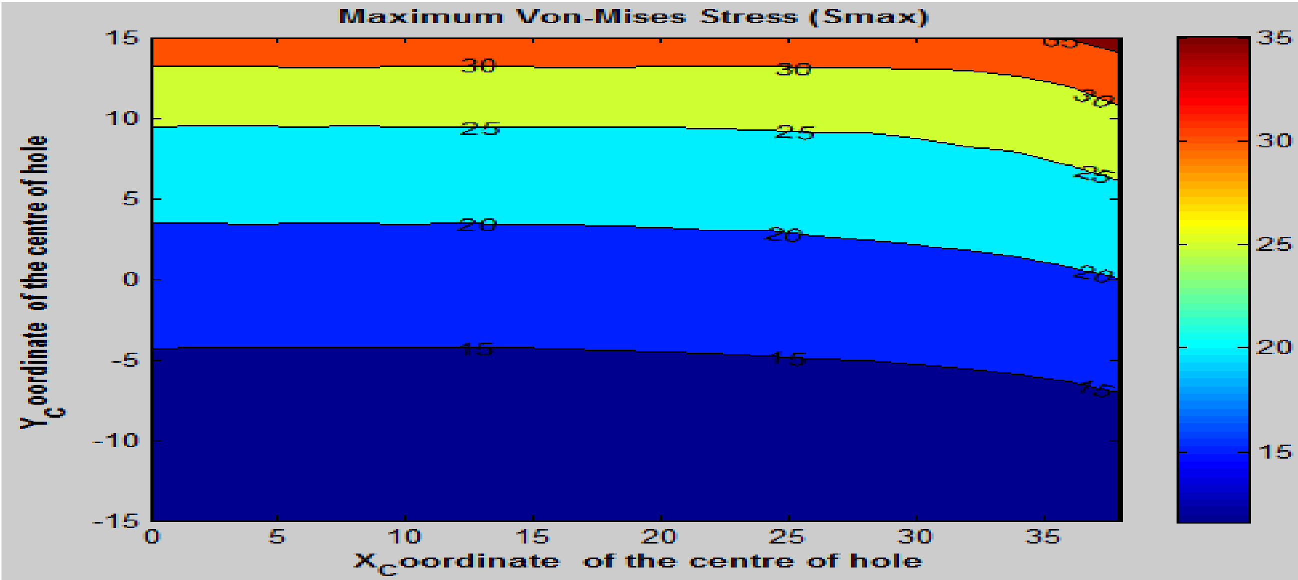


Figure 17: LVL_CPSH_S1

(ii) Section 2: (X1: -39 to 0, Y1: -15 to 15)

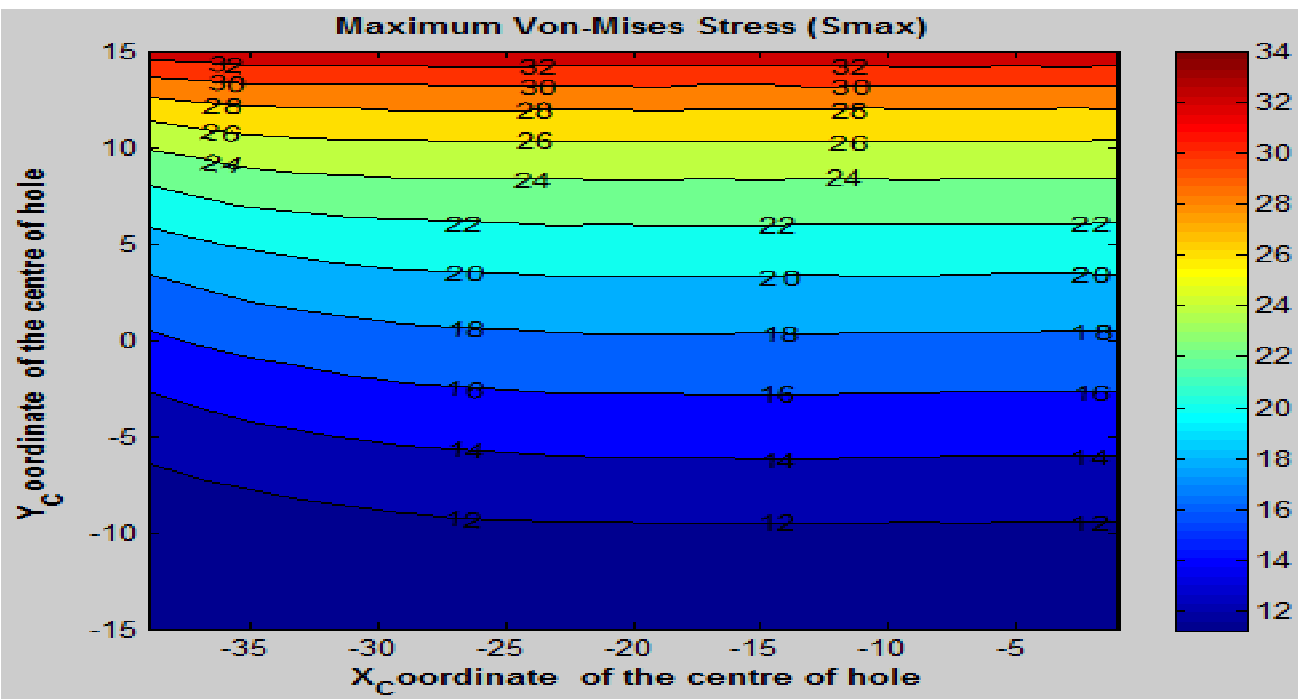


Figure 18: LVL_CPSH_S2

It is understood from the above plots that Section 1 & Section 2 are symmetric to each other. So, Optimization is carried out in Section 1 to find the best location at which stress acting is minimum which is being covered in the following pages

(5) Optimization

For performing optimization, radius 'R1' of the hole is taken as 5 mm. Co-ordinates (X1, Y1) is varied in Section 1 with the limits, X1: 0 to 39, Y1: -15 to 15

Design Variables: X1, Y1- Centre Co-ordinates of the hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints, Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[5 -5]	[0 -15]	[39 15]
2	[1 -1]	[0 -5]	[7 5]
3	[1 -1]	[0 -1]	[1 1]

Table 16: LVL_1H_2D_Boundary Limits

SETS	X1	Y1	SMAX
1	5.03398	-13.5405	11.6008
2	1.1856	-4.9999	14.4746
3	0.9567	-0.9999	16.9643

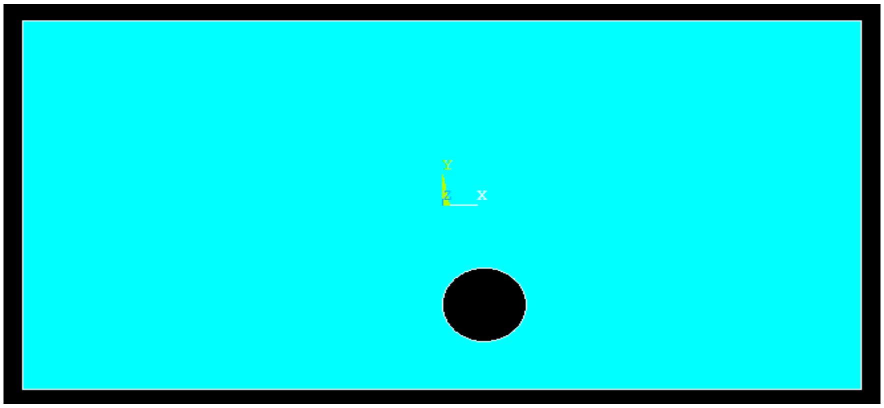
Table 17: LVL_1H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as hole approaches bottom edge of the plate. Hence the best location for a Plate with Single hole subjected to Linearly Varying load is near the region where the load acting is less as it varies from '0' at bottom edge to '10' at the top edge. Also figures 17 & 18, stress is proportional to load and is less near zero load regions

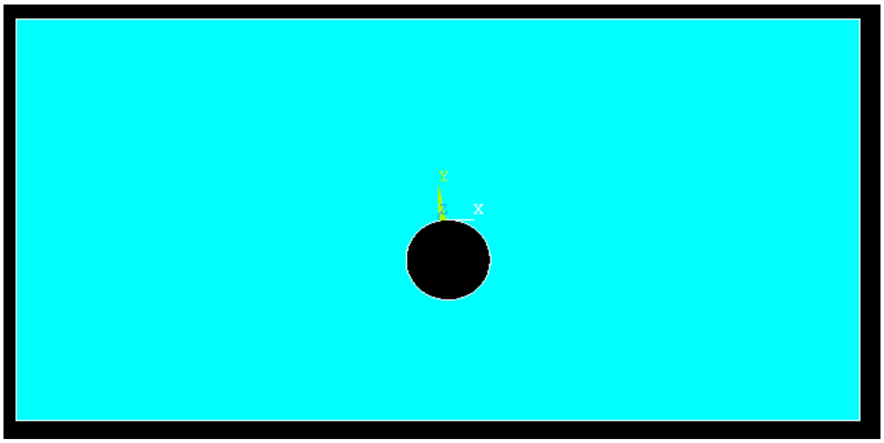
(ii) Plots

The below plots are generated based on values of (X1, Y1) from table 17

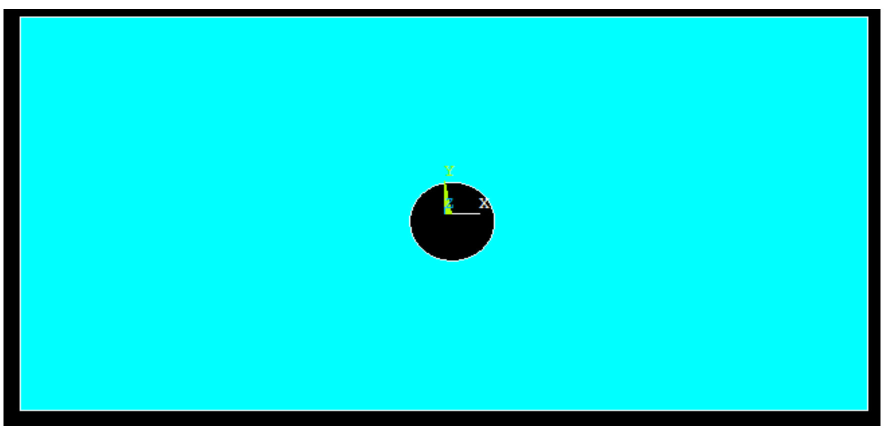
1) SET 1



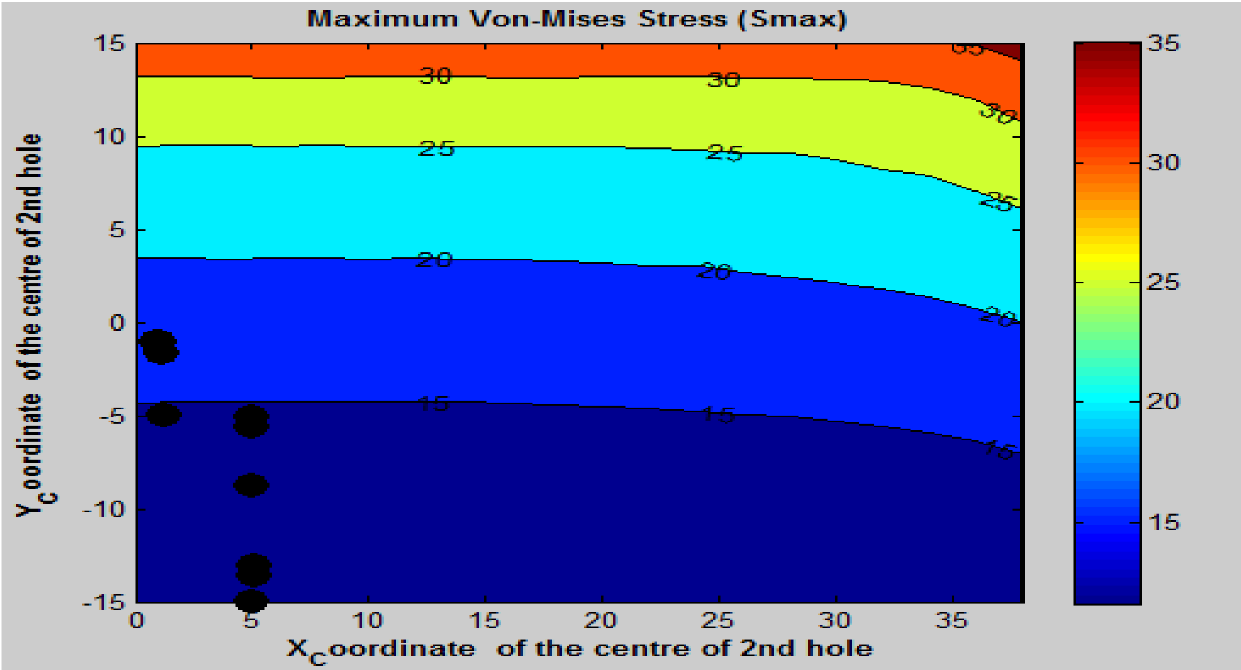
2) SET 2



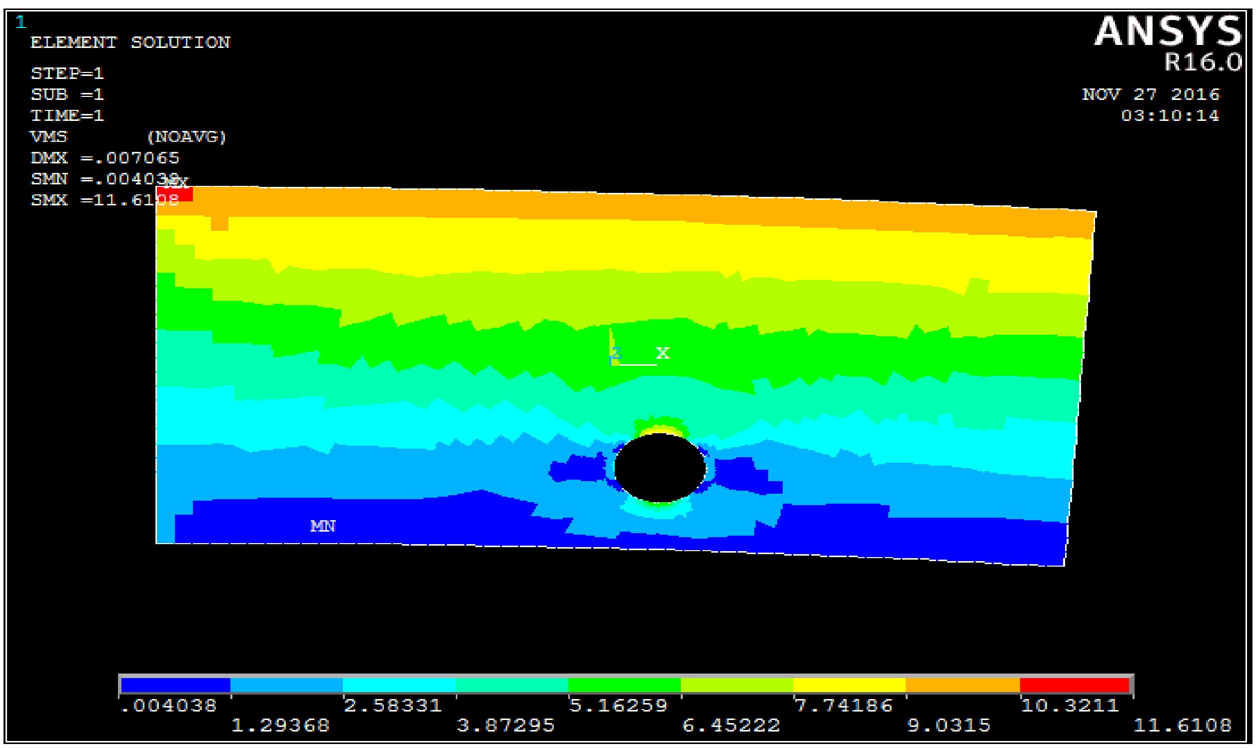
3) SET 3



(iii) Stress Plot with Design Points



(iv) Contour plot of maximum stress concentration



(6) Maximum Von-mises Stress Vs Radius of the Hole
For generating below plot, Hole is fixed at origin (i.e. X1=Y1=0). Only radius of the hole 'R1' is varied to get below plot

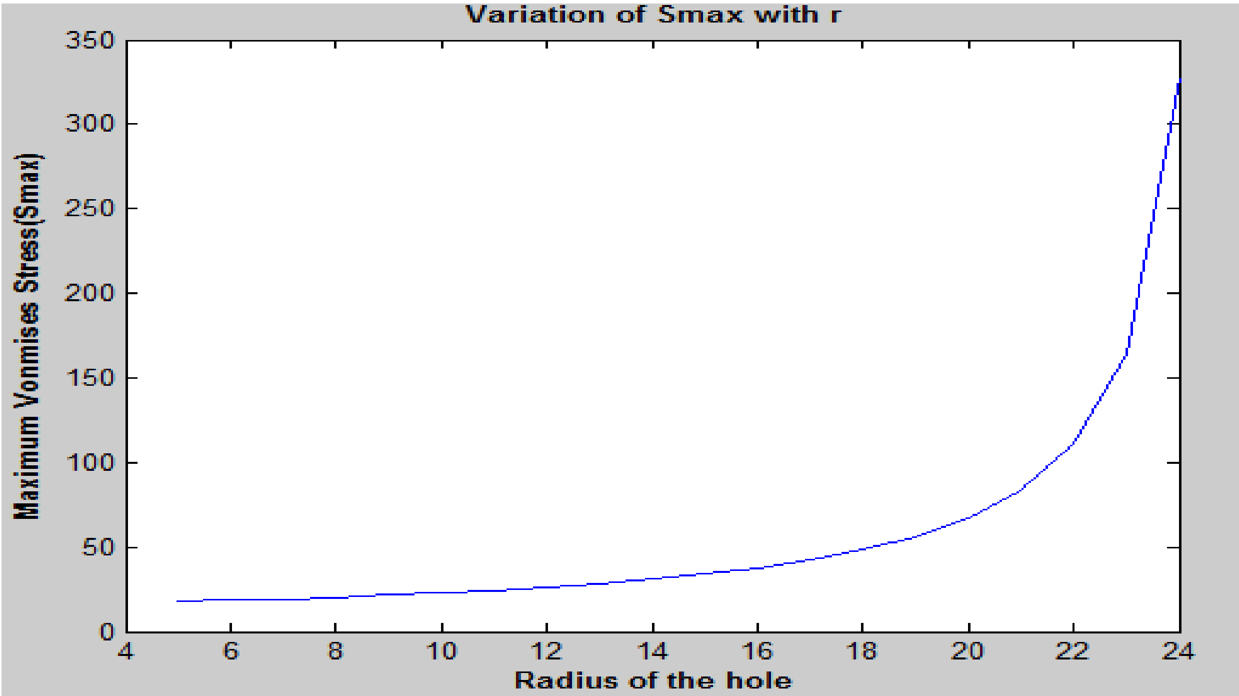


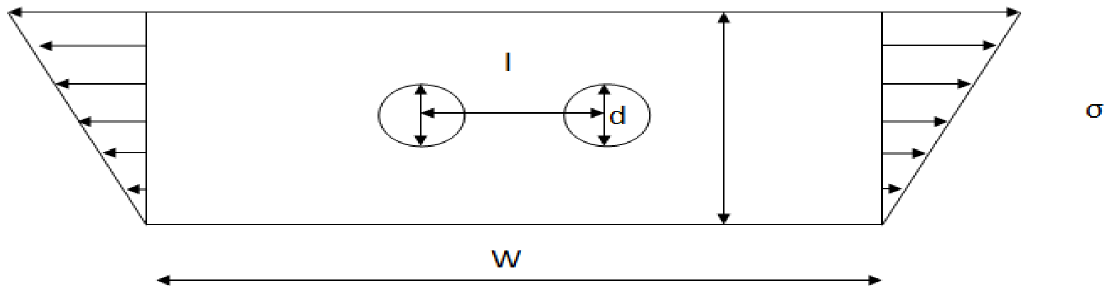
Figure 19: LVL_Smax Vs R1

From the above plot, it is understood that the stress is getting increased whenever the hole size increases

4.2.2 Presence of Two Holes

(1) Actual Model

The pictorial representation of the plate with two holes subjected to Linearly Varying Load (LVL) is given below



d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress, l- centre to centre distance

Figure 20: LVL_PW2H

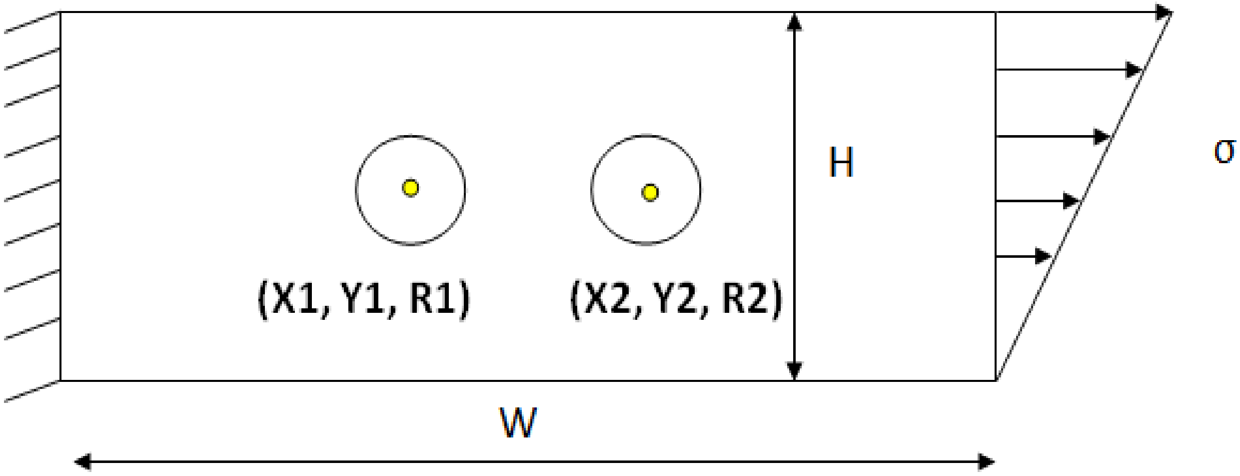
(2) Geometry Details

The geometry details of the plate with two holes subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ: 0 to 10 MPa

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables

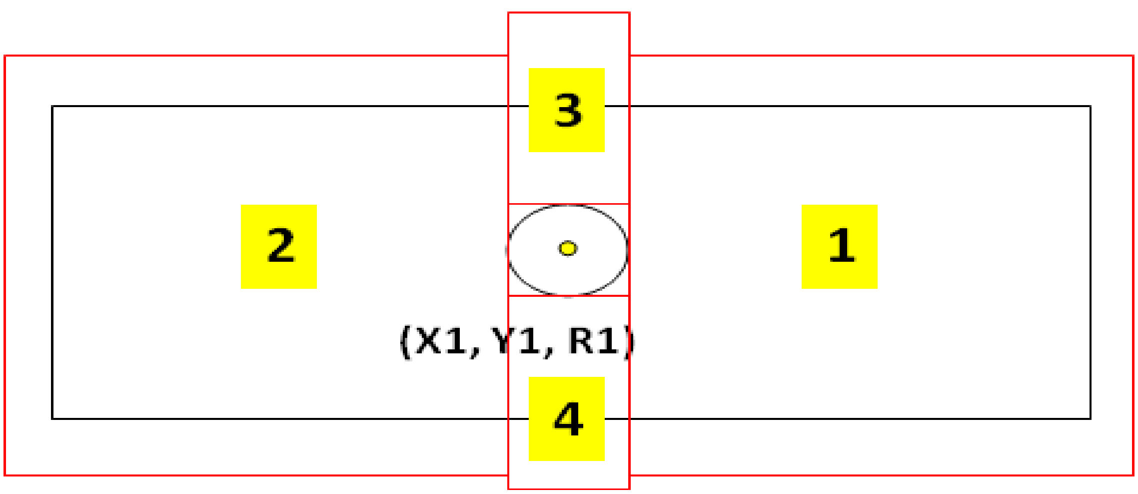


X1, Y1, R1- Center Coordinates & Radius of 1st hole

X2, Y2, R2- Center Coordinates & Radius of 2nd hole

(4) Contour plot of Maximum Von-Mises Stresses (S_{max})

1st hole is fixed at centre of the plate (i.e. at origin). In order to generate the contour plot of Maximum Von-Mises Stresses (S_{max}), plate is divided in to four sections as below.



