ASSIGNMENT

Module Code AME 503

Module Name Finite Element Analysis

Course M.Sc. [Engg] in Machinery Design.

Department Mechanical & Manufacturing Engineering.

POSTGRADUATE ENGINEERING AND MANAGEMENT PROGRAMME – (PEMP)

Name of the Student Prabhaker.P

Reg. No BAB0911009

Batch Full-Time 2011.

Module Leader Dr. Vinod kumar Banthia

M.S.Ramaiah School of Advanced Studies

Postgraduate Engineering and Management Programmes(PEMP)

#470-P Peenya Industrial Area, 4th Phase, Peenya, Bengaluru-560 058

Tel; 080 4906 5555, website: www.msrsas.org

Declaration Sheet			
Student Name	PRABHAKAR.P		
Reg. No	BAB0911001		
Course	MACHINERY DESIGN	Batch	Full-Time 2011
Batch	FT11		
Module Code	AME 503		
Module Title	FINITE ELEMENT ANALYSIS		
Module Date	09 01 2012	to	21 01 2012
Module Leader	Dr. VINOD KUMAR BANTHIA		

Extension requests:

Extensions can only be granted by the Head of the Department in consultation with the module leader. Extensions granted by any other person will not be accepted and hence the assignment will incur a penalty. Extensions MUST be requested by using the 'Extension Request Form', which is available with the ARO. **A copy of the extension approval must be attached to the assignment submitted**.

Penalty for late submission

Unless you have submitted proof of mitigating circumstances or have been granted an extension, the penalties for a late submission of an assignment shall be as follows:

- Up to one week late: Penalty of 5 marks
- One-Two weeks late: Penalty of 10 marks
- More than Two weeks late: Fail - 0% recorded (F)

All late assignments: must be submitted to Academic Records Office (ARO). It is your responsibility to ensure that the receipt of a late assignment is recorded in the ARO. If an extension was agreed, the authorization should be submitted to ARO during the submission of assignment.

To ensure assignment reports are written concisely, the length should be restricted to a limit indicated in the assignment problem statement. Assignment reports greater than this length may incur a penalty of one grade (5 marks). Each delegate is required to retain a copy of the assignment report.

Declaration

The assignment submitted herewith is a result of my own investigations and that I have conformed to the guidelines against plagiarism as laid out in the PEMP Student Handbook. All sections of the text and results, which have been obtained from other sources, are fully referenced. I understand that cheating and plagiarism constitute a breach of University regulations and will be dealt with accordingly.

Signature of the student		Date	
Submission date stamp (by ARO)			

Signature of the Module Leader and date	Signature of Head of the Department and date

Abstract

The Finite element analysis module gave the idea of the approach step involved in finite element method for solving the problem and also validating the result when obtained as the solution based on which the assignment is solved. The Part-A assignment is the debate on convergence and various types of convergence used in FEA like h-method, p-method and hp-method. These are used to obtain the better results in FEA all these three types are widely used but each has its own limitations and constrains it is required the analysist should have the clear idea based on the problem definition should choose the proper convergence obtain the proper result. The debate is about for structural analysis which convergence provides the best result is discussed.

The Part-B is the Hertz contact stress problem for the given diameter and the load the proper idealization is done and by applying the boundary conditions and loads solution is obtained results of various plots and graphs are shown were for solving analysis tool Ansys is used for solving in which all the operations of modeling, element selection, boundary conditions etc, are carried out and results are obtained and to validate the result hand calculations are done and the results obtained from the Ansys are verified the hand calculations are attached in appendix-1. From the Part-B it is required to understand the result obtained as this is used as the bench mark model for solving the Part-C assignment.

The Part-C is the problem is base on gears in contact were the given IGES file is meshed using the dedicated meshing software Hypermesh. And it is taken as the input to the Ansys then the element selection, boundary conditions, contact properties etc, were assigned and the solution is obtained to validate the obtained by consider in the Part-B Hertz contact solution as bench mark model as the gear has the involute profile the contact between the rollers is similar to the contact between the gears and also the element selection and contact properties are same for Part-B and Part-c. In Part –C is solved in two steps first the boundary conditions and loads applied to the single tooth and the result is obtained and then the result obtained for the gears in contact both the results were compared and verified.

Contents

Declaration Sheet ii
Abstract iii
Contents iv
List of Tables vi
List of Figures vii
1.1Introduction: 9
1.2Overview of P, H & HP convergence: 9
1.3Comparison of p, h & hp Convergence. 9
1.4Accuracy and degrees of freedom in convergence: 10
1.5 p-method is better than h-method and hp-method: 10
1.6 Conclusion: 11
Part-B 12
2.1 Decisions for idealizations: 12
Assumptions: 12
2.2 Type and size of element used: 12
2.2.1 Plane 82: 12
2.2.2 TARGE169: 13
2.2.3 CONTA175: 13
2.3 Size of the element: 14
2.4 Loads and boundary conditions: 14
2.5 Solution strategy: 15
2.6 Contact stress distribution along the width of contact surface: 16
2.7 Maximum shear stress in the cylinders and its location: 18
2.8 Approach distances between the centers of cylinders: 18
2.9 Distribution of normal stresses along the depth of the cylinders at the mid-point of the contact patch: 19
Comparison of analytical solution with Ansys solution: 20
PART-C 21
3.0 Introduction: 21

3.1 The scheme used for creating the model: 21
3.2 Element sizes: 22
3.2.1 Element type: 22
3.2.2 Contact175: 22
3.2.3 Targe169: 22
3.3 Loading and boundary conditions: 22
3.3.1 Loading and boundary condition for single gear: 23
3.3.2 Loading and boundary condition for gear in contact: 23
3.3.3 Stress for single gear in X-axis: 24
3.3.4 The contact stress for gear tooth in contact at X-axis: 25
3.3.5 Stress for single gear in Y-axis: 25
3.3.6 The contact stress for gear tooth in contact at Y-axis: 26
3.3.7 Shear stress for single gear in XY-axis: 27
3.3.8 The shear stress for gear tooth in contact at XY-axis: 28
3.3.9 Root stress for single gear: 28
3.3.10 Root stress: 29
3.4 Comparison of result of single and contact pair gears: 30
3.5.1 Justifications for Hertz contact theory validation: 31
2.5.2 Comparing the stresses on the teeth: 32
3.6 Root stress bench mark solution: 32
4.0 Learning outcome: 34
References **35**
Appendix-1(Analytical solution for Part-B) **36**

List of Tables

Table No.	Pg.No.
Table 1. 1 Comparison of p, h & hp.....................................	10
Table 2. 1 Comparison of results of solutions......................	20
Table 3. 1 Comparison of results of gear	30

List of Figures

Figure No.	Title of the figure	Pg.No.
Fig 1. 1	Polynomial orders of p & h method	10
Fig 1. 2	Comparison of h & p elements	10
Fig 1. 3	Result comparision of p, h & hp method	11
Fig 2. 1	Plane 82 element	12
Fig 2. 2	TARGE169 element	13
Fig 2. 3	CONTA175 element	13
Fig 2. 4	Mesh creation in geometry	14
Fig 2. 5	Boundary conditions	15
Fig 2. 6	Contact patch width	17
Fig 2. 7	Stress induced along the load applied	17
Fig 2. 8	Maximum shear stress in contacts	18
Fig 2. 9	Deformation in contacts	19
Fig 2. 10	Graph showing stresses in X,Y &Z axis	20
Fig 3. 1	Mesh created in hyper mesh	21
Fig 3. 2	Boundary condition for single gear	23
Fig 3. 3	Boundary condition for gear in contact	24
Fig 3. 4	Stress in X-axis for single gear	24
Fig 3. 5	Stress in X-axis for contact gear	25
Fig 3. 6	Stress in Y-axis for single gear	26
Fig 3. 7	Stress in Y-axis for contact gear	26
Fig 3. 8	Shear stress for single gear	27
Fig 3. 9	Shear stress of contact gears	28
Fig 3. 10	Root stress for single gear	29
Fig 3. 11	Root stress for contact gears	29
Fig 3. 12	Stress developed in single gear Fig 3. 13 Stress developed in single gear	30
Fig 3. 14	Relation with Hertz contact and gear contact	31
Fig 3. 15	Contacts of gears	31
Fig 3. 16	Stress distribution in Hertz contact and gear contact	32
Fig 3. 17	Elleptical patch obtained in Hertz contact and gear contact	32