The radius 'R1' & 'R2' of the holes are taken as 5 mm. The location of 2nd hole (X2, Y2) is varied in the above sections to generate the plot. S_{max} is plotted by having X2 along x-axis and Y2 along y-axis as in the figures below

(i) Section 1: (X1: 11 to 39, Y1: -15 to 15)

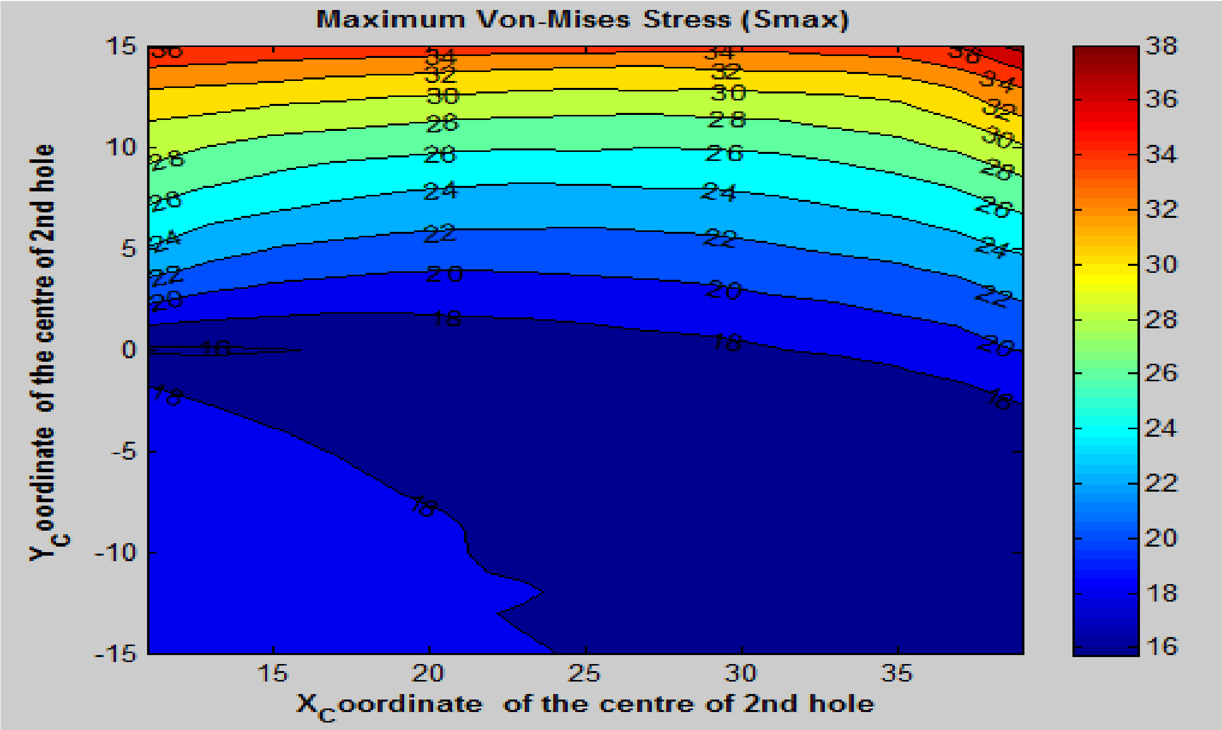


Figure 21: LVL_CP2H_S1

(ii) Section 2: (X1: -39 to -11, Y1: -15 to 15)

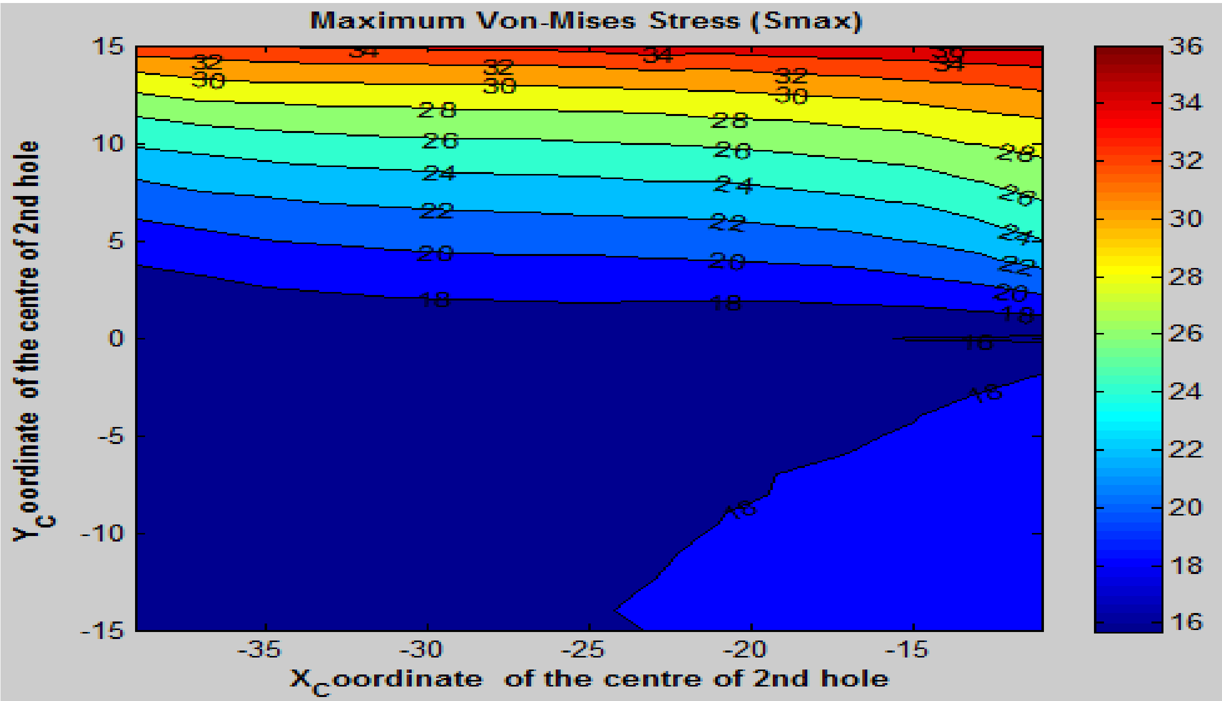


Figure 22: LVL_CP2H_S2

(iii) Section 3: (X1: -10 to 10, Y1: 11 to 15)

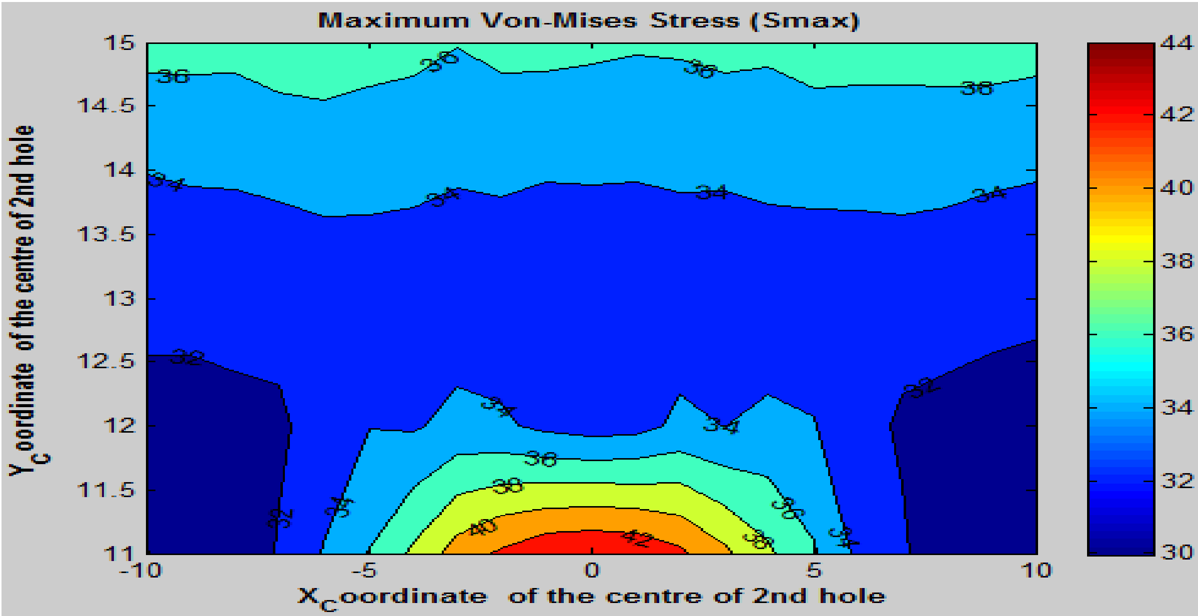


Figure 23: LVL_CP2H_S3

(iv) Section 4: (X1: -10 to 10, Y1: -15 to -11)

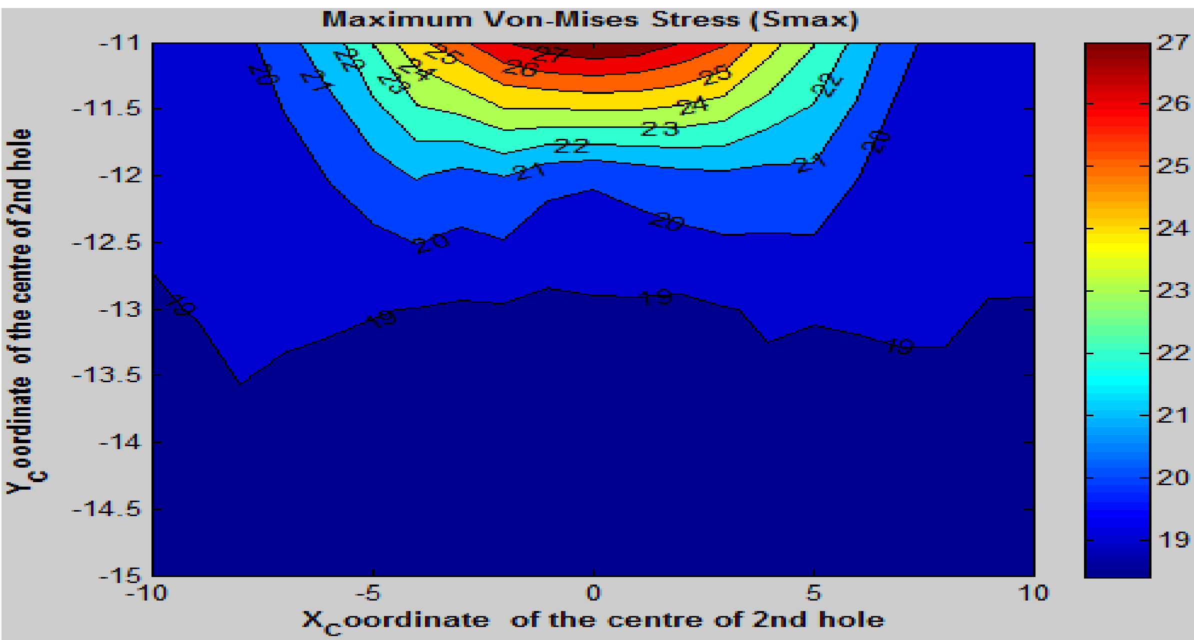


Figure 24: LVL_CP2H_S4

It is understood from the above plots that Sections 1 & 2, Sections 3 & 4 are symmetric to each other. Stress is less in Sections 1 & 2 comparatively. So, Optimization is carried out in Section 1 to find the best location at which stress acting is minimum which is being covered in the following pages

(5) Optimization

For performing optimization, 1st hole is fixed at origin (0, 0) and its radius 'R1' is taken as 5 mm. Co-ordinates (X2, Y2) is varied in Section 1 with the limits, X2: 11 to 39, Y2: -15 to 15

(5.1) Two Design Variables

For this case, radius of 2nd hole 'R2' is taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of the hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints, Upper & Lower Bounds of the design Variables

(i) FMINCON results:

SETS	X	XL	XU
1	[13 -5]	[11 -15]	[39 15]

Table 18: LVL_2H_2D_Boundary Limits

SETS	X2	Y2	SMAX
1	11.7858	-0.0478	15.6604

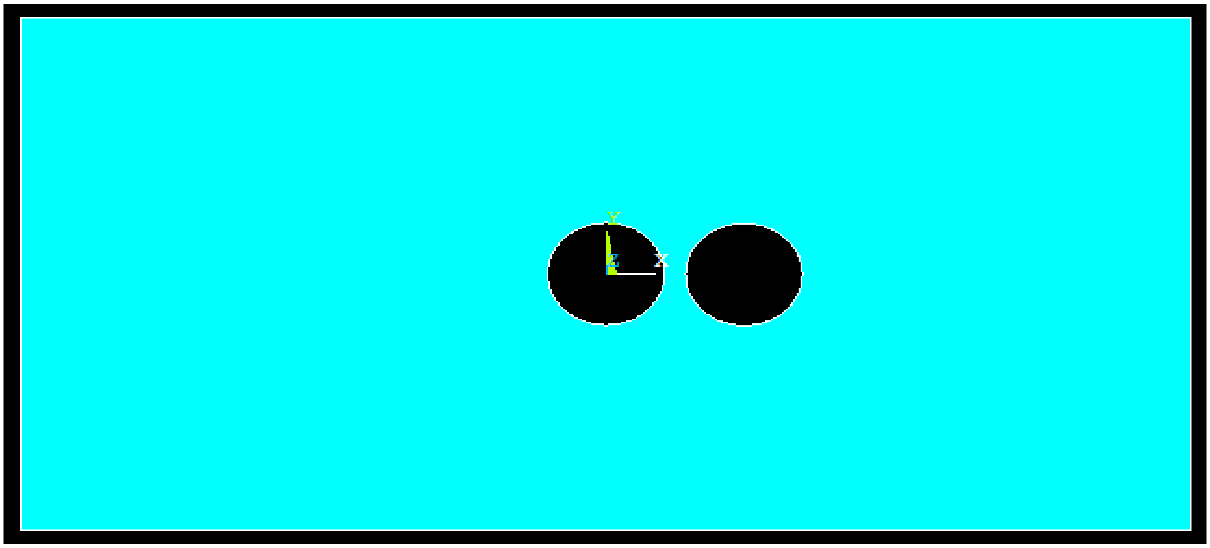
Table 19: LVL_2H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd hole goes closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of two holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure.

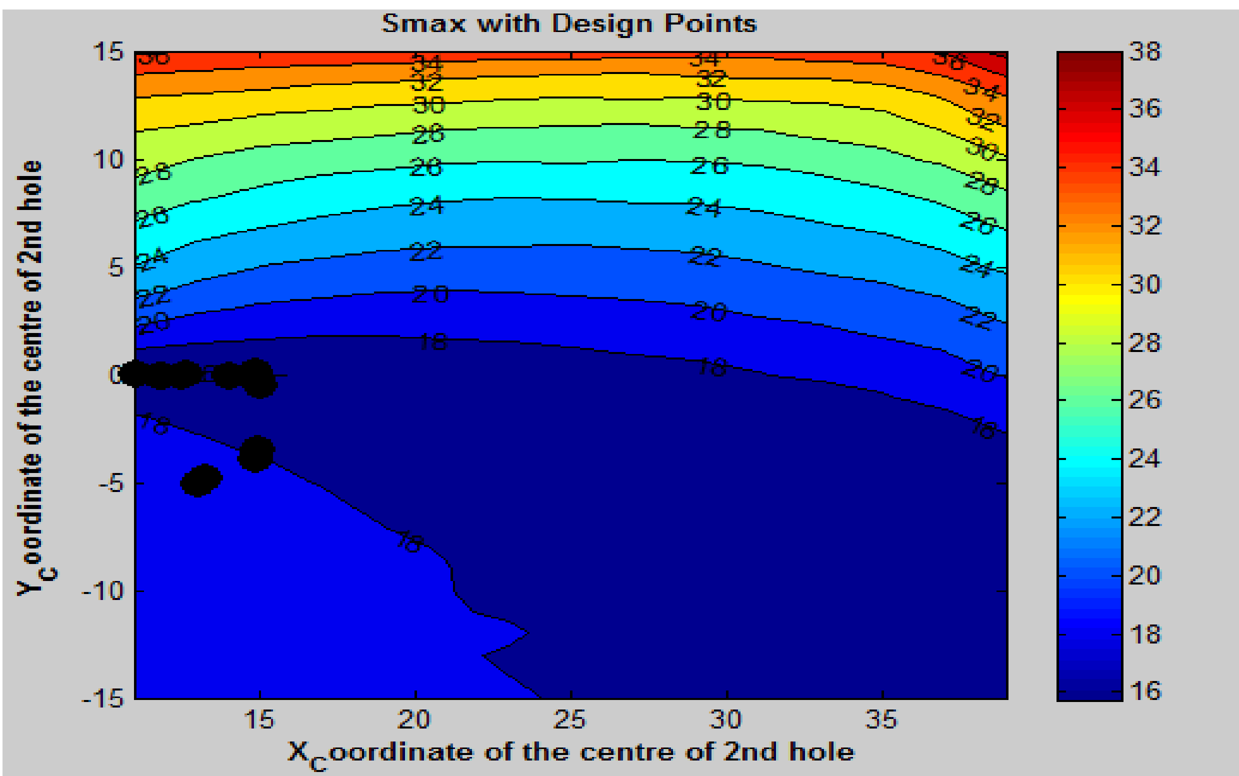
(ii) Plots:

The below plots are generated based on values of (X2, Y2) from table 19

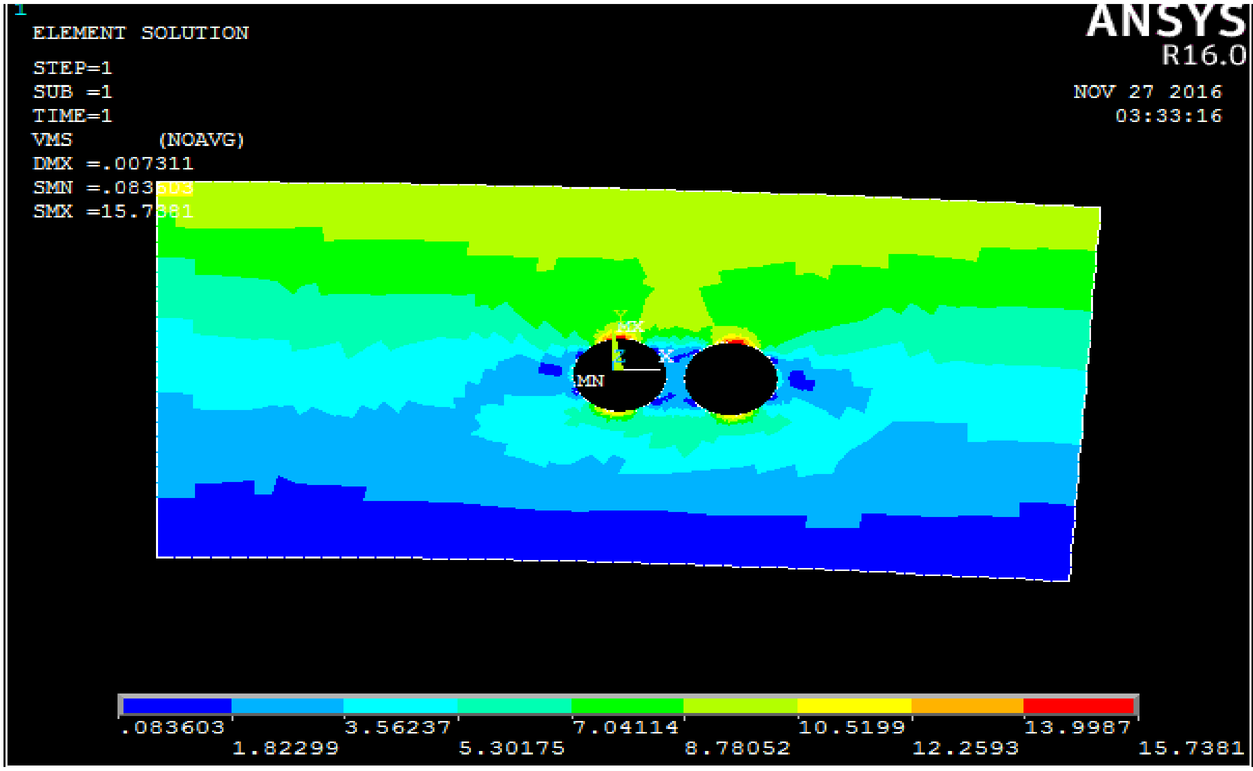
1) SET 1



(iii) Stress Plot with Design Points:



(iv) Contour plot of maximum stress concentration



(5.2) Three Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[13 -5 3.3]	[11 -15 3]	[15 15 5]