Fig 3. 18 Photo elastic theory idealization for single gear 33

1.1 Introduction:

In finite element method results are obtained by simplifying the problem, were the object is descretised into small elements and the results are obtained. These results depend on various parameters like element selection, Mesh quality and Boundary conditions etc. The debate is about various types of convergence used in FEA to obtain the better result. As the solution in FEA depends on various factors and the software gives the solution which is more than the satisfactory level but not the exact solution, The solution depends on modeling of elements which can also referred as P, H and HP type convergence.

1.2 Overview of P, H & HP convergence:

In case of H convergence the closer solution is obtained by increasing the number meshes were polynomial is constant as in this type the result obtained is constant the finer the mesh the accurate result is obtained but this takes more calculation time.

In case of P convergence by increasing the order of element by imparting polynomial function closer result is obtained were the mesh is constant, the result variation is linear therefore even with the course mesh the better result is obtained.

In case of HP convergence is the combination of the H and P convergences were the mesh will is very fine as well as the order of the polynomial will be increased to obtain the accurate result.

1.3 Comparison of p, h & hp Convergence.

Description	H-convergence	P-convergence	Hp-convergence
Mesh size	Fine mesh	Coarse mesh	Fine mesh
Polynomial	Lower order	Higher order	Mixed
Plot of result	Constant	Linear	Mixed
Computation time	High	Less	Moderate
Accuracy	Moderate	Precise	Very precise
Results at sharp corners	Precise	Singularity error occurs	Precise
Results in the curved profile	Poor due to less flexibility	Good due to higher polynomial order	Very good

Table 1. 1 Comparison of p, h & hp

1.4 Accuracy and degrees of freedom in convergence:

In h convergence in order to obtain the better result it is required to do fine meshing as each node has N-DOF (degrees of freedom) by increasing the number of nodes by fine mesh the degrees of freedom is increased. In p convergence number of nodes in the element is increased in turn increases the degrees of freedom. In many cases the selection of convergence depends on accuracy and computational time based on that convergence is selected .The hp is perfect method to solve non linear problems were it automatically refines the grids around the singularities and automatically increases the polynomial order were the result required to be smooth thus the degrees of freedom gets increased.

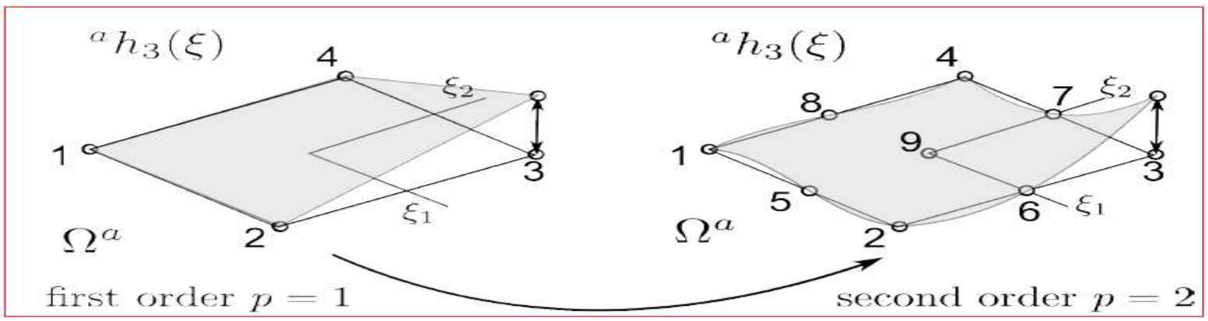


Fig 1. 1 Polynomial orders of p & h method

The Fig 1.1 shows that increase in polynomial result in increase in node in turn increase the flexibility and DOF in the element which provides better result.