Table 20: LVL_2H_3D_Boundary Limits

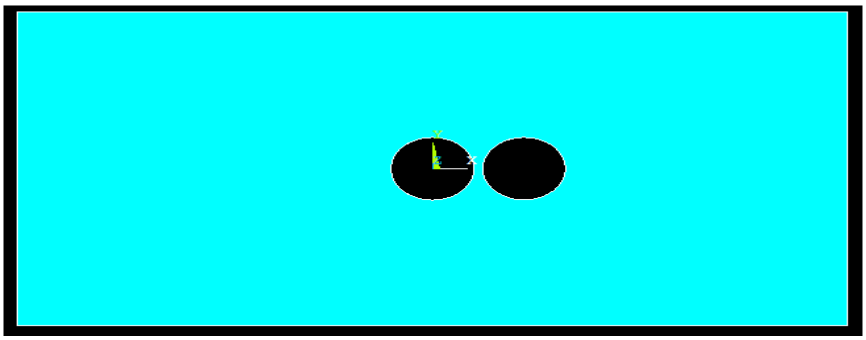
SETS	X2	Y2	R2	SMAX
1	11.0265	0.01716	4.9869	15.6429

Table 21: LVL_2H_3D_Optres

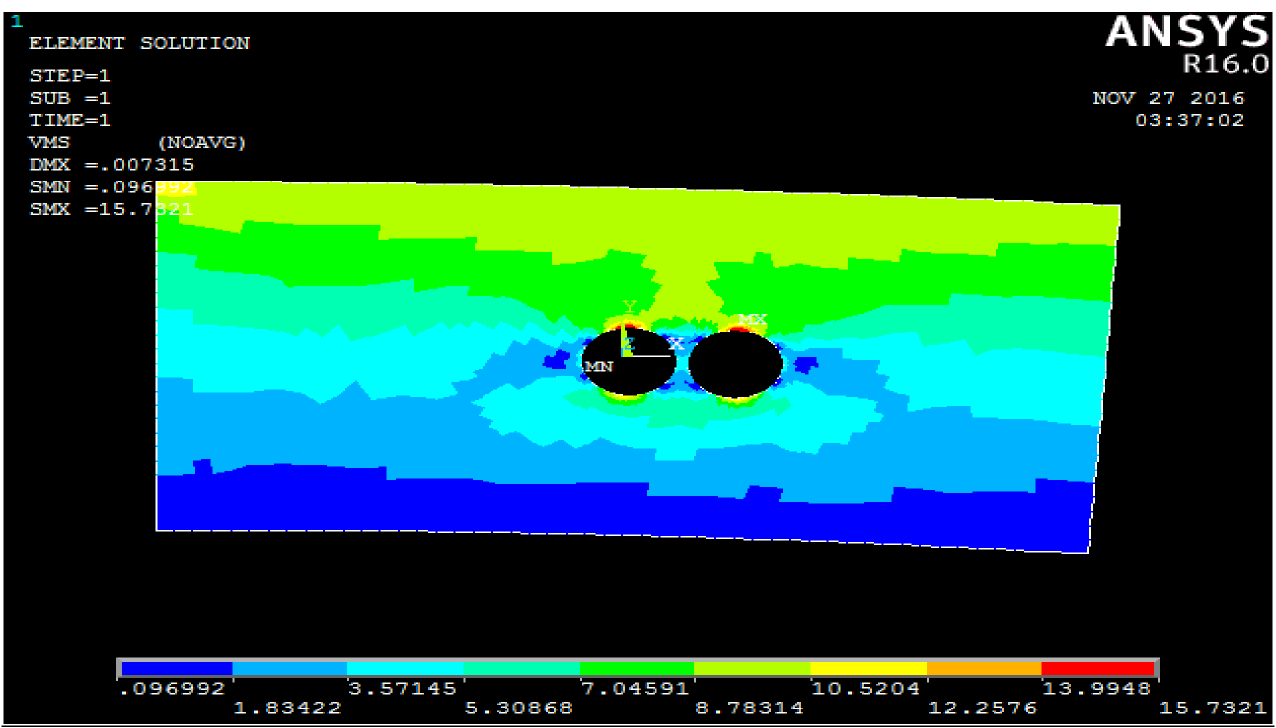
(ii) Plots

The below plot is generated based on values of (X2, Y2) from table 21

1) SET 1



(iii) Contour plot of maximum stress concentration



(6) Maximum Von-mises Stress Vs X- Coordinate of 2nd hole

For generating below plot, 1st Hole is fixed at origin (i.e. X1=Y1=0). Radius of the holes ‘R1’ & ‘R2’ are taken as 5 mm, 2nd hole is made collinear with 1st hole (i.e Y2=0 for all values of X2). Only X2 is varied to generate below plot

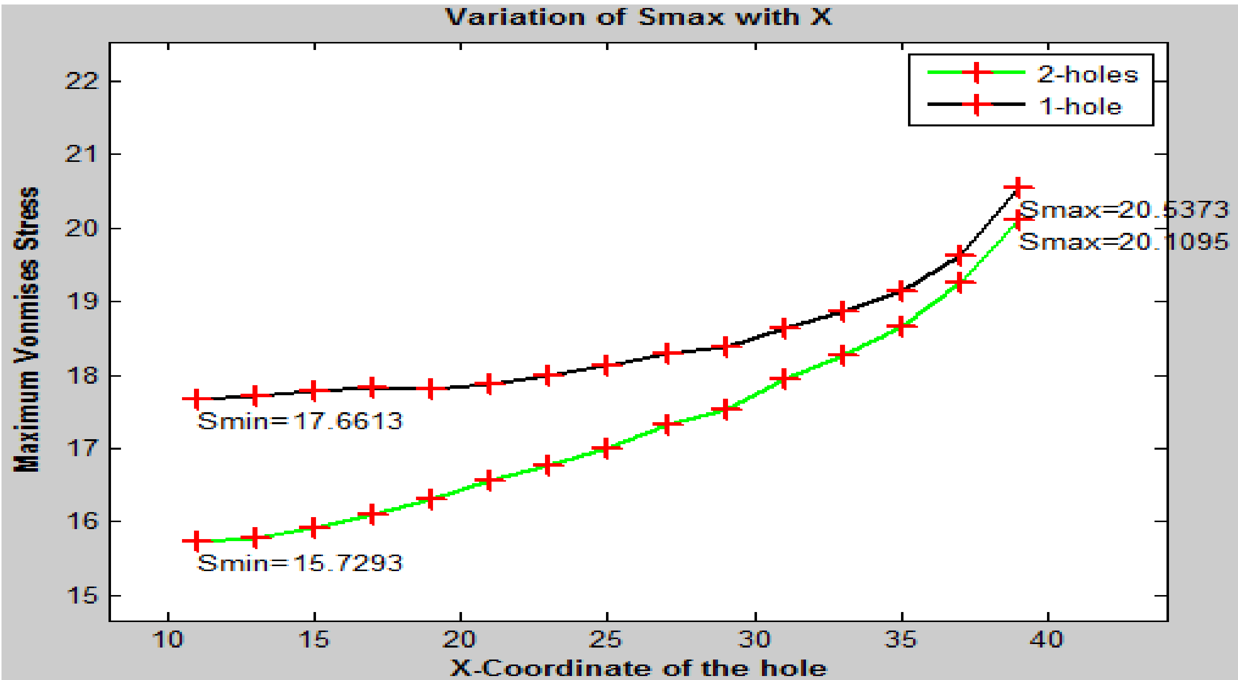


Figure 25: LVL_Smax Vs X2

(7) Maximum Von-mises Stress Vs Radius of 2nd hole

For generating below plot, 1st hole is fixed at origin (i.e. X1=Y1=0). Radius of 1st hole is taken as 5 mm, 2nd hole is made collinear with 1st by fixing it at X2=11, Y2=0. Only 'R2' is varied to generate below plot

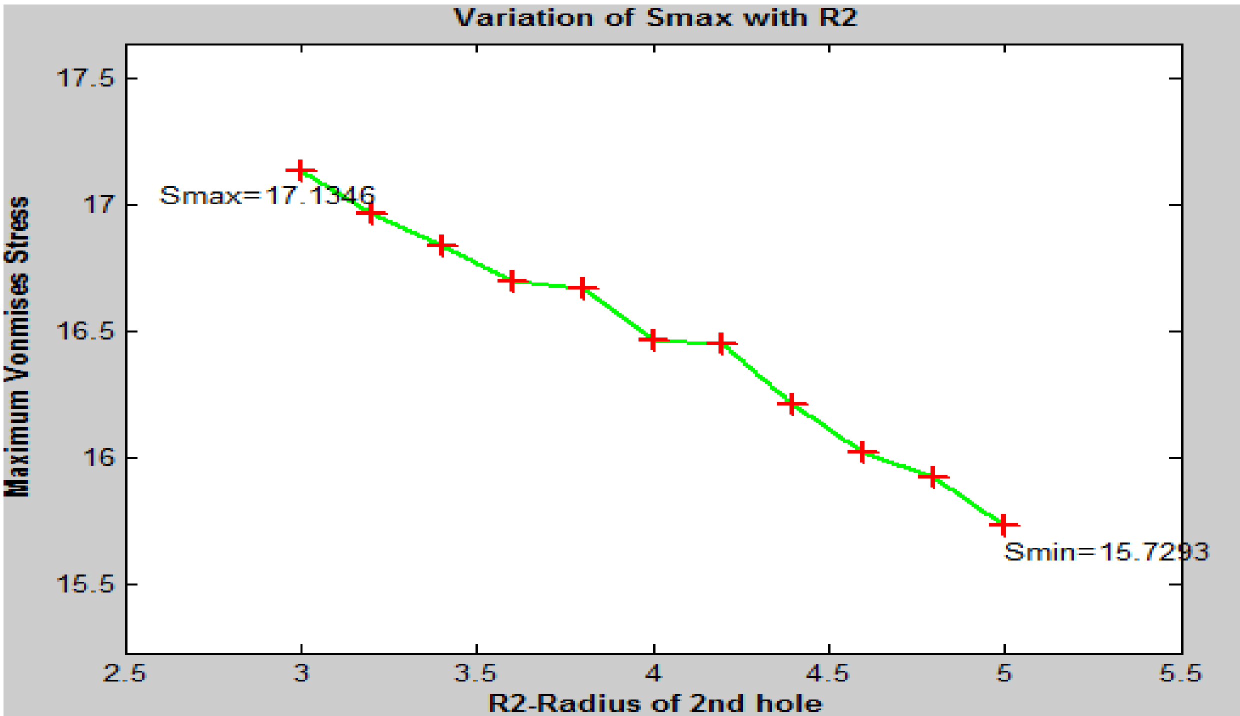


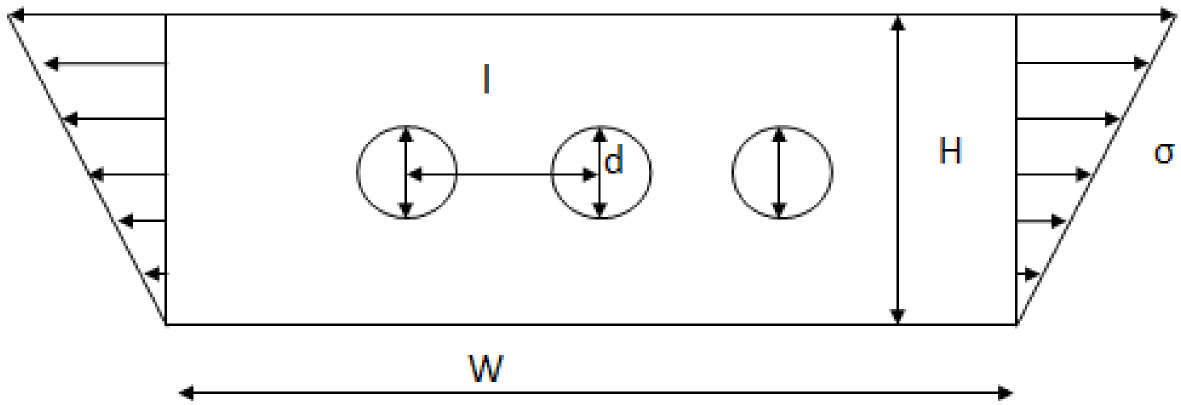
Figure 26: LVL_Smax Vs R2

From figures 25 & 26, it is evident that stress is getting relieved & drops to a lower value only when the holes are of same size and close to each other

4.2.3 Presence of Three Holes

(1) Actual Model

The pictorial representation of the plate with three holes subjected to Linearly Varying Load (LVL) is given below



d- diameter of the hole, H- Width of the plate, W- Length of the plate

σ- Applied Stress, l- centre to centre distance

Figure 27: LVL_PW3H

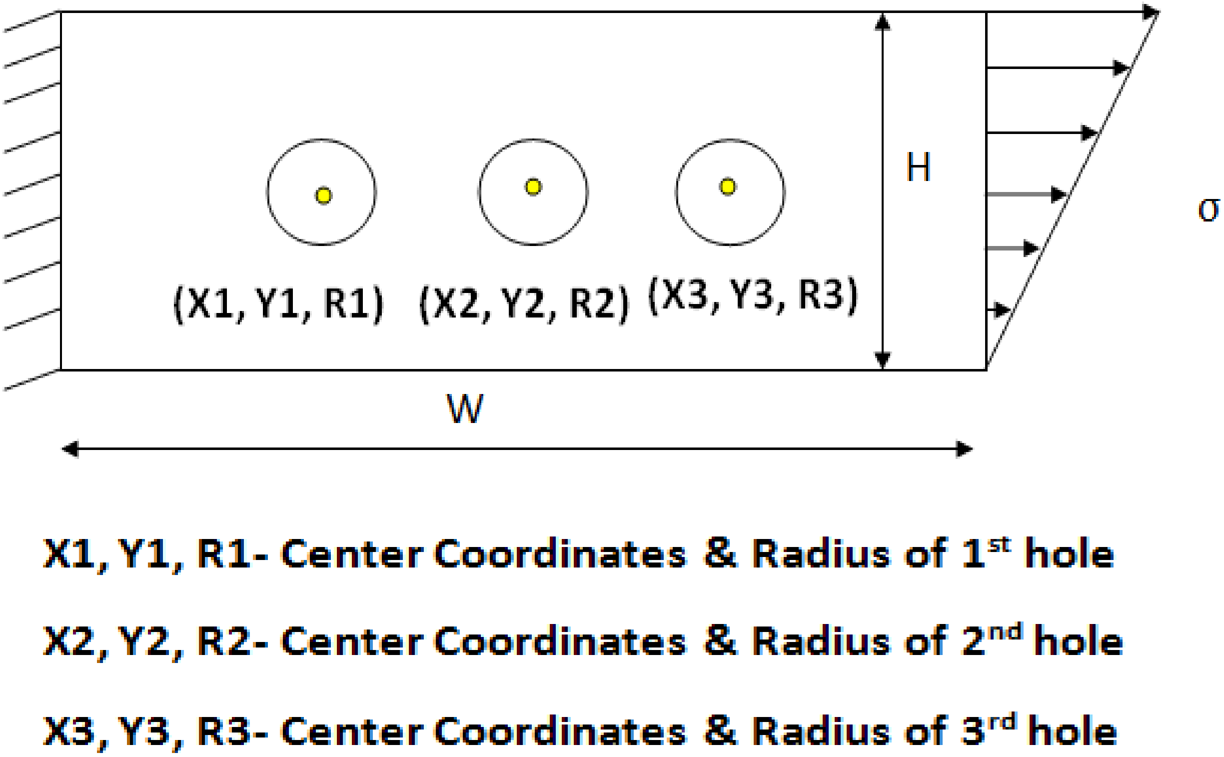
(2) Geometry Details

The geometry details of the plate with three holes subjected to Tensile Load considered for optimization is given below

d= 10 mm, H= 50 mm, W= 100 mm, σ: 0 to 10 MPa

(3) Parameterized Model in ANSYS

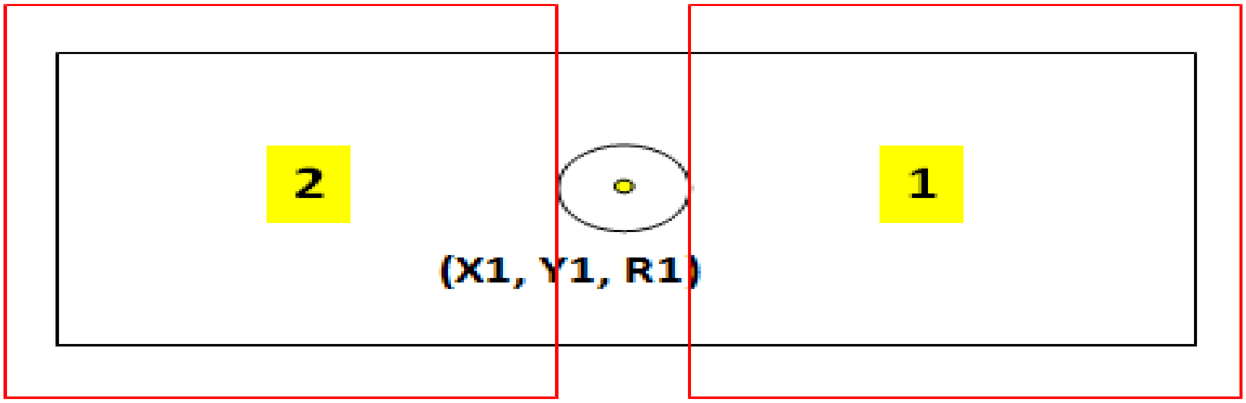
The above model is parameterized in ANSYS with the following Design Variables



(4) Optimization

(4.1) Part 1

For performing optimization, 1st hole is fixed at origin (0, 0) and its radius 'R1' is taken as 5 mm. Co-ordinates (X2, Y2) of 2nd hole is varied in negative x-axis (Section 2) with the limits, X2: -39 to -11, Y2: -15 to 15. Co-ordinates (X3, Y3) of 3rd hole are varied in positive x-axis (Section 3) with the limits, X3: 11 to 39, Y3: -15 to 15.It is further pictorially represented below



(4.1.1) Four Design Variables

For this case, radius of 2nd & 3rd holes are taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of 2nd hole,
X3, Y3- Centre Co-ordinates of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[-14 0 14 0]	[-39 -15 11 -15]	[-11 15 39 15]
2	[-12 0 12 0]	[-13 -15 11 -15]	[-11 15 13 15]
3	[-11.3 0 11.3 0]	[-11.5 -1 11 -1]	[-11 1 11.5 1]

Table 22: LVL1_3H_4D_Boundary Limits

SETS	X2	Y2	X3	Y3	SMAX
1	-14.0595	-0.664	13.9112	-0.7077	14.3501
2	-12.0099	-0.4112	11.9024	-0.3774	14.3267
3	-11.4612	-0.5263	11.3685	-0.5618	14.2379

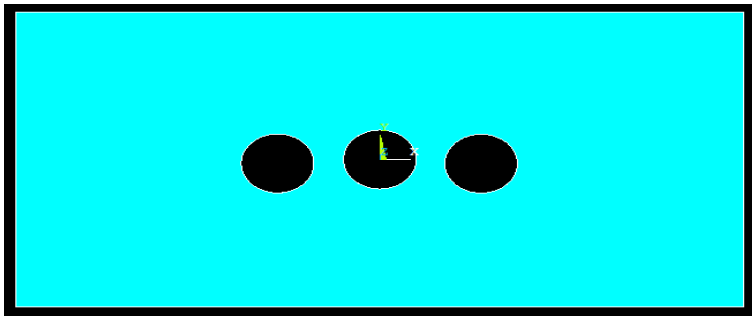
Table 23: LVL1_3H_4D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd & 3rd holes gets closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure

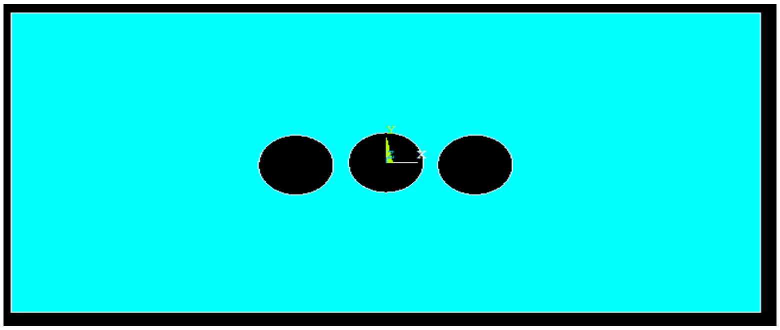
(ii) Plots

The below plots are generated based on values of (X2, Y2) & (X3, Y3) from table 23

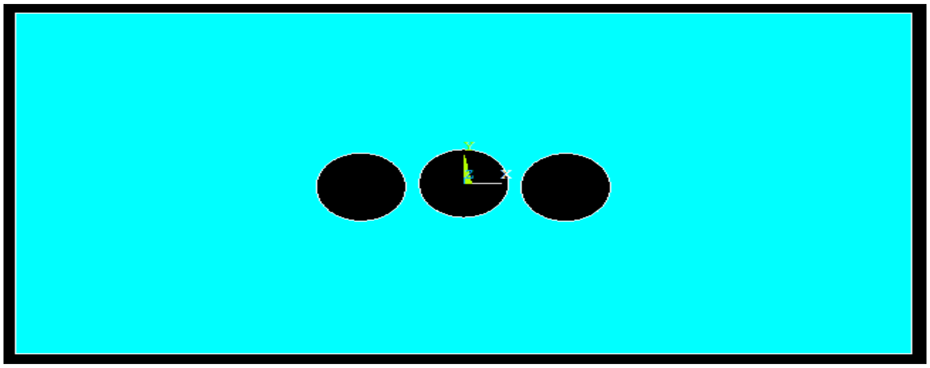
1) SET 1



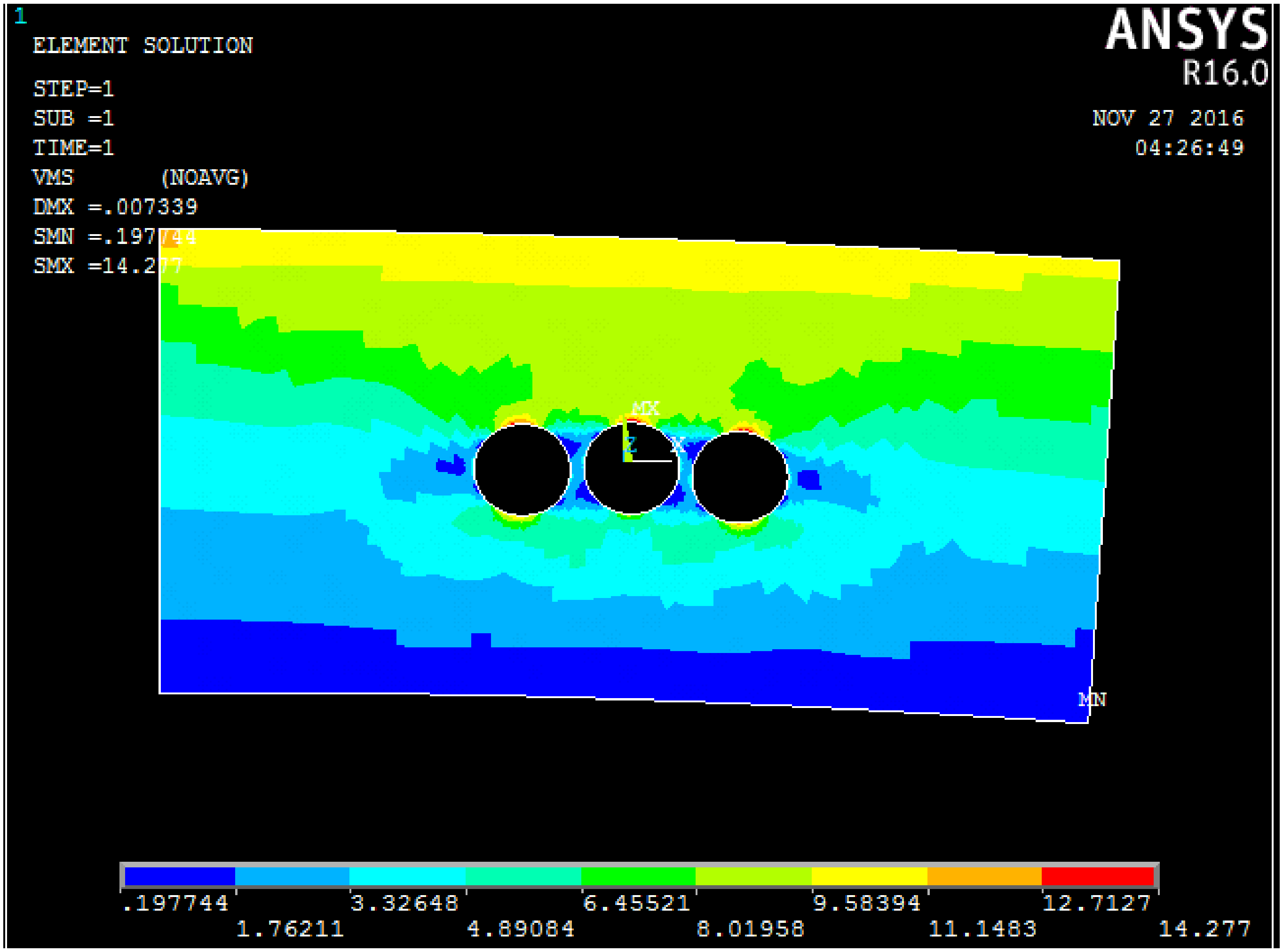
2) SET 2



3) SET 3



(iii) Contour plot of maximum stress concentration:



(4.1.2) Six Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

X3, Y3, R3- Centre Co-ordinates & Radius of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[-14 0 4 14 0 4]	[-39 -15 3 11 -15 3]	[-11 15 5 39 15 5]
2	[-12 0 4.5 14 0 4.5]	[-14 -1 4 11 -1 4]	[-11 1 5 14 1 5]
3	[-11.5 0 4.8 11.5 0 4.8]	[-12 -0.5 4.5 11 -0.5 4.5]	[-11 0.5 5 12 0.5 5]

Table 24: LVL1_3H_6D_Boundary Limits

SETS	X2	Y2	R2	X3	Y3	R3	SMAX
1	-14.5311	0.2236	3.657	14.2264	-0.3815	4.2464	14.7231
2	-12.2646	0.0786	4.3229	14	-0.3117	4.3161	14.3213
3	-11	-0.0172	4.5	11	-0.0122	4.5901	14.0088

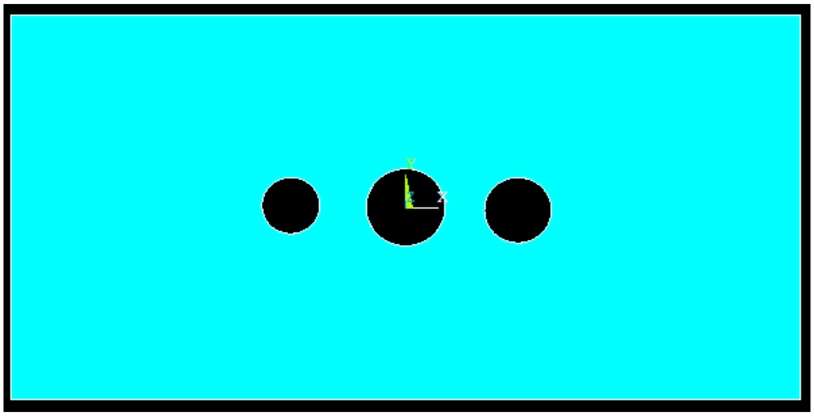
Table 25: LVL1_3H_6D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd & 3rd holes gets closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure. Also, when holes are of different size, when they get closer, they try to attain same size for better stress relieving.

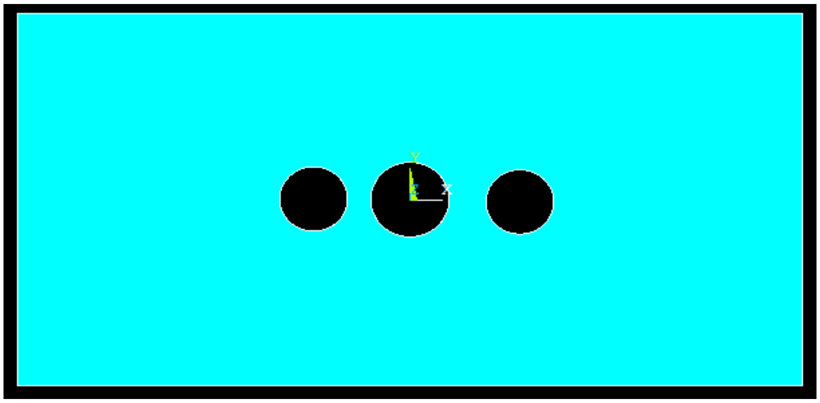
(ii) Plots

The below plots are generated based on values of (X2, Y2, R2) & (X3, Y3, R3) from table 25

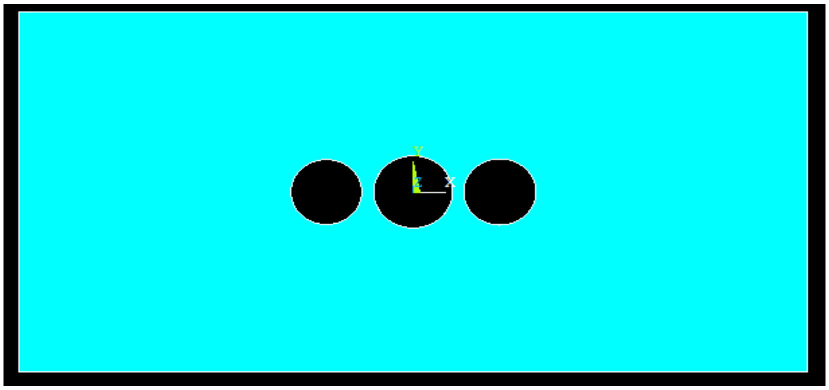
1) SET 1



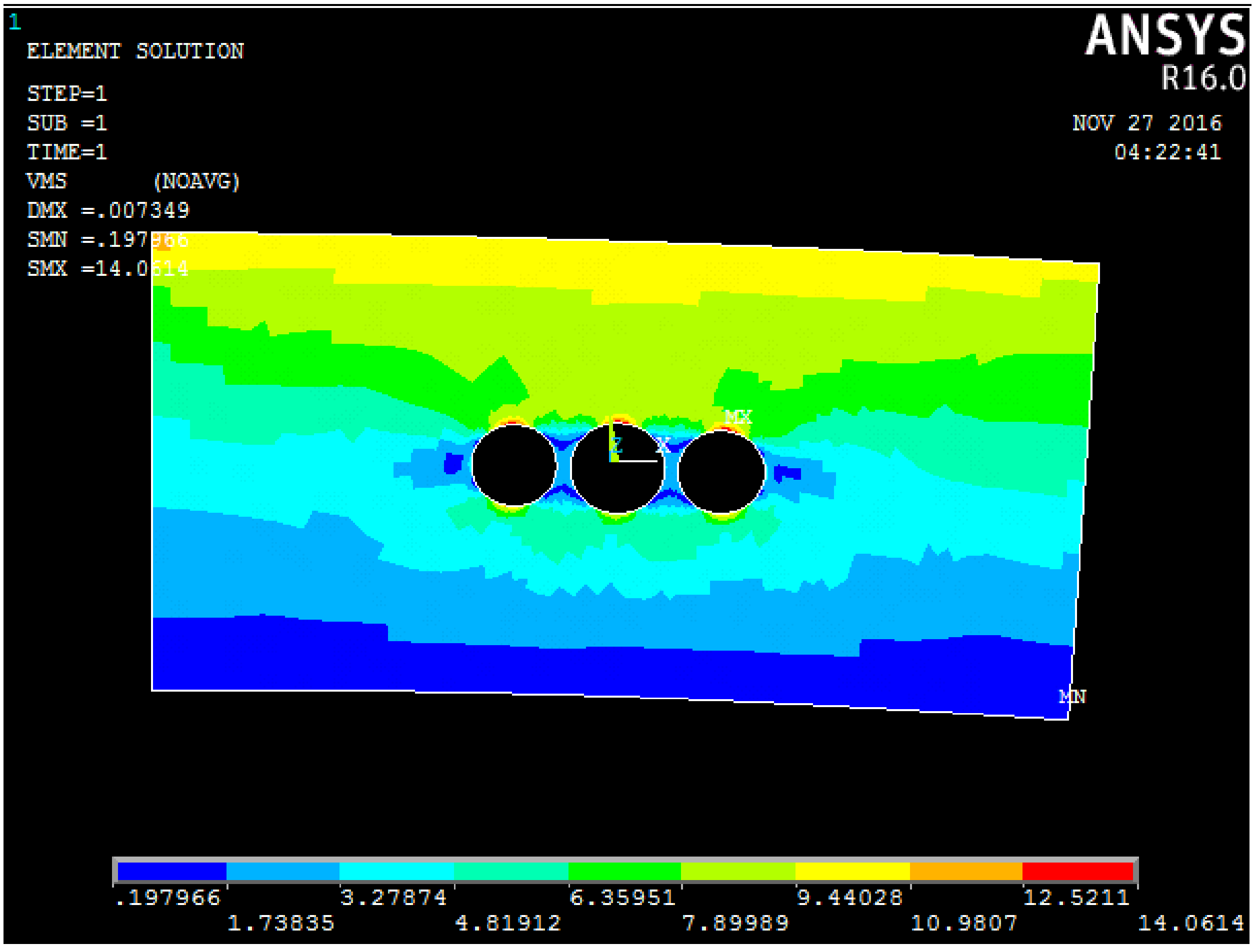
2) SET 2



3) SET 3

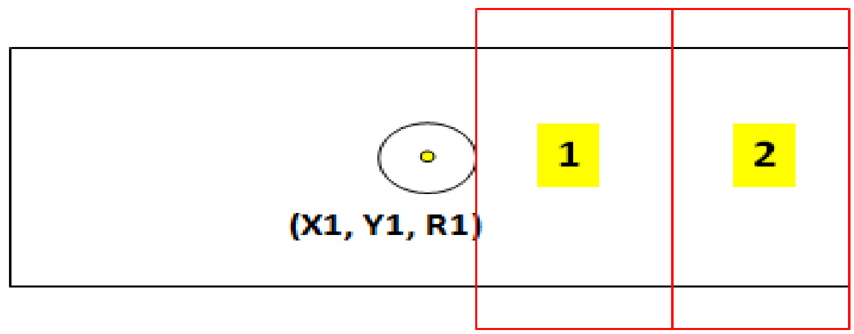


(iii) Contour plot of maximum stress concentration



(4.2) Part 2

For performing optimization, 1st hole is fixed at origin (0, 0) and its radius 'R1' is taken as 5 mm. Co-ordinates of both the holes are varied in positive x-axis. X2: 11 to 20, Y2: -15 to 15 (Section 1), X3: 31 to 40, Y2: -15 to 15 (Section 2). It is further pictorially represented below



(4.2.1) Four Design Variables

For this case, radius of 2nd & 3rd holes are taken as 5 mm

Design Variables: X2, Y2- Centre Co-ordinates of 2nd hole,

X3, Y3- Centre Co-ordinates of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[15 0 35 0]	[11 -15 31 -15]	[20 15 40 15]
2	[12 0 32 0]	[11 -15 31 -15]	[20 15 40 15]
3	[19 0 39 0]	[11 -15 31 -15]	[20 15 40 15]
4	[19 0 32 0]	[11 -15 31 -15]	[20 15 40 15]

Table 26: LVL2_3H_4D_Boundary Limits

SETS	X2	Y2	X3	Y3	SMAX
1	13.2896	0.1085	31.8575	-2.565	15.1403
2	13.0832	0.0129	31.7031	-2.5347	15.1504
3	18.7328	-0.1407	35.8249	-1.4833	15.5798
4	18.6636	-0.1786	31.5715	-1.37	15.2559

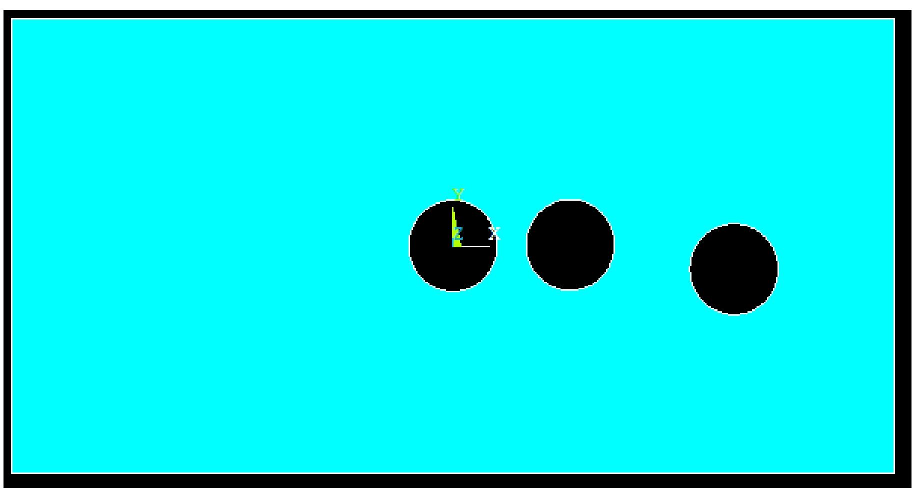
Table 27: LVL2_3H_4D_Optres

From the above optimized values, it is evident that the stress is getting reduced whenever two of the three holes are getting closer. This is because stress relieving is happening when the holes are close to each other. Hence, in case of two holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure.

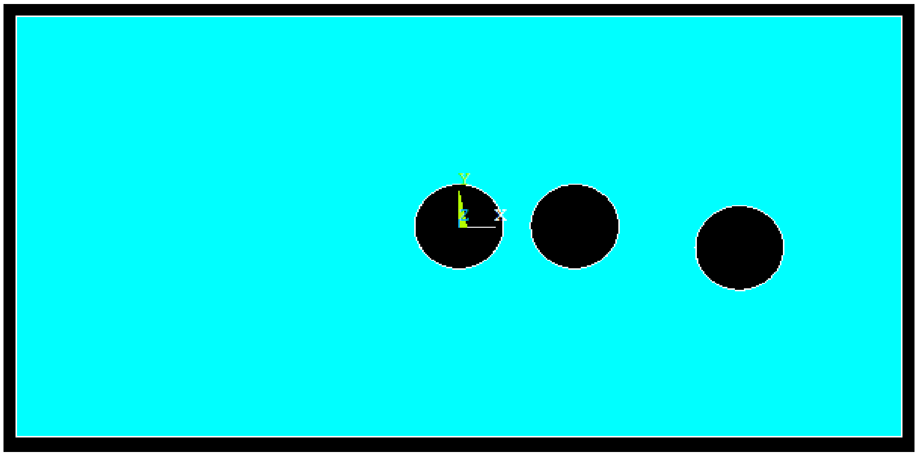
(ii) Plots

The below plots are generated based on values of (X2, Y2) & (X3, Y3) from table 27

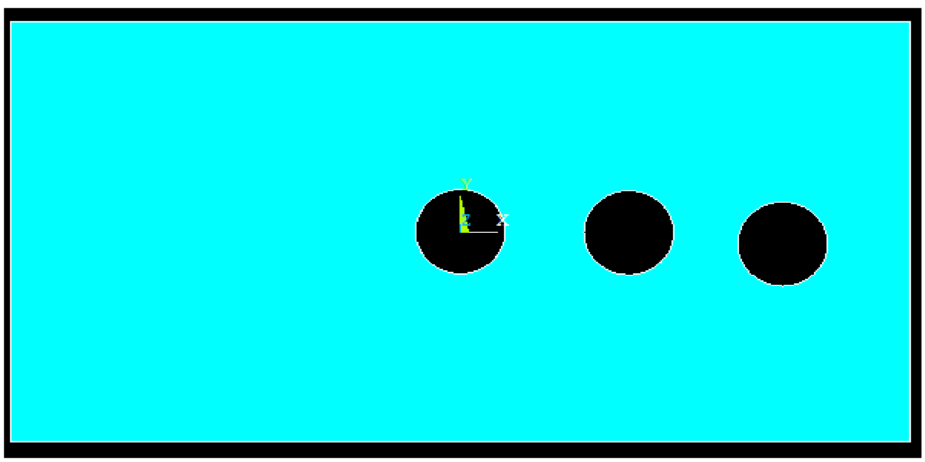
1) SET 1



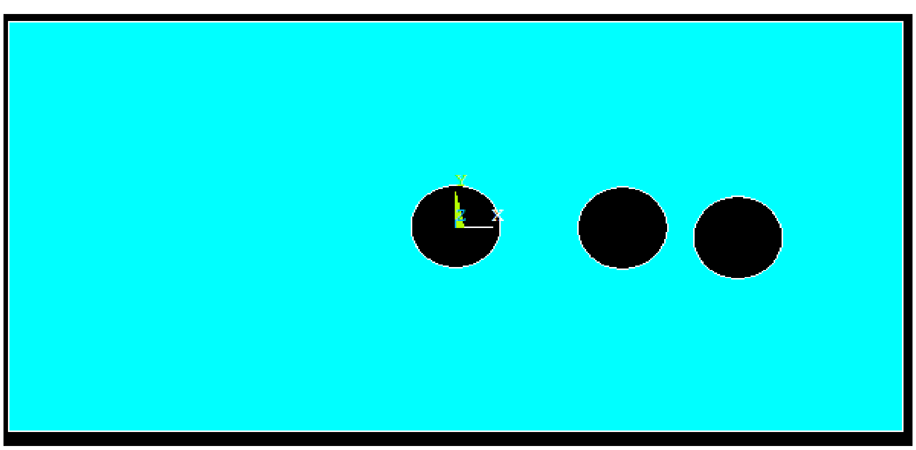
2) SET 2



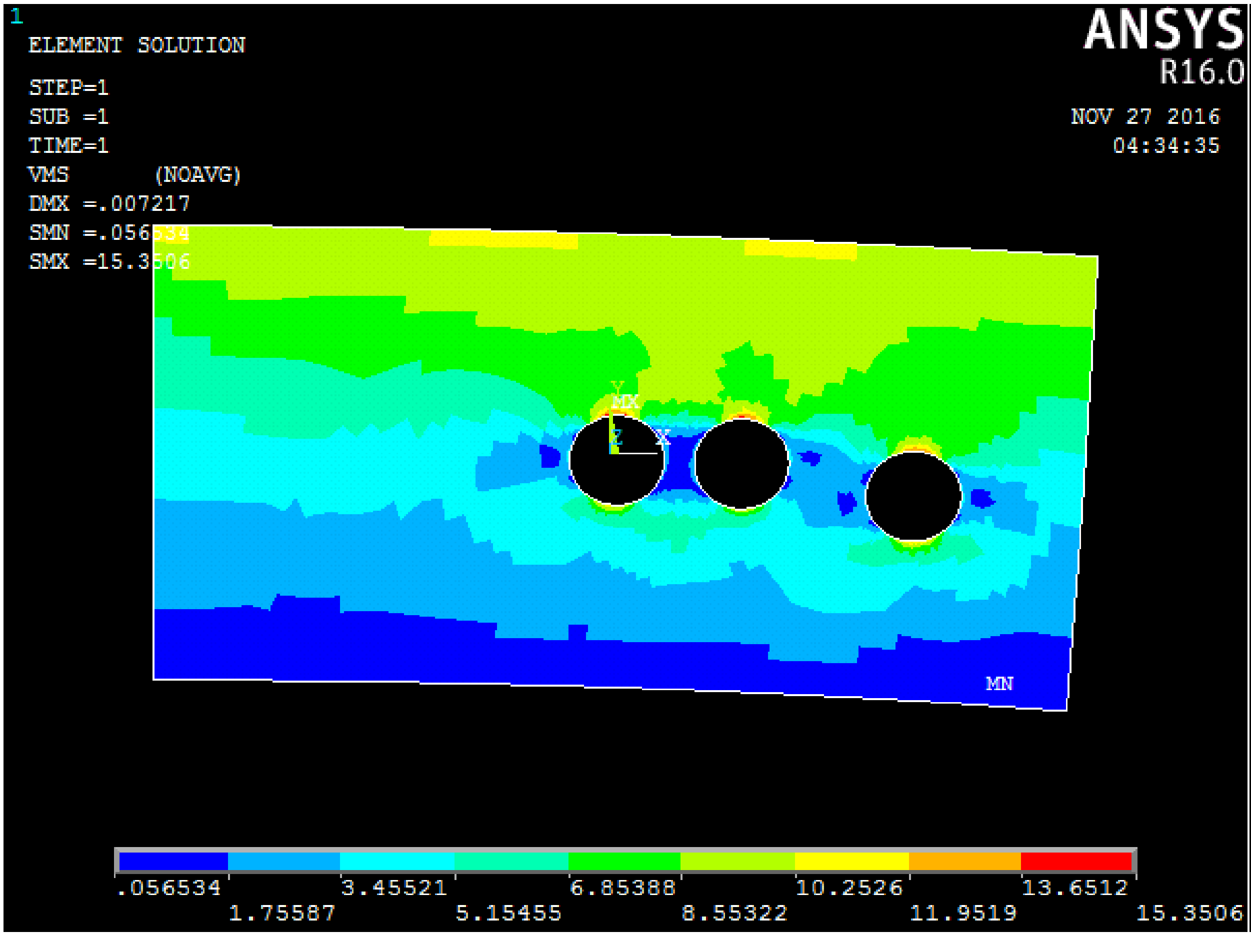
3) SET 3



4) SET 4



(iii) Contour plot of maximum stress concentration



(4.2.2) Six Design Variables

Design Variables: X2, Y2, R2- Centre Co-ordinates & Radius of 2nd hole

X3, Y3, R3- Centre Co-ordinates & Radius of 3rd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[15 0 4 35 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
2	[12 0 4 31 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
3	[19 0 4 39 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]
4	[19 0 4 32 0 4]	[11 -15 3 31 -15 3]	[20 15 5 40 15 5]

Table 28: LVL2_3H_6D_Boundary Limits

SETS	X2	Y2	R2	X3	Y3	R3	SMAX
1	14.2978	0.0262	5	34.4949	-0.4375	3.0056	15.0489
2	11.4721	0.0072	4.9999	31.0411	-0.2313	3.0013	14.9108
3	19.1959	0.0467	4.9999	38.8118	-0.347	3.0041	15.443
4	19.3642	-1.0722	4.6779	31.5548	-0.3996	3.1013	15.8199

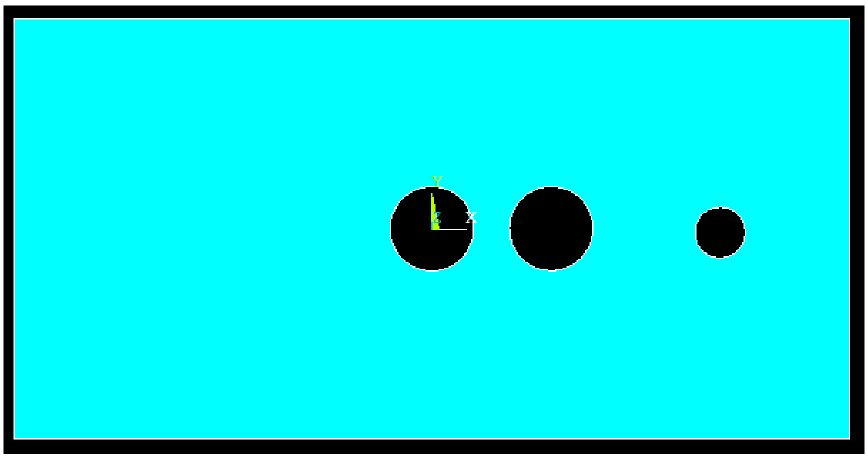
Table 29: LVL2_3H_6D_Optres

From the above optimized values, it is evident that the stress is getting reduced whenever two of the three holes are getting closer. This is because stress relieving is happening when the holes are close to each other. Hence, in case of three holes, centre to centre distance should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure. Also, when holes are of different size, when they get closer, they try to attain same size for better stress relieving.