1.5 p-method is better than h-method and hp-method:

While comparing all these methods h-method is widely used as it is initially developed and available in all initial software developed for meshing the h-refinement method has been more widespread in the commercial programs than the p-refinement method. But the mathematical approach and literature proves that p-method is the best method than h-method. By simple mesh and with less iteration solution is obtained in p-method. And for complicated profiles of curved portion h-method is not suitable as shown in the fig 1.2 below

Fig 1. 2 Comparison of h & p elements

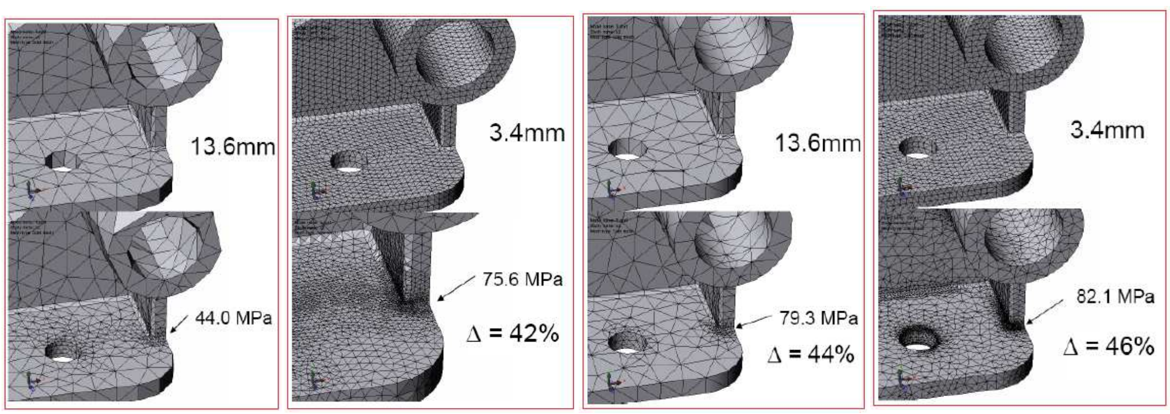
While comparing the results of the same shape by h-method, p-method &hp-method as shown below in the Fig-1.3 we can observe that the hp-method shows the better result than the other two methods and p-method shod shows the better result only after the fine refinement of the mesh and the h-method provides the better result even in the coarse mesh. From this we can eliminate the hp-method even though it provides the better result due to the following

i)It has fine mesh as h-method increases computation time.

ii)It provides accurate result than other two methods for structural analysis such accuracy may not be required in general cases.

h-method is the initially developed method which is widely used accuracy will be increased by increasing the mesh size even in the fine mesh it is not providing so accurate result as shown in the Fig below.

In p-method mesh size is larger than the h-method and hp-method and only the polynomial order is increased which provides better results than h-method and closer result to hp-method were the computation time is less than the other to hence p-method is more suitable method than other two method.



a)Initial mesh b)h-method c)p-method d)hp-method

[5]Fig 1. 3 Result comparision of p, h & hp method

1.6 Conclusion:

As the debate is about the convergence for the structural analysis the result obtained from the p-method is sufficient as it has the linear variation over the deformation, stress and strain and quadratic variation over the element of axial displacement and with coarse mesh the better result is obtained the p-method is better than other two method.

Part-B

2.1 Decisions for idealizations:

As in the gear while meshing the load along the width is uniformly distributed as in spur gear the contact takes place face to face at a time, The reaction along the thickness of the gear will be the same therefore the result obtained at any contact segment will be same, The problem can be idealized to 2D problem.

In the gear profile when it is considered as 2D only the results are obtained from the nodes in contact while meshing therefore for this problem gears with meshing can be idealized as two cylinders in contact to derive Hertz contact stresses. The half portion of the cylinders is idealized and the loads are applied as the cylinders will not be changing its orientation while analyzing.

The problem can be classified under the plane strain condition as there is no stresses acting in its length therefore deformation along the length is considered as zero and the load is acting only in the face of teeth (in transverse direction).

While both the cylinders are in contact only a small surface to surface contact exist in order to do analysis fine mesh can be created only around the surface which is in contact in order to reduce the computing time. The conditions of flexible to flexible contact with deformable bodies are applied to the nodes which are in contact.

Assumptions:

- In contact it is assumed there is no frictional loss.
- The material is considered to have same material property throughout the section.