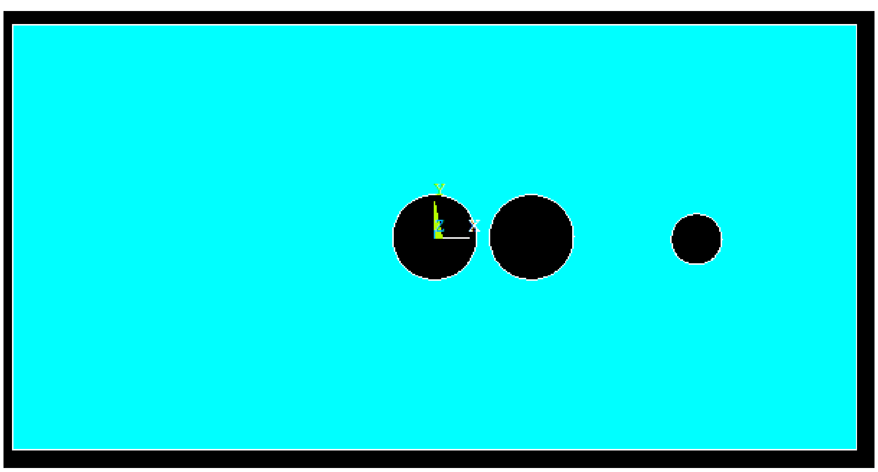
(ii) Plots

The below plots are generated based on values of (X2, Y2, R2) & (X3, Y3, R3) from table 29

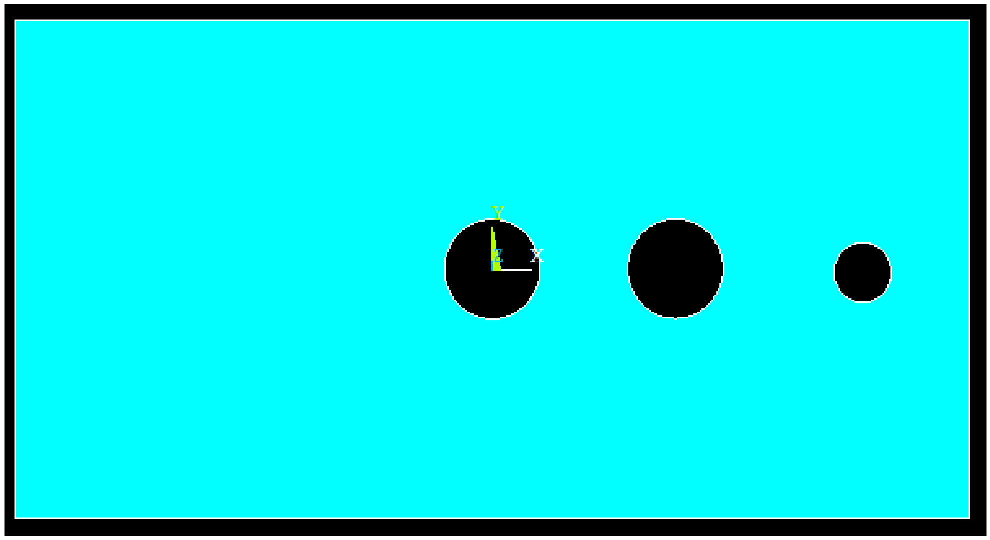
1) SET 1



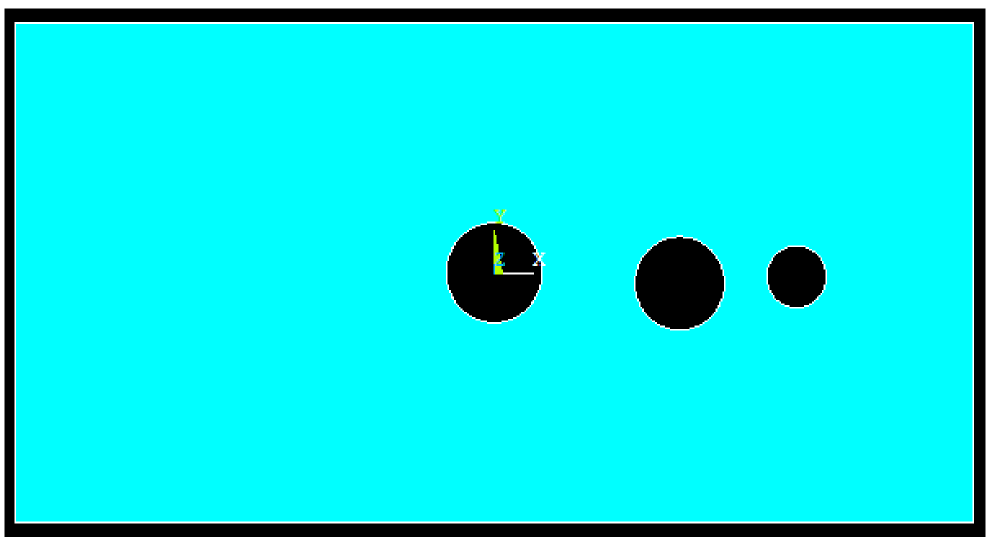
2) SET 2



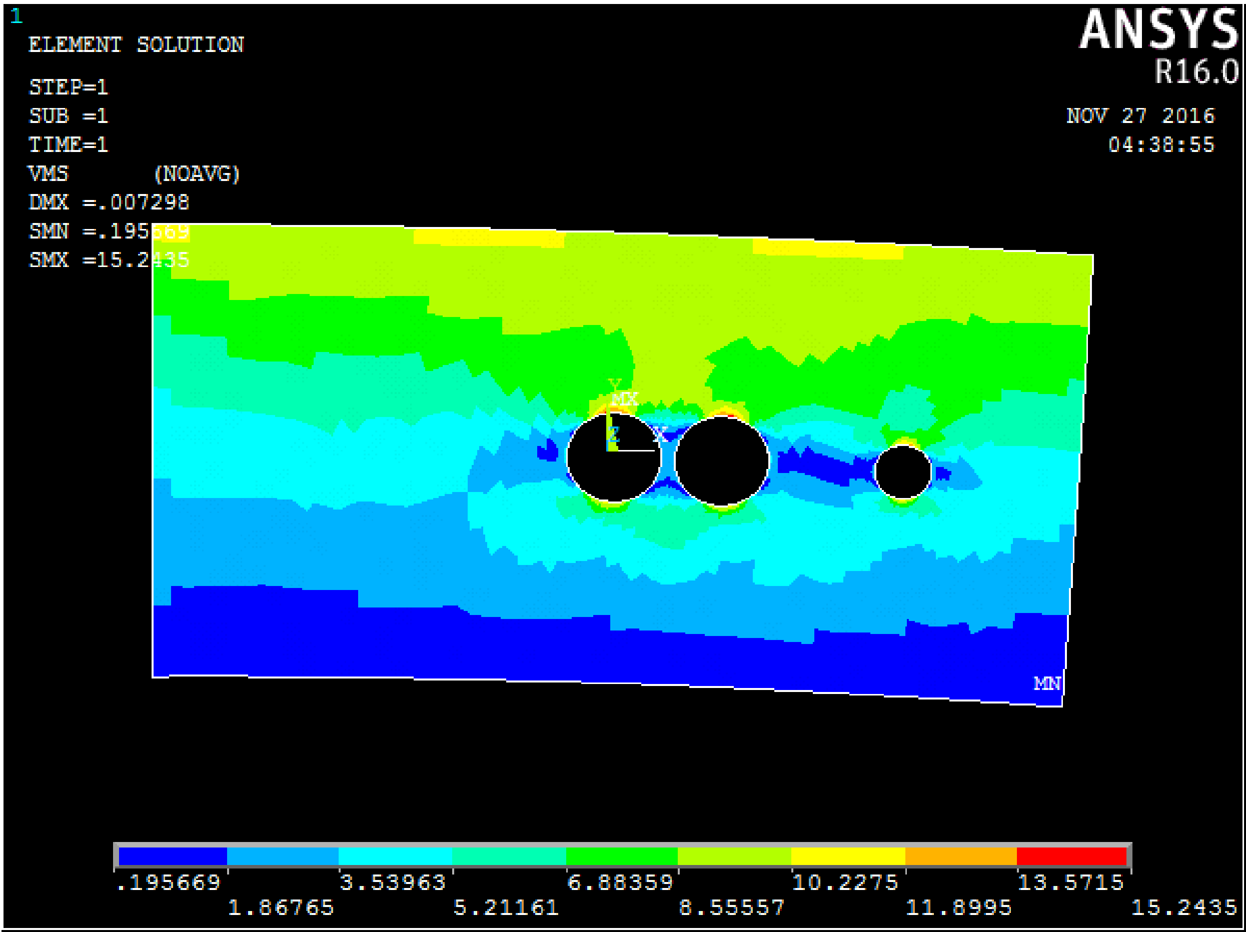
3) SET 3



4) SET 4



(iii) Contour plot of maximum stress concentration

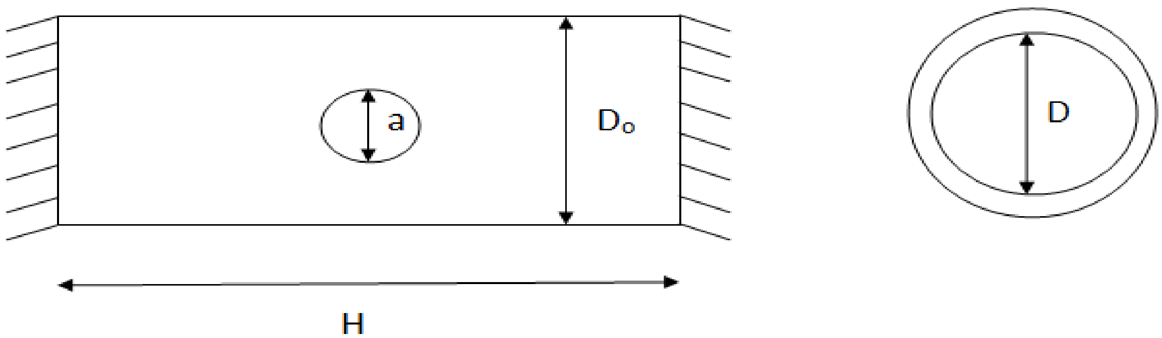


4.3 A CYLINDER WITH AN INTERNAL PRESSURE

4.3.1 Presence of a Single Hole

(1) Actual Model

The pictorial representation of a cylinder with hole subjected to an Internal Pressure (IP) is given below



H- Height of the Cylinder, R_o- Outer Radius, a- Diameter of the hole

R- Inner Radius, D_o- Outer Diameter, D- Inner Diameter, h- thickness of the shell

Figure 28: CIP_CWSH

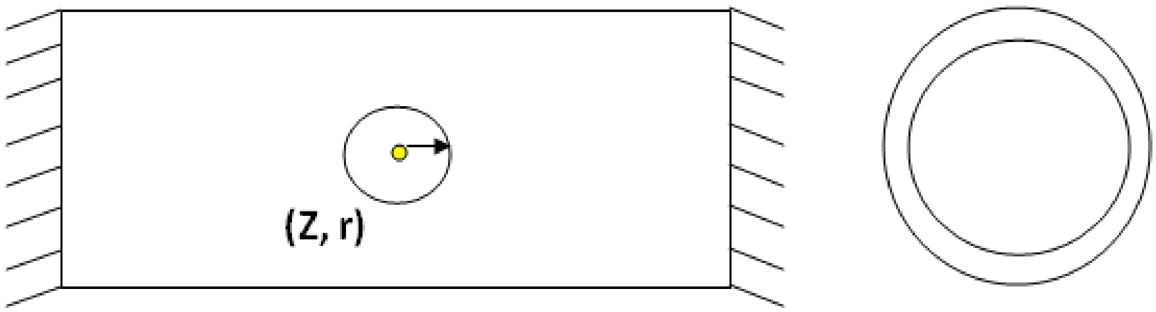
(2) Geometry Details

The geometry details of the cylinder with a hole subjected to Internal Pressure considered for optimization is given below

H= 8 m, R= 3.8 m, R_0= 4 m, D=2*R, D_o=2*R_o, h=R_o-R_i, P- Internal Pressure applied (5 MPa), a=3 m, r=a/2=1.5 m.

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables



Z- Location of the hole along the height of the cylinder

r- Radius of the hole in cylinder

(4) Contour plot of Maximum Von-Mises Stresses (S_{max})

The Contour plot of S_{max} is generated by varying 'Z' and 'r' with the following ranges

Z: 2 to 6 m, r: 0.5 to 1.5 m to understand stress distribution due to internal pressure. Plots has been generated for 'Full Model' of the cylinder and for 'symmetric half model' of it as well

(i) Full Model

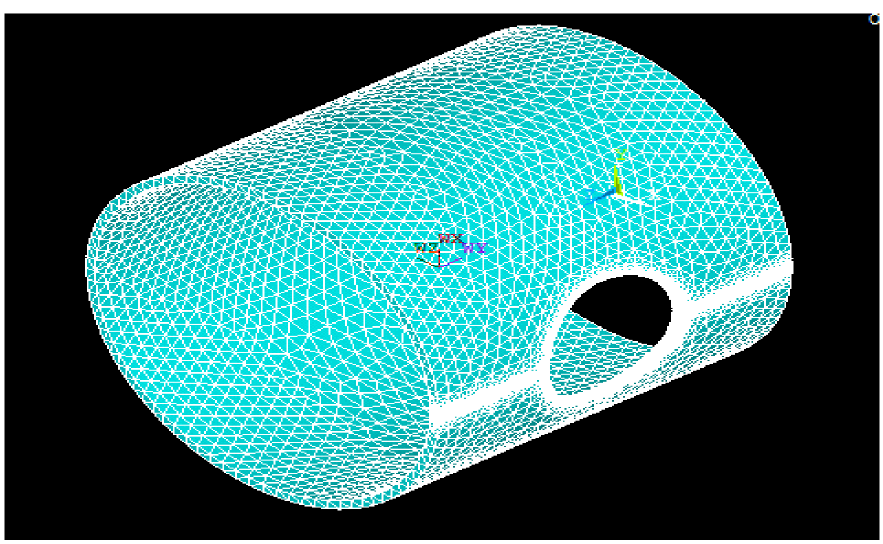


Figure 29: Full_Model_Cylinder

(ii) Maximum Von-mises stresses (S_{max})

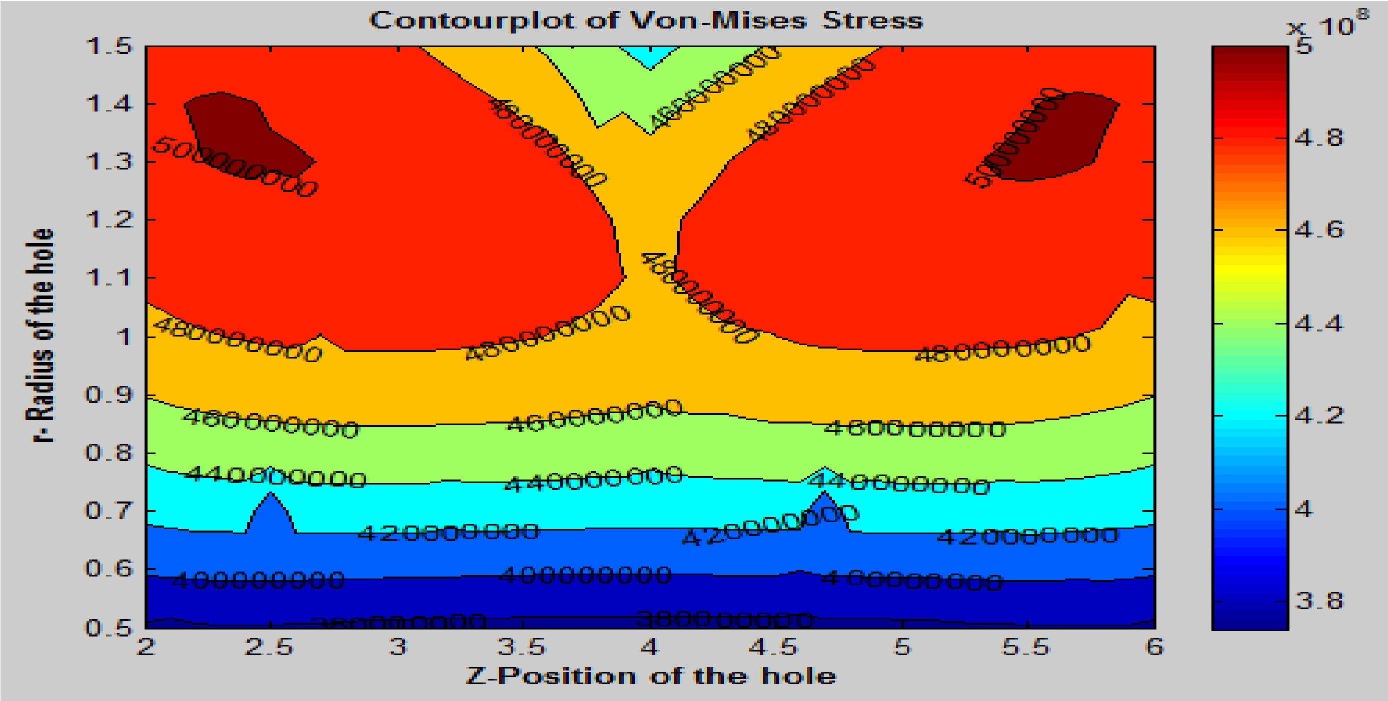


Figure 30: CIPF_CPSH

Cntrplot	1	37229.279 s	37223.521 s

Hours: 37223.521/3600= 10.4 hours

The time consumed in ANSYS to generate the above plot is almost around 10 hours

(iii) Half Model

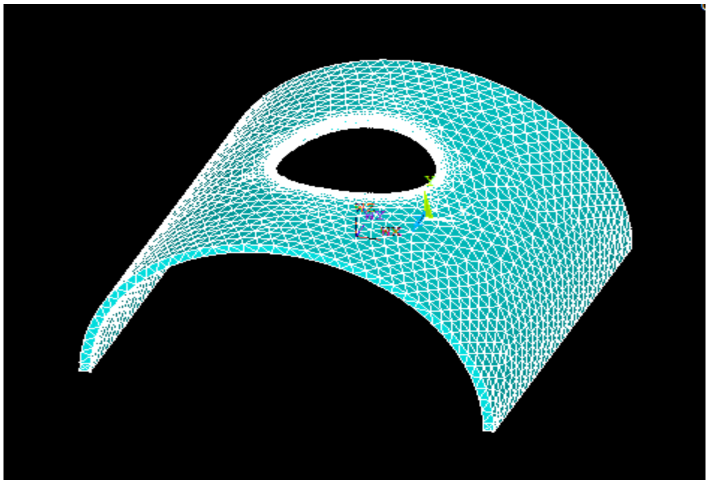


Figure 31: Half_Model_Cylinder

(iv) Maximum Von-mises stresses (S_{max})

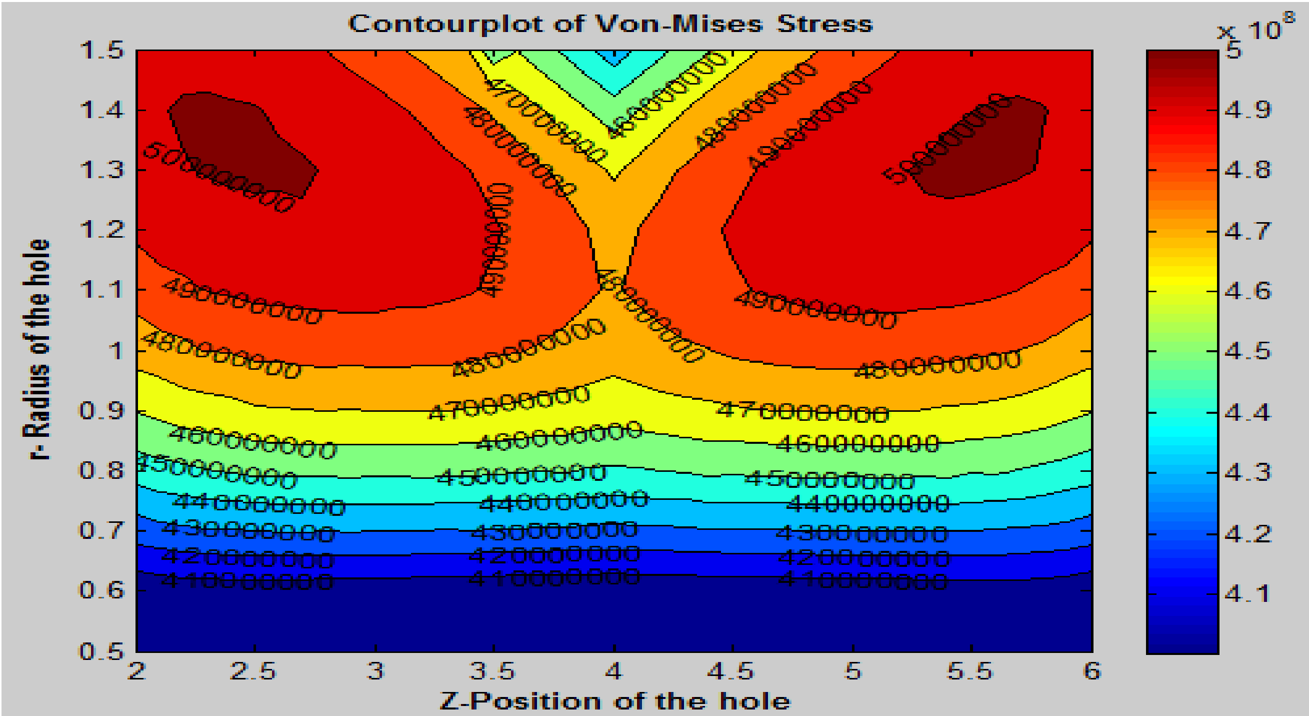


Figure 32: CIPH_CPSH

Cntrplot	1	16950.704 s	16944.665 s

Hours: 16944.665/3600= 4.7 hours

The time consumed in ANSYS to generate the above plot is almost around 5 hours

From Figure 30 & 32, it is clearly seen that both plots look similar. It is best to use symmetric half model to generate stress contour plot for a cylinder with a hole subjected to internal pressure. It saves plenty of time, can be used subsequently for optimization too.

(5) Optimization

The range considered for optimization are, Z: 2 to 6 m & r: 0.5 to 1.5 m

Objective:

1. The main objective was to find the location where the stress acting is less for a cylinder with hole of radius 1.5 m. Unlike plate, the location design Variable is ‘’Z’’ (just one variable) in case of cylinder which is trivial and gives error in FMINCON. So revising the objective function as a weight minimization problem with stress constraint finds the best location.

Design variables: Z, r- Location and radius of the hole

Constraint:

1. Side Constraints- Upper & Lower bounds of the design variable
2. Maximum Von-mises stress (S_{max}) should be less than the allowable stress (S_{all}= 440 MPa)

(5.1) Full Model

(i) FMINCON results

SETS	X	XL	XU
1	[3 1]	[2 0.5]	[6 1.5]
2	[3.5 1.4]	[2 0.5]	[6 1.5]
3	[5 1.4]	[2 0.5]	[6 1.5]

Table 30: CIPF_1H_2D_Boundary Limits

SETS	Z	r	Smax	Volume
1	4.0989	1.5	438464360	37.7656
2	3.9413	1.5	435580470	37.7656
3	4.1162	1.5	439631150	37.7656

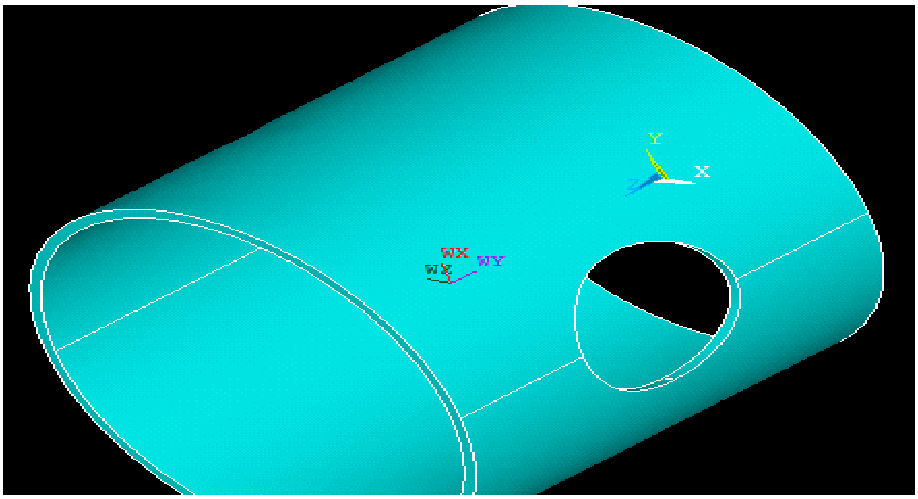
Table 31: CIPF_1H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as hole approaches mid height of the cylinder Hence the best location for the cylinder with a hole of radius 1.5 m is at Z=4 m . From figure 30, it is clear that stress values are less near the middle height of the cylinder

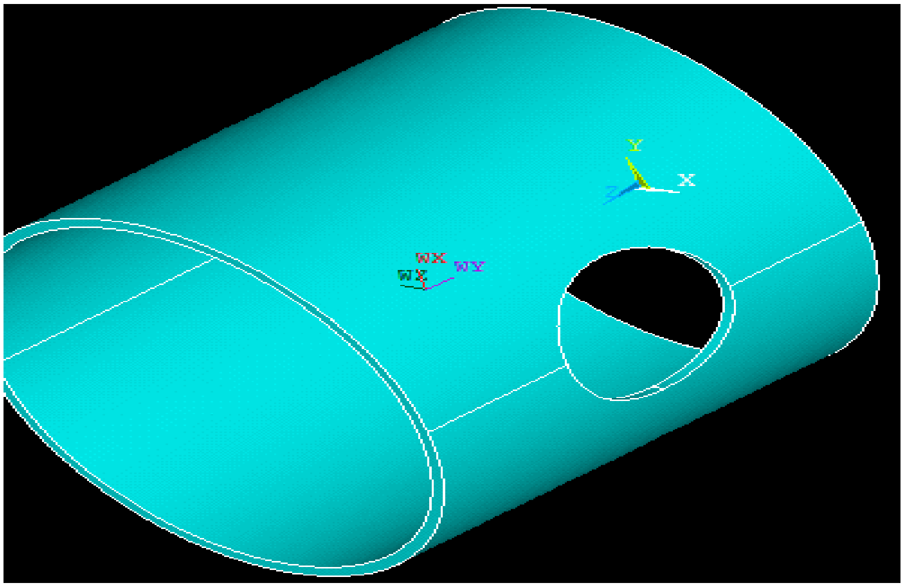
(ii) Plots

The below plots are generated based on values of (Z, r) from table 31

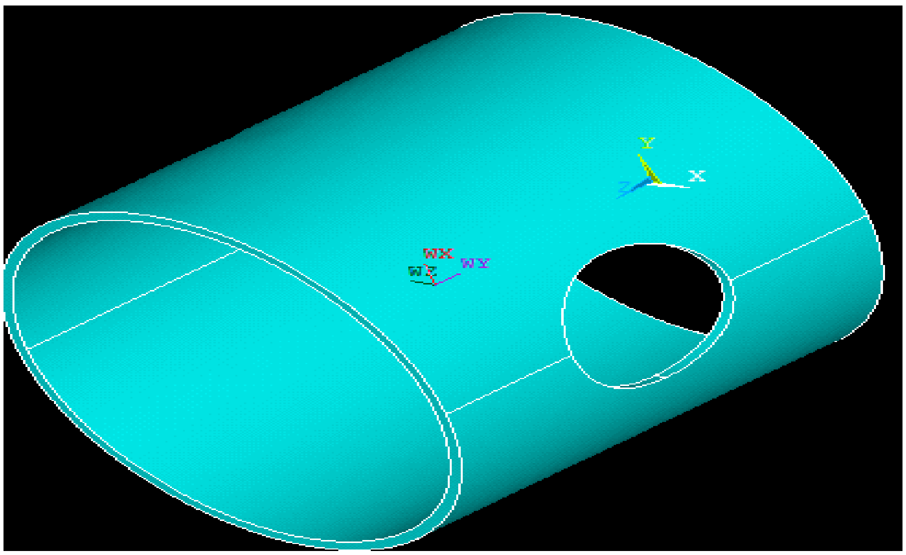
1) SET 1



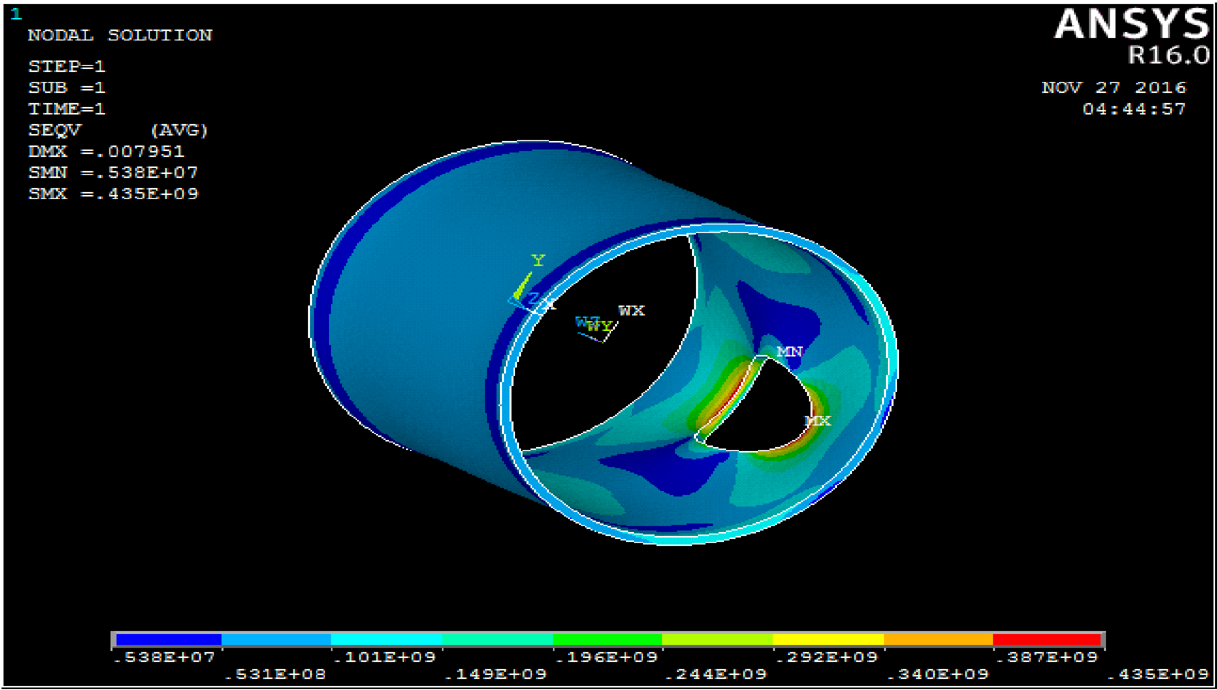
2) SET 2



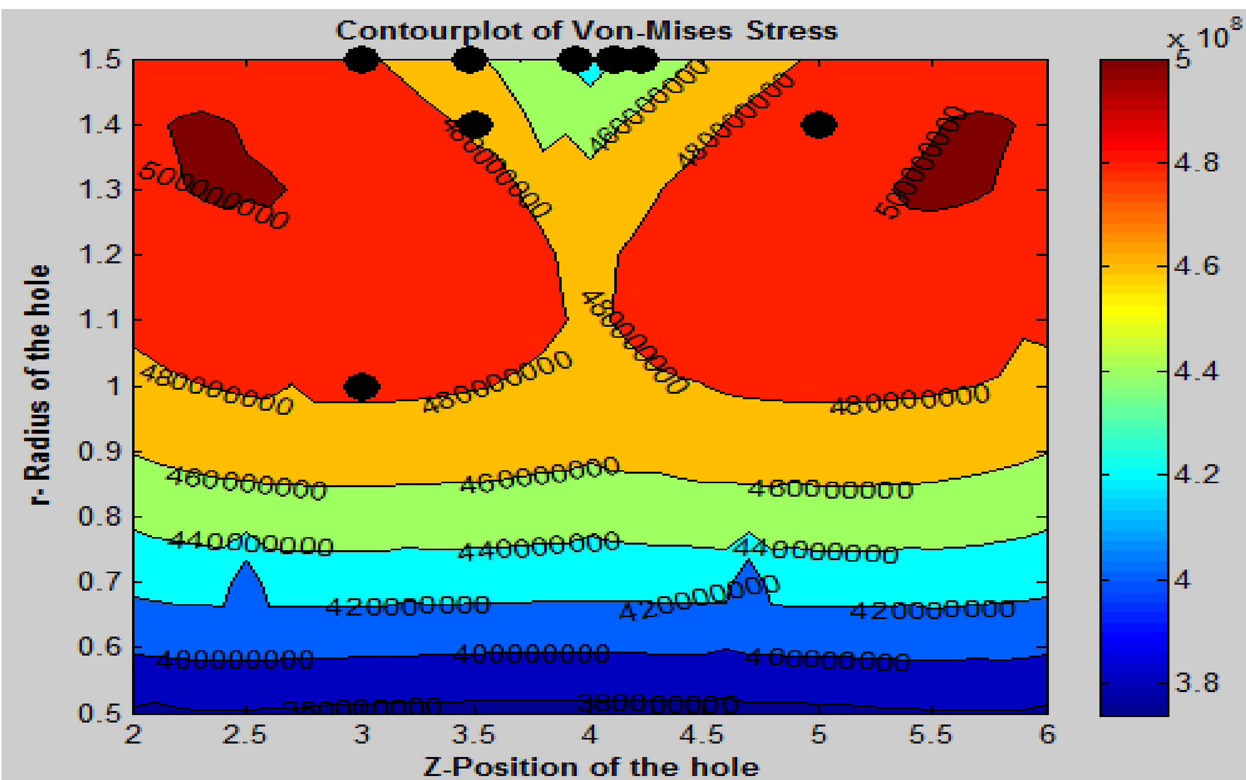
3) SET 3



(ii) Contour plot of maximum stress concentration



(ii) Stress Plot with Design Points



(5.2) Half Model

(i) FMINCON results

SETS	X	XL	XU
1	[3 1]	[2 0.5]	[6 1.5]
2	[3.5 1.3]	[2 0.5]	[6 1.5]

Table 32: CIPH_1H_2D_Boundary Limits

SETS	Z	r	Smax	Volume
1	4.0344	1.5	437191210	18.1621
2	3.9601	1.5	437592850	18.1621

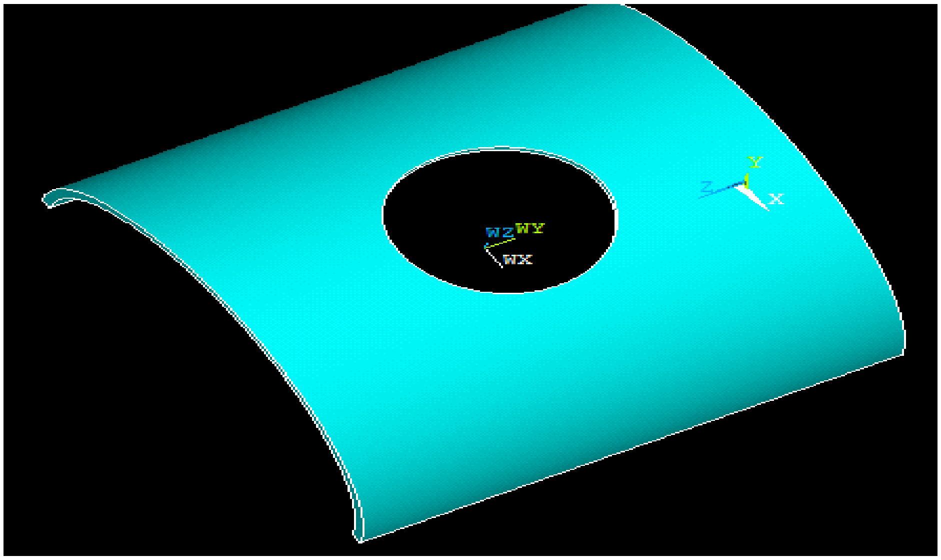
Table 33: CIPH_1H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as hole approaches mid height of the cylinder Hence the best location for the cylinder with a hole of radius 1.5 m is at Z=4 m . From figure 32, it is clear that stress values are less near the middle height of the cylinder

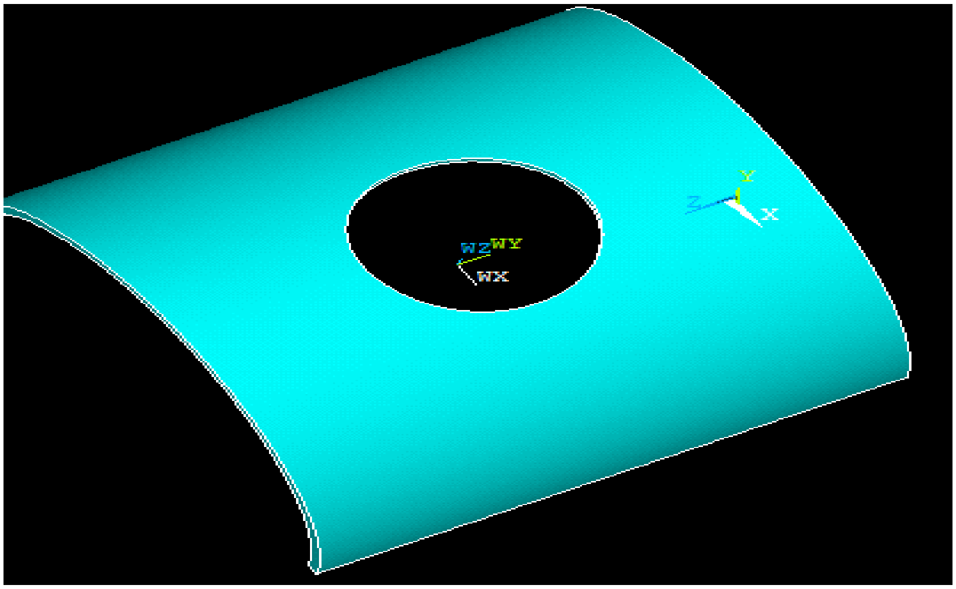
(ii) Plots

The below plots are generated based on values of (Z, r) from table 33

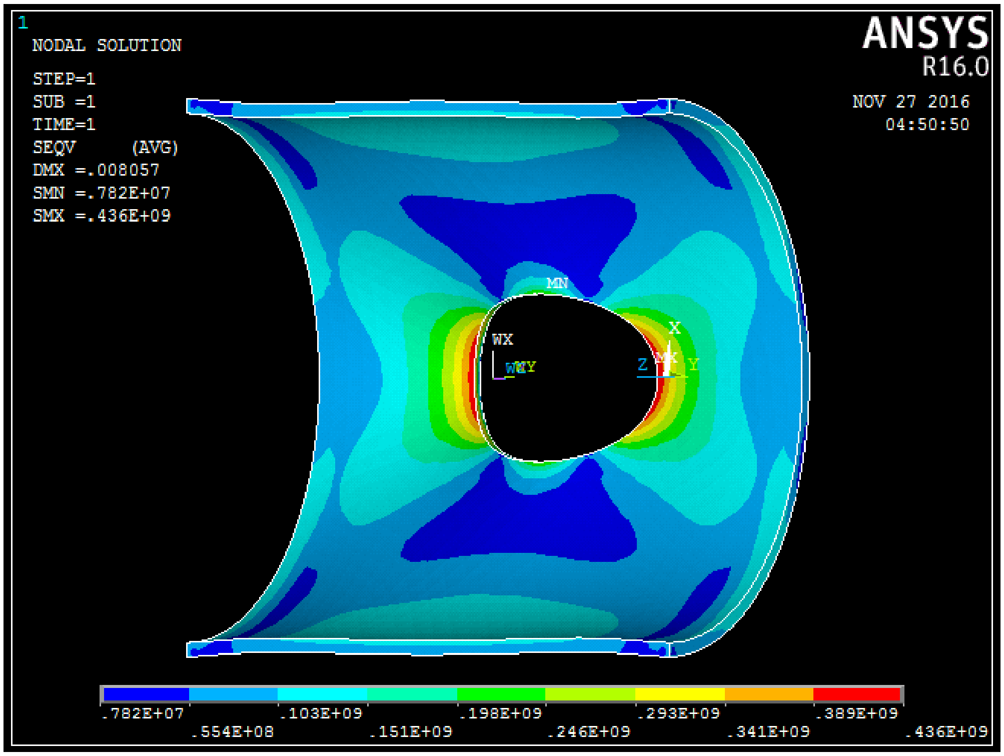
1) SET 1



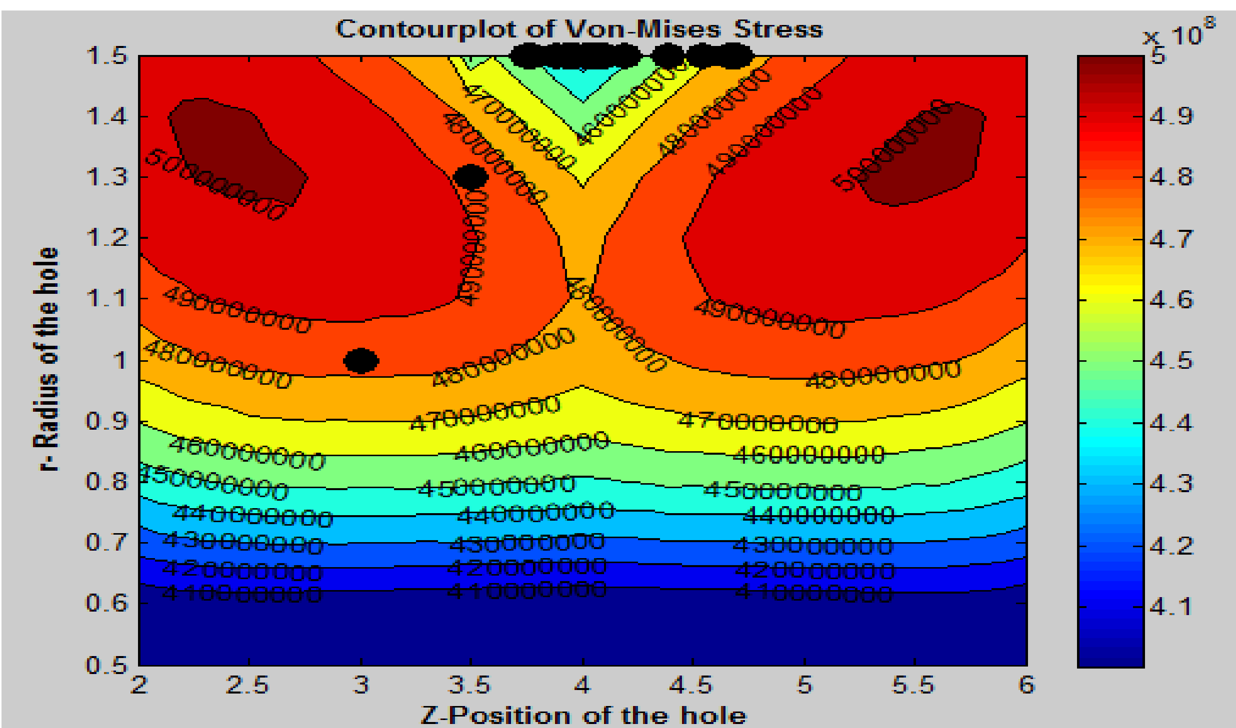
2) SET 2



(ii) Contour plot of maximum stress concentration



(ii) Stress Plot with Design Points



(6) Maximum Von-mises Stress Vs Radius of the Hole

For generating below plot, Hole is fixed at Z= 4m, only radius ‘r’ of the hole is varied to generate the below plot