2.2 Type and size of element used:

2.2.1 Plane 82:

Solid- 8node82 is selected to obtain Plane 82. The Plane 82 is 2D higher order 8 nodded elements. It provides more accurate result for irregular shapes and has very good displacement pattern and most suitable for curved geometry.

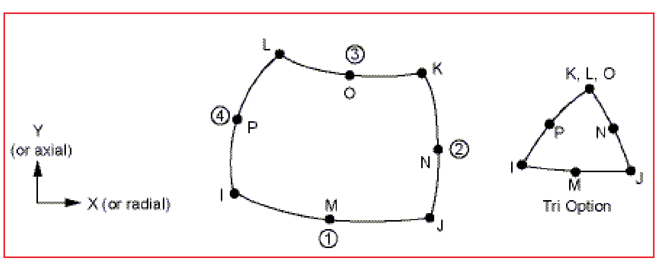


Fig 2. 1 Plane 82 element

The pressure can be applied to nodes which is shown inside the circle as each node has 2 degrees of freedom the element will have 16 degrees of freedom which is well suited for contact problem analysis were it is to model fine displacement at the nodes in contact.

2.2.2 TARGE169:

Contact – 2D target169 is selected for TARGE169. It is used to represent target surface for contact elements describing the boundary of deformable body which are in contact with target surface. It provides flexible to flexible contact system which is required for Hertz contact analysis.

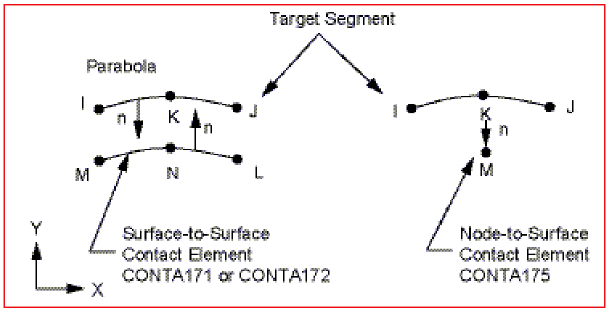


Fig 2. 2 TARGE169 element

As the problem solved is contact problem between two cylindrical rollers were node to surface contact is used by selecting CONTA175.

2.2.3 CONTA175:

Contact-point to surface 175 is selected for CONTA175. It is used to represent the contact and sliding between the two surfaces between node and surface or between line and surface.

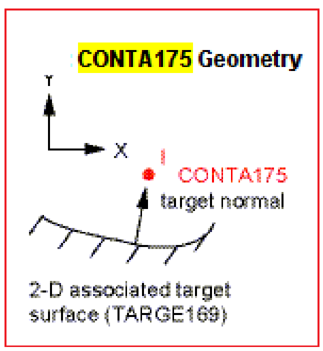


Fig 2. 3 CONTA175 element

In this problem as shown in the above Fig 2.3 the surface of TARGE169 is made to contact with CONTA175.

2.3 Size of the element:

As the h-method is adopted for solving the problem it is required a fine mesh to obtain a better result as fine mesh throughout the section increases the computation time. In order to reduce the computation time a small boundary is created around the contact and meshed to the size of 0.1mesh size as mapped and rest of the portion is free meshed to larger element size as shown below in the Fig 2.4