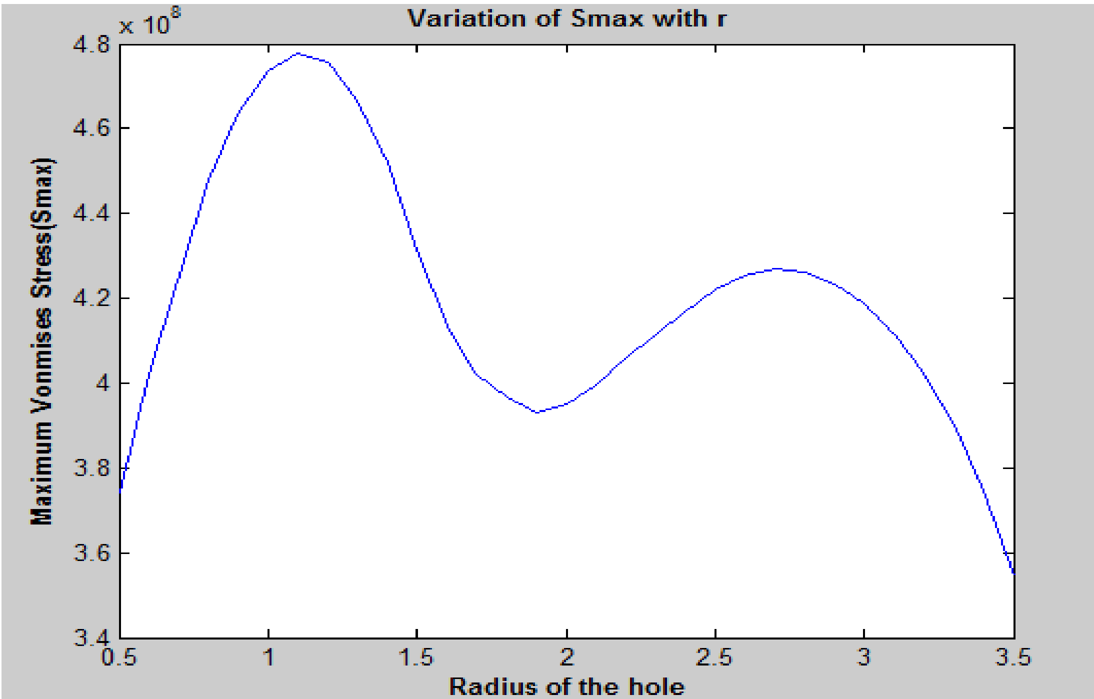


Figure 33: CIP_Smax_r

From figure 33, it is clear that stress distribution around the circumference varies non-linearly with respect to radius of the hole

(7) Analytical Solution

For a Cylinder with hole at its mid-height, an approximate analytical solution is derived by *Vandyke (1965)*

$\boldsymbol{\beta = \left(\sqrt[3]{\frac{(1-\vartheta)^2}{2}}\right) * \left(\frac{a}{\sqrt{R*h}}\right)}$ **,** Poisson’s ratio (*v*) =0.3

Now above equation becomes,

$$\boldsymbol{\beta = 0.639 * \left(\frac{a}{\sqrt{R*h}}\right)}$$

(i) Stress Concentration Factor

$$K_{tn}(\beta) = 2.5899 + (0.8022 * \beta) + (4.0112 * \beta^2) - (1.8235 * \beta^3) + (0.3751 * \beta^4)$$

(ii) Maximum Stress

$$\sigma_n = \frac{\sigma}{(1-\frac{d}{H})} \quad , \quad \sigma_{max} = \sigma_n * K_{tn}$$

σ_{max} – Maximum Stress developed due to discontinuity in Structure

σ – Applied Stress, p- Internal Pressure, h- thickness of the shell

R- Inner/ Mean radius of the cylindrical shell, a- radius of the hole

Substituting: **h=0.2 m, p=5 MPa**

we get: **σ= 95 MPa**

(iii) Notes

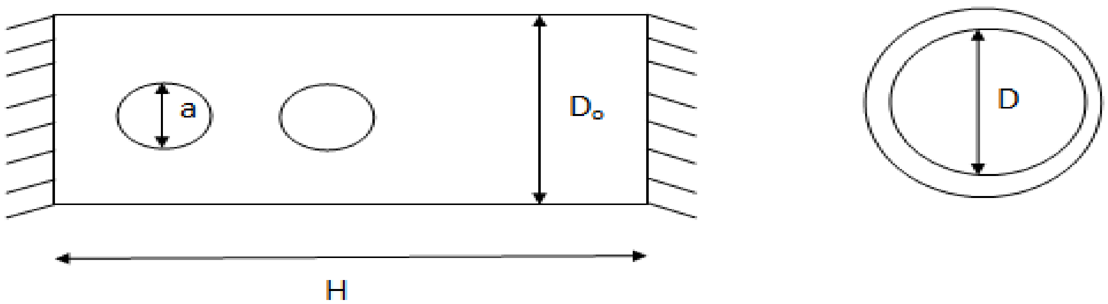
Radius(a)	β	K_{tn}	ANSYS	Analytical	%Error
0.5	0.37	3.35	379	318.25	16
1	0.733	4.723	473	449	5
1.5	1.099	6.44	435	612	29

1. ***K_{tn} Values are quite large for the larger values of β, compared to unity***
2. ***β= 0.5, K_{tn} Values are not usually large***
3. ***β: 0.7- 0.8, K_{tn} Values are appropriate***

4.3.2 Presence of two holes

(1) Actual Model

The pictorial representation of the cylinder with two holes subjected to Internal Pressure is given below



H- Height of the Cylinder, R_o- Outer Radius, a- Diameter of the hole

R- Inner Radius, D_o- Outer Diameter, D- Inner Diameter, h- thickness of the shell

Figure 34: CIP_CW2H

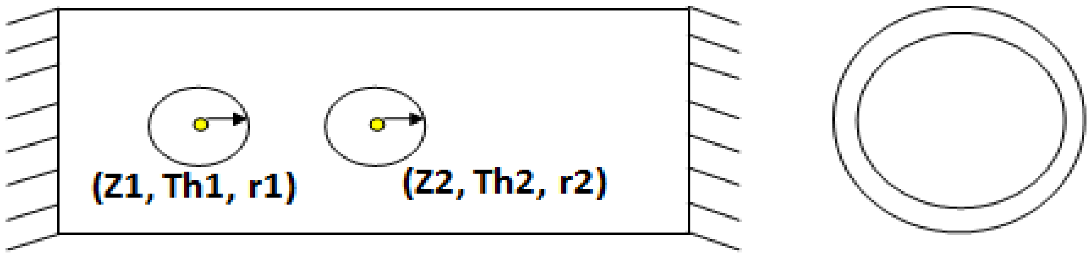
(2) Geometry Details

The geometry details of the cylinder with two holes subjected to Internal Pressure considered for optimization is given below

H= 10 m, R= 3.8 m, R_0= 4 m, D=2*R, D_o=2*R_o, h=R_o-R_i, P- Internal Pressure applied (5 MPa), a=2 m, r=a/2=1.0 m.

(3) Parameterized Model in ANSYS

The above model is parameterized in ANSYS with the following Design Variables



Z1- Location of 1st hole along height of the cylinder

Th1- Angular Location of 1st hole along circumference of the cylinder

r1- Radius of 1st hole

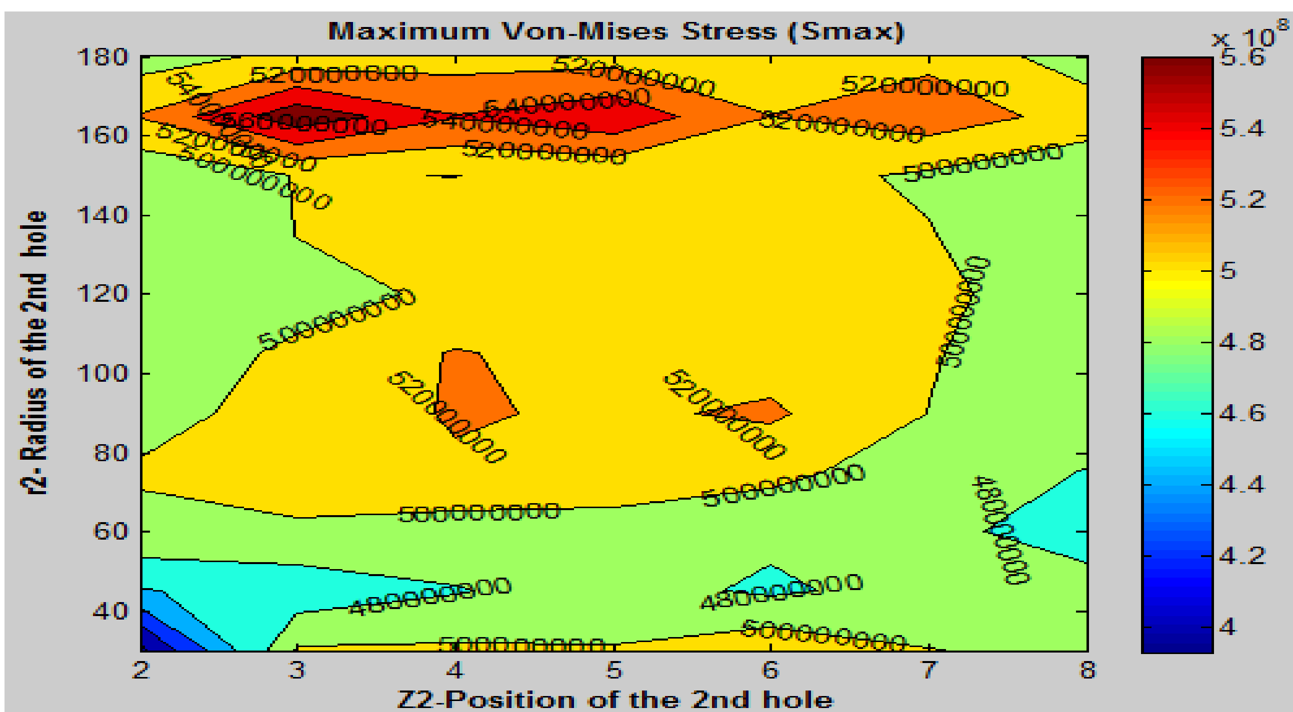
Z2- Location of 2nd hole along height of the cylinder

Th2- Angular Location of 2nd hole along circumference of the cylinder

r2- Radius of 2nd hole

(4) Contour Plot of Maximum Von-Mises stresses

For generating this plot, 1st hole is fixed at Z1=2 & Th1=0. Stress contour plot is generated by varying Z2 & Th2



(5) Optimization

For performing optimization, 1st hole is fixed at 'Z1=2' & 'Th1=0' and its radius 'r1' is taken as 1 m. Co-ordinates (Z2, Th2) is varied in the cylinder with the limits Z2: 2 to 8 m, Th2: 30 to 180 degree

(5.1) Two Design Variables

For this case, radius of 2nd hole 'r2' is taken as 1 m.

Design Variables: Z2, Th2- Location Variables of 2nd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints, Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[7 50]	[2 30]	[6 60]
2	[3 50]	[2 30]	[6 60]

Table 34: CIP_2H_2D_Boundary Limits

SETS	Z2	Th2	Smax
1	7.5097	59.1089	471939700
2	2	36.2156	411426080

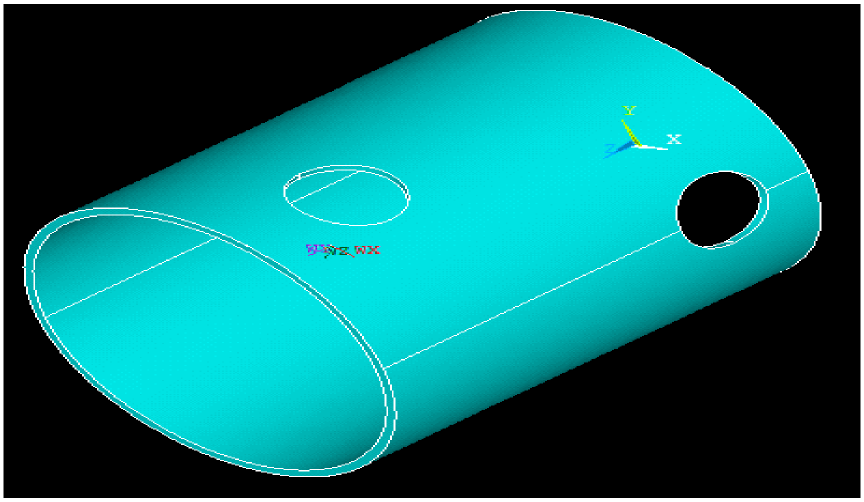
Table 35: CIP_2H_2D_Optres

From the above optimized values, it is evident that the stress is getting reduced as soon as 2nd hole goes closer to 1st hole. This is because stress relieving is happening when the holes are close to each other. Hence, in case of two holes, centre to centre distance, orientation around the circumference of the cylinder should be maintained as minimum as possible with out holes overlapping, to get the minimum stress in the structure.

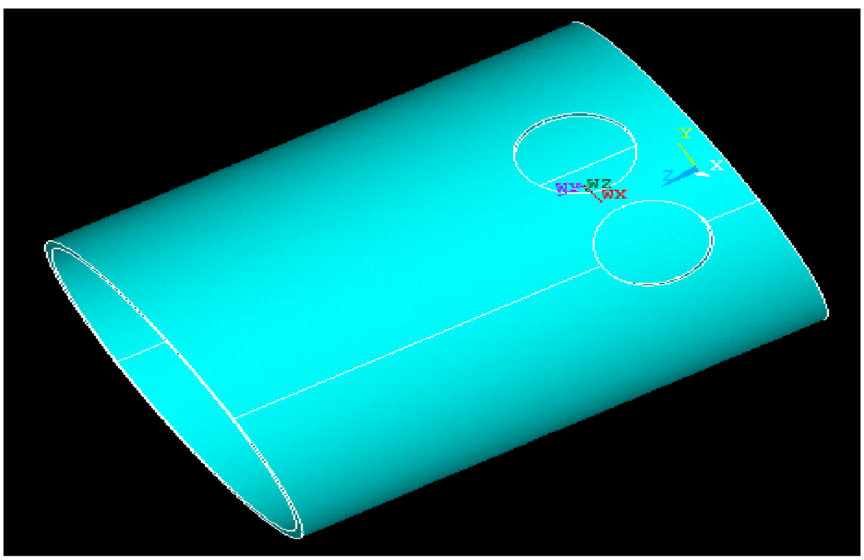
(ii) Plots

The below plots are generated based on values of (Z2, Th2) from table 35

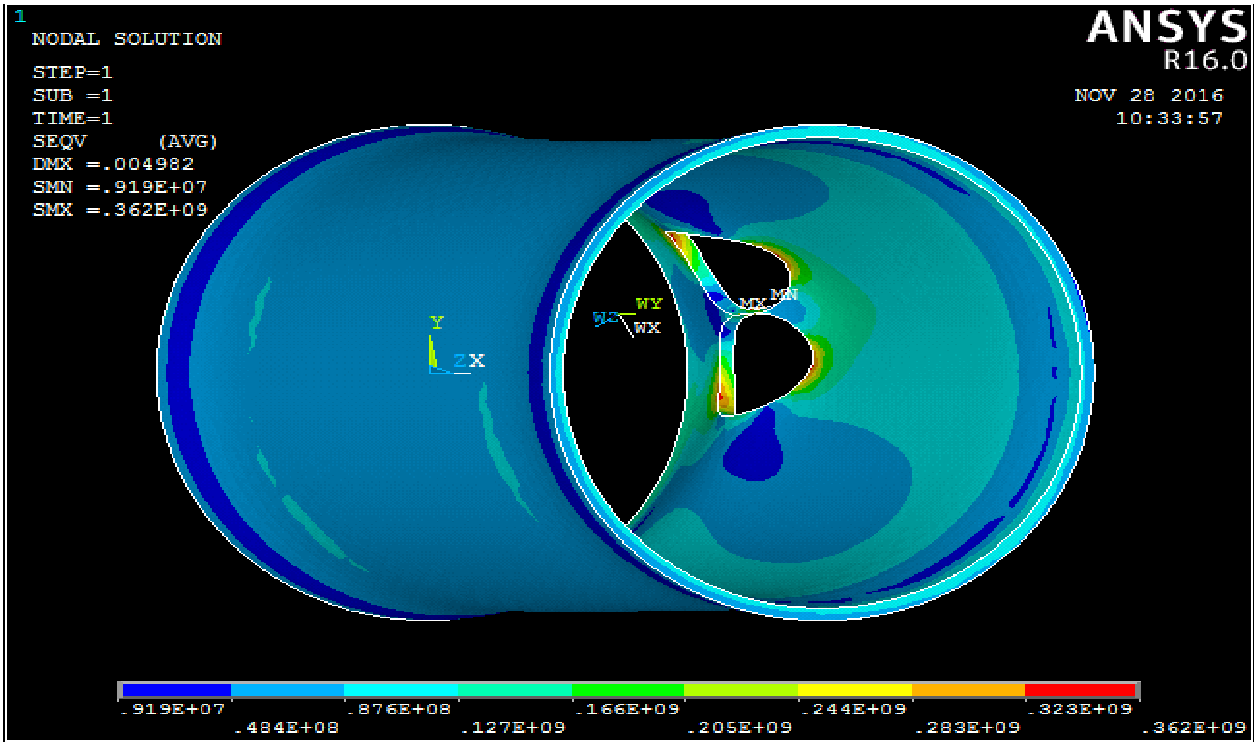
1) SET 1



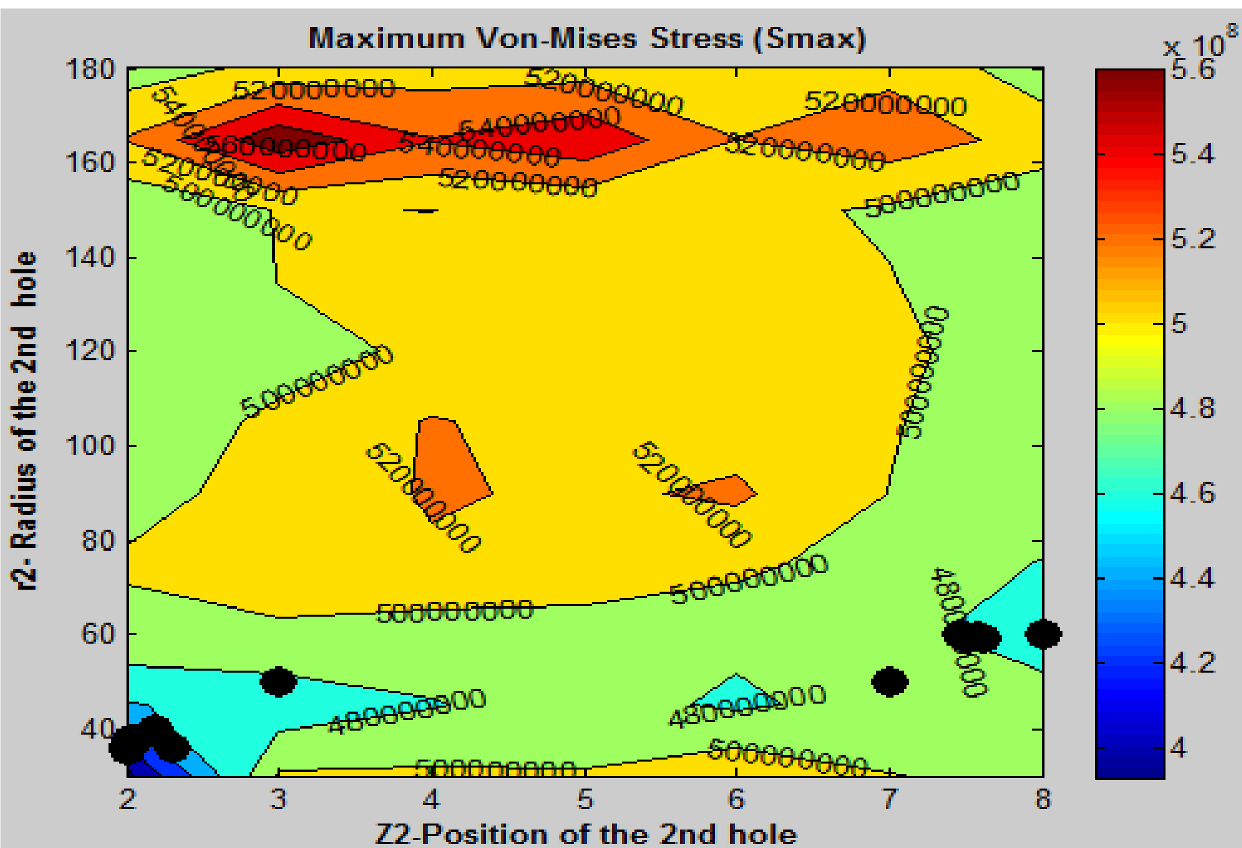
2) SET 2



(iii) Contour plot of maximum stress concentration



(iii) Stress Plot with Design Points



(5.2) Three Design Variables

Design Variables: Z2, Th2, R2- Location variables & Radius of 2nd hole

Objective Function: Minimizing the Maximum Von mises stress

Constraint Function: Side Constraints- Upper & Lower Bounds of the design Variables

(i) FMINCON results

SETS	X	XL	XU
1	[8 50 0.5]	[2 30 0.5]	[8 60 1.0]
2	[2 50 0.5]	[2 30 0.5]	[8 60 1.0]

Table 36: CIP_2H_3D_Boundary Limits

.

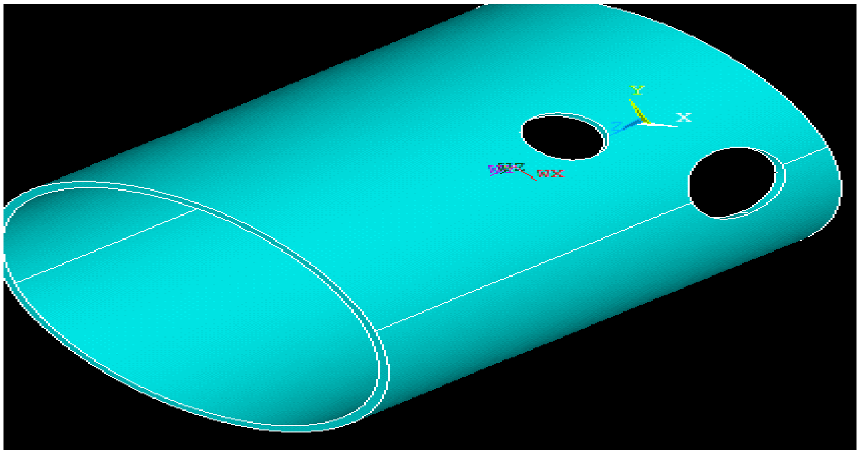
SETS	Z2	Th2	r2	Smax
1	3.8947	38.952	0.7053	425872310
2	2	30	1	393011090

Table 37: CIP_2H_3D_Optres

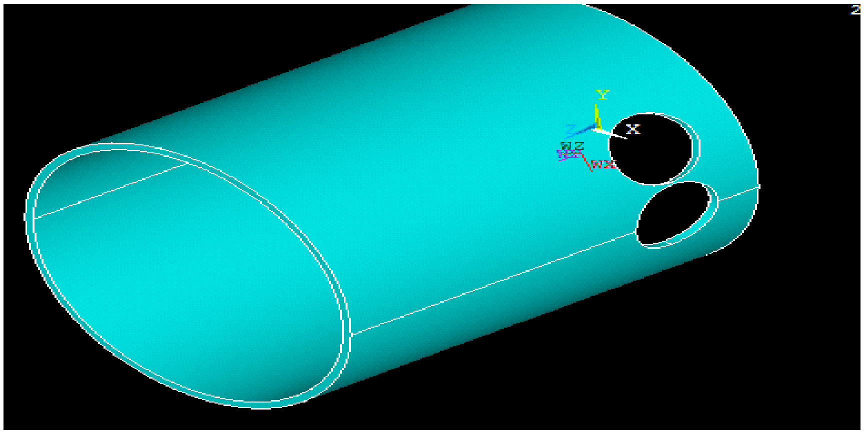
(ii) Plots

The below plots are generated based on values of (Z2, Th2, r2) from table 37

1) SET 1



2) SET 2



(6) Maximum Von-mises Stress Vs Theta of 2^{nd} hole

1^{st} Hole is fixed at Z1=2 & Th1=0.Radius of the holes 'r1' & 'r2' are taken as 1 m, Z2 is changed at each point and then correspondingly Smax is plotted with Th2 of 2^{nd} hole to generate below subplots

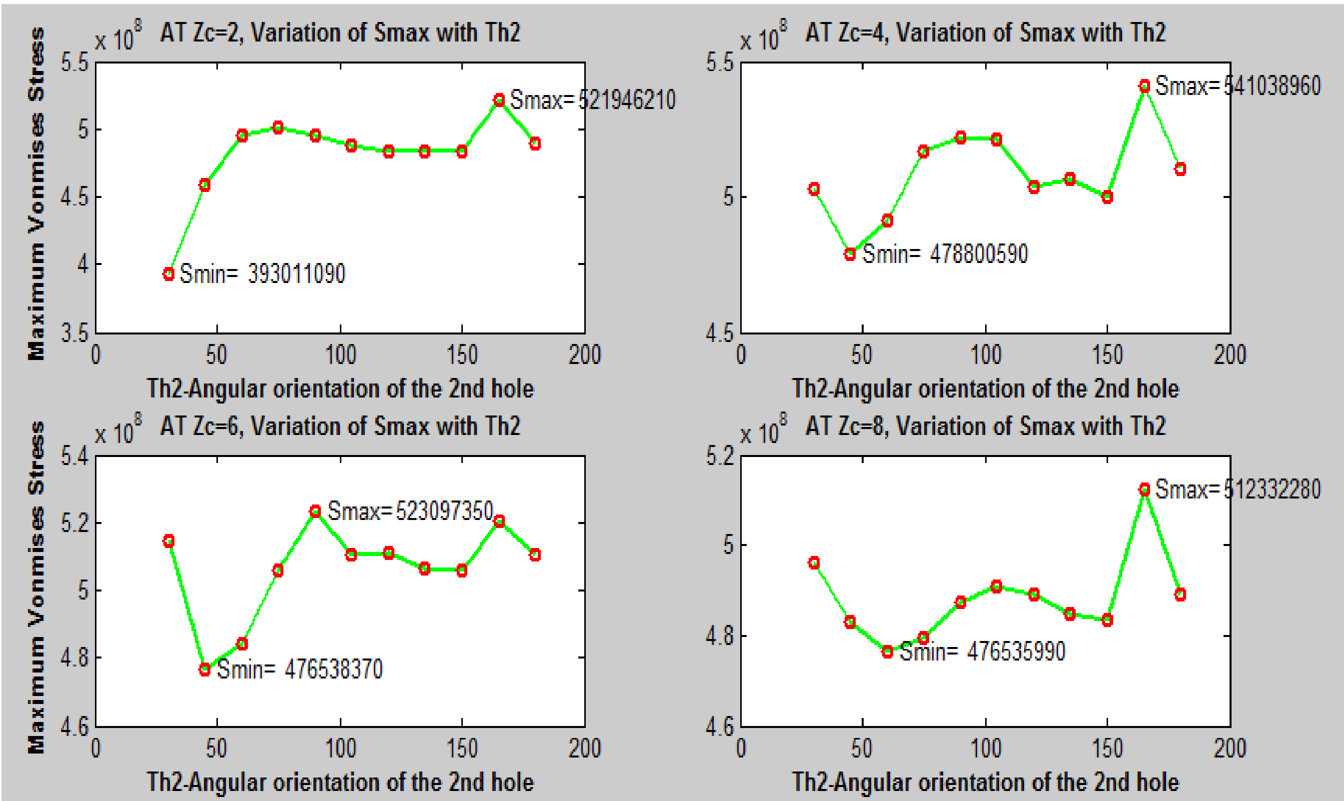


Figure 35: CIP_Smax Vs Th2

7) Maximum Von-mises Stress Vs Radius of 2nd hole

1st Hole is fixed at Z1=2 m & Th1=0 deg. Radius of 1st hole 'r1' is taken as 1 m, 2nd hole is made collinear with 1st by fixing it at Z2=2 m,Th2=30 deg. Only 'r2' is varied to generate below plot

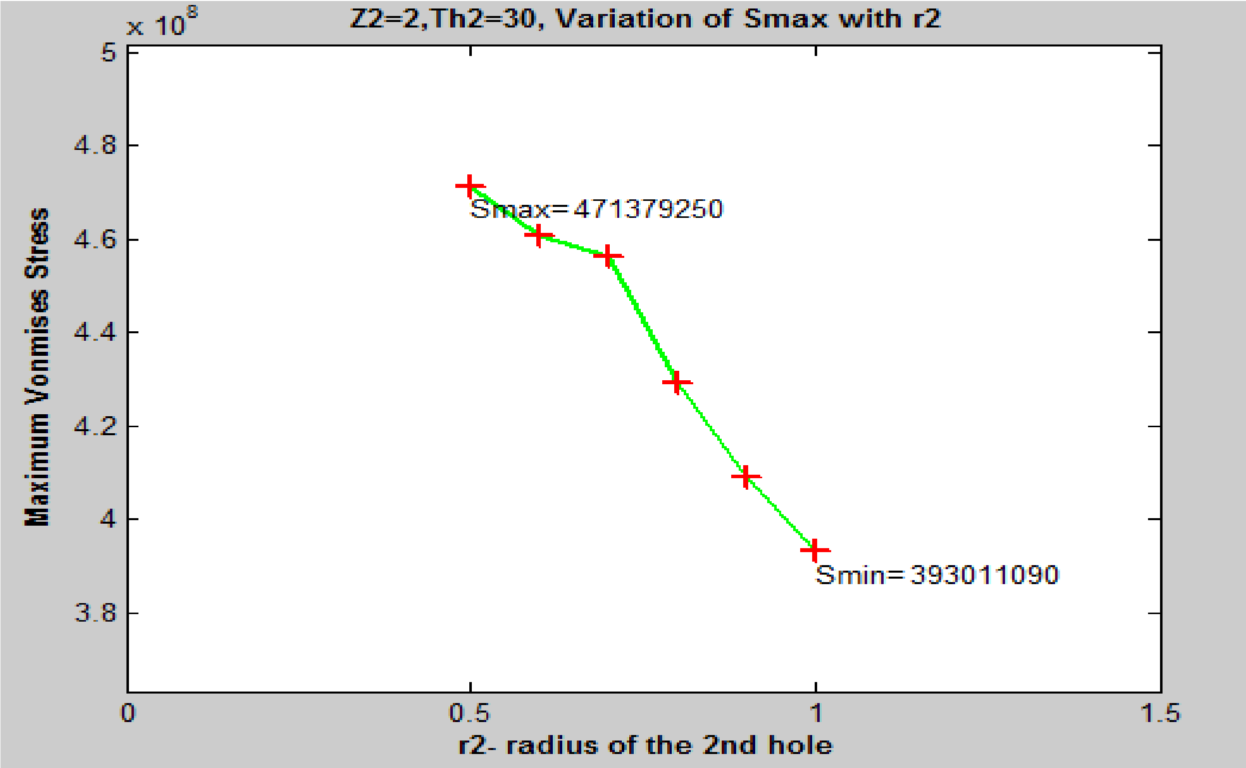
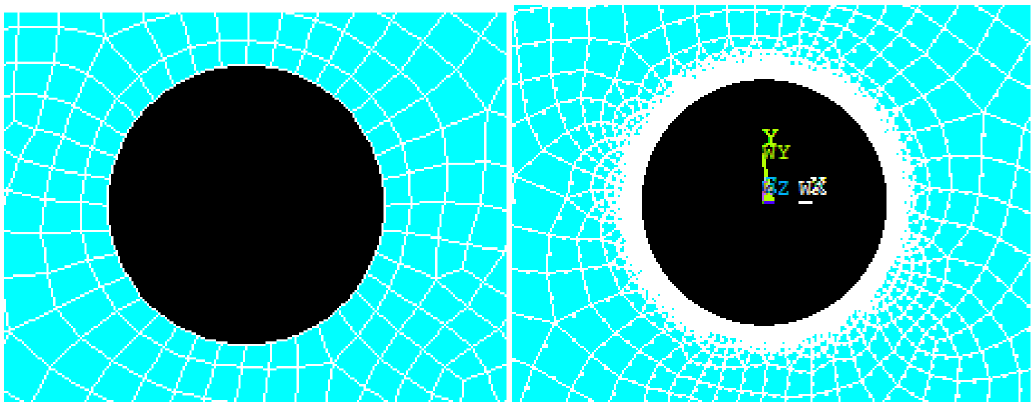


Figure 36: CIP_Smax Vs R2

From figures 35 & 36, it is evident that stress is getting relieved & drops to a lower value only when the holes are of same size and close to each other

4.4 Lessons Learnt

(i) Convergence Criteria



Elemental Von Mises Stress(Smax)	Nodal Von Mises Stress(Smax)	Error (%)
12.8046	15.7666	19
15.7222	15.8608	0.87

Table 38: Convergence_criteria

Notes

There are two type of Vonmises stresses available in ANSYS output. 1) Nodal Vonmises stresses, 2) Elemental Vonmises stresses. Whenever there is a discontinuity in the structure, values of these two stresses differ much, in order to reduce the gap, meshes around the discontinuity region should be refined. From table 38, it is clear that these two stress values are converged only after refining as shown in the figure above

(ii) Minimum Change in Design Variable using FMINCON

It is important to define step size for your design variables whenever FMINCON is used, otherwise the optimization get stuck at the initial points itself. It won't do search in the design space defined by user

options = optimoptions('DiffMinChange',1.0)

Chapter 5

Conclusion & Future Work

From this thesis work, we can conclude that, (i) for a single hole case, the best location depends totally on the type of load applied. (ii) Whenever two / three holes are present, they should be of same size and also kept as close as possible for better stress relieving. (iii) Interfacing ANSYS and MATLAB to automate the optimization is less time consuming, efficient in finding better results

Future work could be interfacing few more software's like Hyper-mesh to mesh the geometry and use ANSYS for analysis, can extend it to Composite plates/ cylinder with holes, can optimize the location parameters for other type of discontinuities like rectangular / elliptical holes in a plate and cylindrical shell

References

[1] Praveen Mirji- 'Optimization of rectangular plate with two holes subjected to in-plane static loading'- ISSN: 2229-5518

[2] M. Javed Hyder, M. Asif- 'Optimization of location & size if opening in a pressure vessel cylinder using ANSYS'- 'Elsevier-Engineering Failure Analysis', 15 (2008) p 1-19

[3] Ch. Ramesh, Dr. Bandaru Sateesh- 'Optimization of location, size of opening hole in a pressure vessel cylinder'- ISSN: 2445-0876; IF: RJIF 5.44

[4] Shantkumar B Bochare- 'Optimization of location and size of opening in a pressure vessel cylinder for spherical and elliptical head'- ISSN: 2249-3255

[5] Viraj H. Barge, Dr. S.S.Gawade, Vinaay Patil- 'Thermal-Structural Analysis & Optimization of pressure vessel using Finite Element Analysis'- ISSN: 2249-8974

[6] V G Ukadgaonker, R R Avargerimath & SD Koranne- 'Stress Analysis of an infinite plate containing two collinear elliptical holes under in-plane stresses at infinity' – Indian Journal of Engineering & Material Sciences Vol. 2, April 1995, pp. 62-79

[7] Shyam R Gupta, Ashish Desai- 'Optimization nozzle location for minimization of stress in pressure vessel'- ISSN: 2349-6010

[8] University of Alberta Website (http://www.mece.ualberta.ca/tutorials/ANSYS/)

[9] Professor Dr. Kent Lawrence, The University of Texas at Arlington, ANSYS Examples (http://mae.uta.edu/~lawrence/ANSYS/ANSYS_examples.htm)

Appendix A

Sample FMINCON Optimization script in MATLAB:

File name: ‘runfincon.m’

```
%%%%%%%%%%%%%% Beginning of the main program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%
function [history,searchdir] = runfmincon

% Set up shared variables with OUTFUN
history.x = [];
history.fval = [];
searchdir = [];

% Notes:
% Algorithm 'sqp' gives good results
% [X,Y]=meshgrid(11:2:39,-15:1:15); Best Section for Plate Optimization
% based on contour plots

% Design Parameters: X=[X2, Y2]- Centre Co-ordinates of 2nd hole
x=[11.3 0];xl=[11 -1];xu=[11.5 1];

options = optimoptions(@fmincon,'OutputFcn',@outfun,...

'Algorithm','sqp','Display','iter','DiffMinChange',0.1,'MaxFunEvals',5000);
% op=optimoptions(@fmincon,'Algorithm','active-
set','Display','iter','DiffMinChange',0.1,'TolFun',1e-2,'TolCon',1e-5);
xsol = fmincon(@PVCH_obj,x,[],[],[],[],xl,xu,[],options);

%%%%%%%%%%%%%% Subfunctions %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
 % Subfunction 1: Function to extract optimization history:
 %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
    function stop = outfun(x,optimValues,state)
     stop = false;

     switch state
         case 'init'
             hold on
         case 'iter'
         % Concatenate current point and objective function
         % value with history. x must be a row vector.
           history.fval = [history.fval; optimValues.fval];
           history.x = [history.x; x];
         % Concatenate current search direction with
         % searchdir.
           searchdir = [searchdir;...
                        optimValues.searchdirection'];
           plot(x(1),x(2),'o');
         % Label points with iteration number and add title.
         % Add .15 to x(1) to separate label from plotted 'o'
           text(x(1)+.15,x(2),...
                num2str(optimValues.iteration));
           title('Sequence of Points Computed by fmincon');
         case 'done'
```

```
            hold off
        otherwise
    end
  end

 %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Subfunction 2: Objective Function:
 function f = PVCH_obj(x)

    R1=5;R2=5;
    r=[R1 R2];
    XD=[x r];

% Current Working Folder/ Directory:

cd('C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H')

% Command to update Design Variables in ANSYS input file:
feval('Plate_opt',XD);

% DOS command to run ansys in batch mode:
dos('"C:\Program Files\ANSYS Inc\v160\ANSYS\bin\winx64\ansys160.exe"  -p
ane3fl -dir "C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H" -
j "file" -s read -l en-us -b -i
"C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H\IP_UDL.txt" -o
"C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H\Output.txt"');

delete('file.BCS','file.err','file.esav','file.full','file.log','file.mntr','
file.rst','file.stat');

 % Extracting Output saved in a text file from ANSYS:

VMS=load('VMS.txt');
gmax=max(abs(VMS));

f=gmax;
 end

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%%%%%%%%%%% End of Subfunctions %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

 % Saving Optimization/ Iteration history in a table format:
save('ophis7.mat','history','searchdir')

end

%%%%%%%%%%%%%% End of the main program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```

Sample script to update design variables in ANSYS input file:

File name: 'Plate_opt.m'

```
%%%%%%%%%%%%%% Beginning of the program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
function  Plate_opt(X)

fid = fopen('IP_UDL.txt','r+');
B=textscan(fid,'%s','Delimiter','');
A=B{1};
p=strfind(A,'*SET,X2');
e(1)=find(~cellfun(@isempty,p));
q=strfind(A,'*SET,Y2');
e(2)=find(~cellfun(@isempty,q));
l=strfind(A,'*SET,R2');
e(3)=find(~cellfun(@isempty,l));
s=strfind(A,'*SET,R1');
e(4)=find(~cellfun(@isempty,s));
a=[X(1) X(2) X(3) X(4)];

Y=cellfun(@num2str, num2cell(a), 'UniformOutput', false);
 for j=1:numel(e)
      b=strsplit(A{e(j)},',');
      n=max(size(b));
      b{1,n}=Y{1,j};
      c=strjoin(b,',');
      A{e(j)}=c;
 end
fid = fopen('IP_UDL.txt', 'w');
fprintf(fid, '%s\n', A{:});

fclose('all');


%%%%%%%%%%%%%% End of the program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```

Sample ANSYS I/P file in text format:

File name: 'IP_UDL.txt'

```
%%%%%%%%%%%%%% Beginning of the program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

/PREP7
!*
ET,1,PLANE183
*SET,L,100
*SET,W,50
*SET,R1,5
*SET,R2,5
*SET,X1,0
*SET,Y1,0
*SET,X2,11.4435
*SET,Y2,0.0187
*SET,T,20
!*
KEYOPT,1,1,0
```

```
KEYOPT,1,3,3
KEYOPT,1,6,0
!*
!*
R,1,T,
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,200e3
MPDATA,PRXY,1,,0.3
BLC5,0,0,L,W
CYL4,X1,Y1,R1
CYL4,X2,Y2,R2
FLST,3,2,5,ORDE,2
FITEM,3,2
FITEM,3,-3
ASBA,       1,P51X
FLST,2,1,4,ORDE,1
FITEM,2,4
DL,P51X, ,ALL,
FLST,2,1,4,ORDE,1
FITEM,2,2
SFL,P51X,PRES,-5,
!*
MSHKEY,0
SMRTSIZE,1
MSHKEY,0
CM,_Y,AREA
ASEL, , , ,       4
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
!*
AMESH,_Y1
!*
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
!*
FLST,5,8,4,ORDE,2
FITEM,5,5
FITEM,5,-12
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,S,_Y
CMDELE,_Y
!*
!*
LREFINE,_Y1, , ,1,1,1,1
CMDELE,_Y1
!*
FLST,5,8,4,ORDE,2
FITEM,5,5
FITEM,5,-12
CM,_Y,LINE
LSEL, , , ,P51X
```

```
CM,_Y1,LINE
CMSEL,S,_Y
CMDELE,_Y
!*
!*
LREFINE,_Y1, , ,1,1,1,1
CMDELE,_Y1
!*
FLST,5,8,4,ORDE,2
FITEM,5,5
FITEM,5,-12
CM,_Y,LINE
LSEL, , , ,P51X
CM,_Y1,LINE
CMSEL,S,_Y
CMDELE,_Y
!*
!*
LREFINE,_Y1, , ,1,1,1,1
CMDELE,_Y1
!*
FINISH
/SOL
/STATUS,SOLU
SOLVE
FINISH
/POST1
AVPRIN,0, ,
ETABLE,Volume,VOLU,
SSUM
*GET,Volume,SSUM, ,ITEM,VOLUME
*CFOPEN,C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H\Volobj,
txt,,
*VWRITE,Volume, , , , , , , , ,
(G16.8)
*CFCLOS
ETABLE,VMS,S,EQV
*VGET,VMS,ELEM,1,ETAB,VMS, , ,2
*CFOPEN,C:\Users\Sound\Desktop\Thesis\Ansys_Projects\PV_CH\Plate_W_2H\VMS,txt
,,
*VWRITE,VMS(1), , , , , , , , ,
(G16.8)
*CFCLOS
!*
/EFACET,1
!PLNSOL, S,EQV, 0,1.0
!PLESOL, S,EQV, 0,1.0
!*


%%%%%%%%%%%%%% End of the program %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```

Biographical Information

Soundararaj Thangavel received his Bachelors' degree in Mechanical Engineering from The College of Engineering Guindy, Anna University, Chennai, Tamilnadu, India. After his graduation he worked as an Equipment Engineer in Saipem India Projects Limited for two years. He began his Masters' program in Mechanical Engineering in fall 2014 at The University of Texas at Arlington, Arlington, Texas, USA and received his degree in December 2016. He is interested in Design and Structural Engineering field of Mechanical Engineering