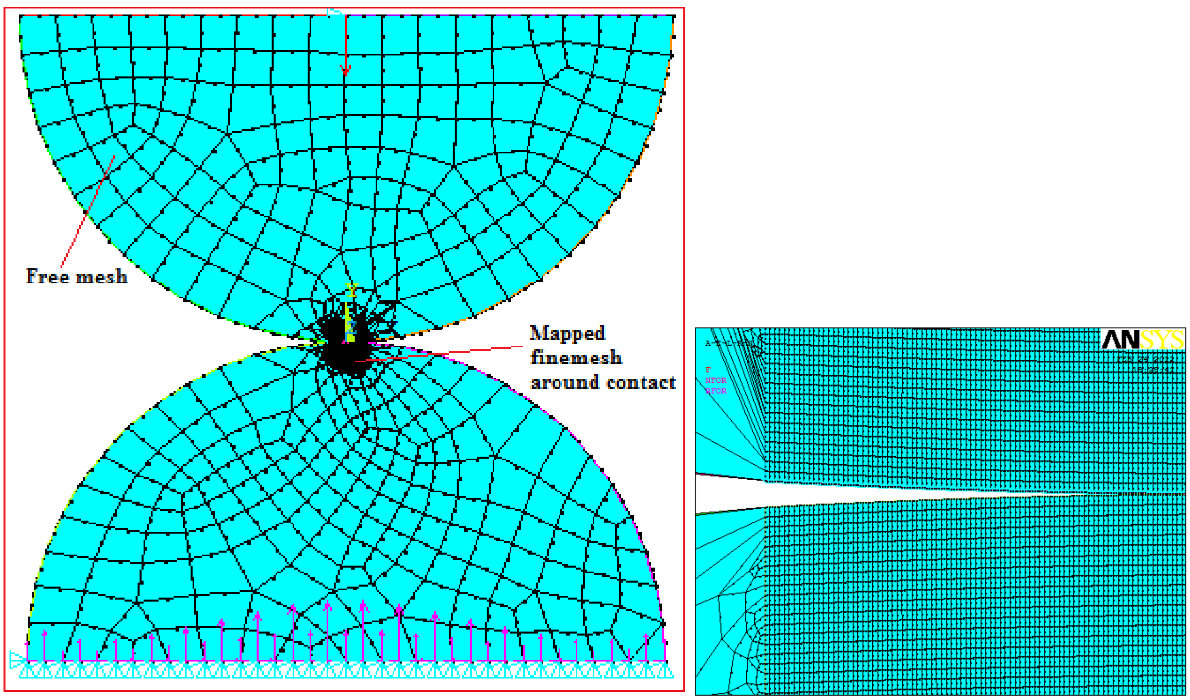


Fig 2. 4 Mesh creation in geometry

2.4 Loads and boundary conditions:

For this analysis only the two half portion of the cylinders are considered in which the load is applied to one half portion of the cylinder, in the other half no load is applied in order to avoid the displacement of the bottom cylinder in Y axis as load applied in top cylinder is in Y axis. The bottom cylinder Y axis is constrained along with this X axis is also constrained to avoid uneven deformation takes place therefore X and Y axes are together constrained.

In the top half cylinder at the middle node were load is to be applied only the X axis is constrained as to avoid deformation in the area surrounded by the node in top half cylinder therefore middle node in X axis is constrained so it will not move in Y axis while application of load. Now the given load of 1049 is applied in negative direction on the same middle node as constant load which is uniformly distributed throughout the top half of the cylinder.

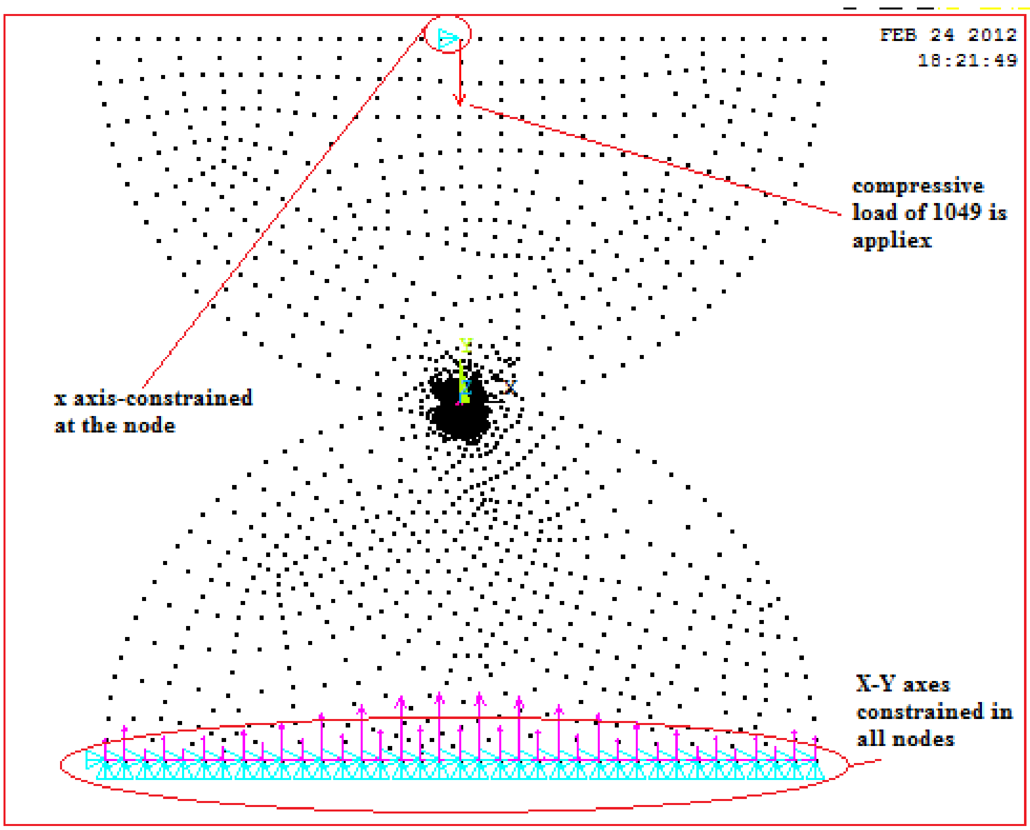


Fig 2. 5 Boundary conditions

2.5 Solution strategy:[5]

To cross check the result obtained from Ansys the result is checked with analytical solution by substituting the following formulas.

i) Maximum contact pressure occurs along the center line of contact.

$$\boxed{p_0 = \frac{2 \cdot F/l}{\pi \cdot b}}$$

ii) Contact patch width.

$$\boxed{b = \left[\frac{2 \cdot F/l}{\pi} \cdot \frac{D_1 \cdot D_2}{D_1 + D_2} \cdot \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right) \right]^{1/2}}$$

iii) Maximum shear stress

$$\boxed{\text{Maximum shear stress} = 0.3 \text{ x Normal stress}}$$

iv) Stresses induced in the cylinder along the X- axis, Y- axis and Z- axis.

$$\sigma_z = -2p_0 \cdot v\left[\left(1+\left(\frac{y}{b}\right)^2\right)^{1/2} - \frac{y}{b}\right]$$

$$\sigma_x = -p_0 \cdot \left(1+\left(\frac{y}{b}\right)^2\right)^{1/2} \cdot \left[2-\left(1+\left(\frac{y}{b}\right)^2\right)^{-1}\right] - 2\cdot\frac{y}{b}$$

$$\sigma_y = -p_0 \cdot \left(1+\left(\frac{y}{b}\right)^2\right)^{-1/2}$$

v) Approach distance between the centers of the cylinders by Roark's approach solution.

$$\delta = \frac{2 \cdot F/l \cdot \left(1-v^2\right)}{\pi \cdot E}\left(\frac{2}{3} + \ln\frac{D_1}{b} + \ln\frac{D_2}{b}\right)$$

2.6 Contact stress distribution along the width of contact surface:

The contact stress is calculated to identify the stress distribution in width in contact as a small area is created around the for analysis while the rollers are in contact not the total width of fine mesh created area will be in contact only a point will be in contact and the stresses will be developed only in the small width which is identified from the graph shown below Fig 2.6 obtained from Ansys were in x-axis dimensions of width is plotted and in Y-axis stress developed in the Y-axis is plotted the result obtained from the graph are the maximum compressive stress experienced by the roller is 878.137 and the graph is in "U" shape as the stress distribution is gradually increasing along the width.

The Stress result obtained by the analytical calculation attached to appendix is for compressive in Y axis is 889.93MPa and the result compressive stress in the graph is 878.137 MPa. The variation in the result is 1.33% which is within the limit.

From the graph the contact width obtained is 1.54 which is by one division in graph is 0.754 and the graph covers 2 division therefore the contact width is 1.54 and the analytical result obtained by calculation is 1.54 which is equal as obtained from the graph.

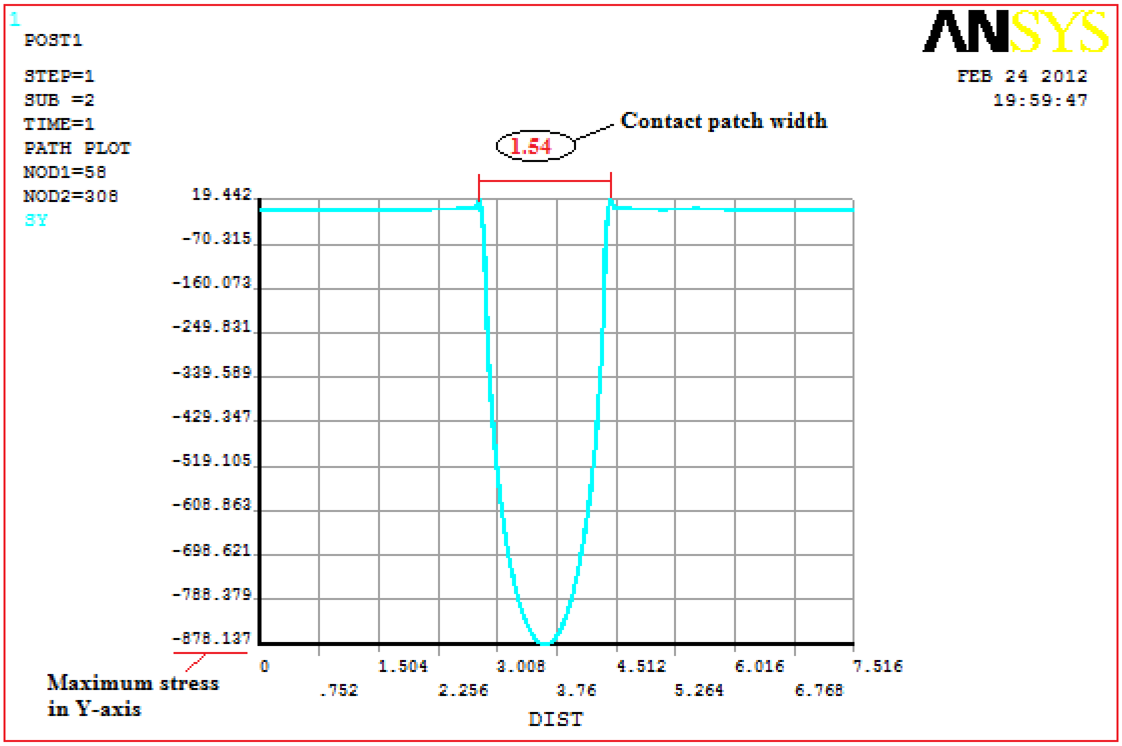


Fig 2. 6 Contact patch width

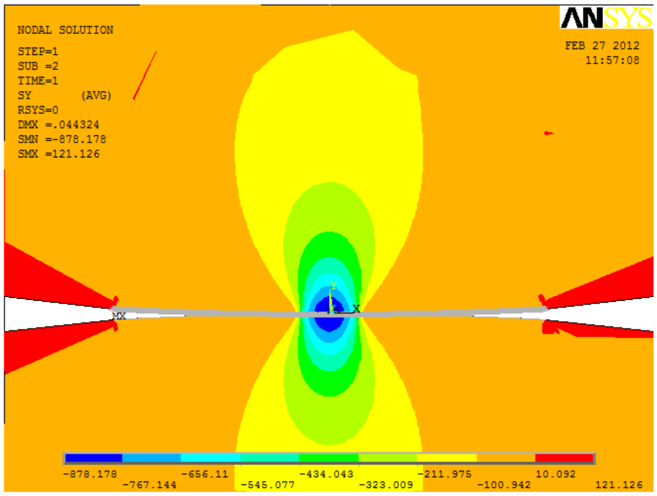


Fig 2. 7 Stress induced along the load applied

2.7 Maximum shear stress in the cylinders and its location:

The shear stress will be maximum at 45° shear stress develops as the rollers in contact oppose each other when the load is applied. The magnitude of shear stress depends on the force and type of contact. In the figure it can be noticed that the stresses are plotted at an angle which is shown as the blue and red region. The maximum shear stress obtained from Ansys is shown in the Fig 2.8 the result obtained is 218.60MPa. And the result obtained by the analytical solution attached in appendix is 263.4MPa and the deviation in the result is about

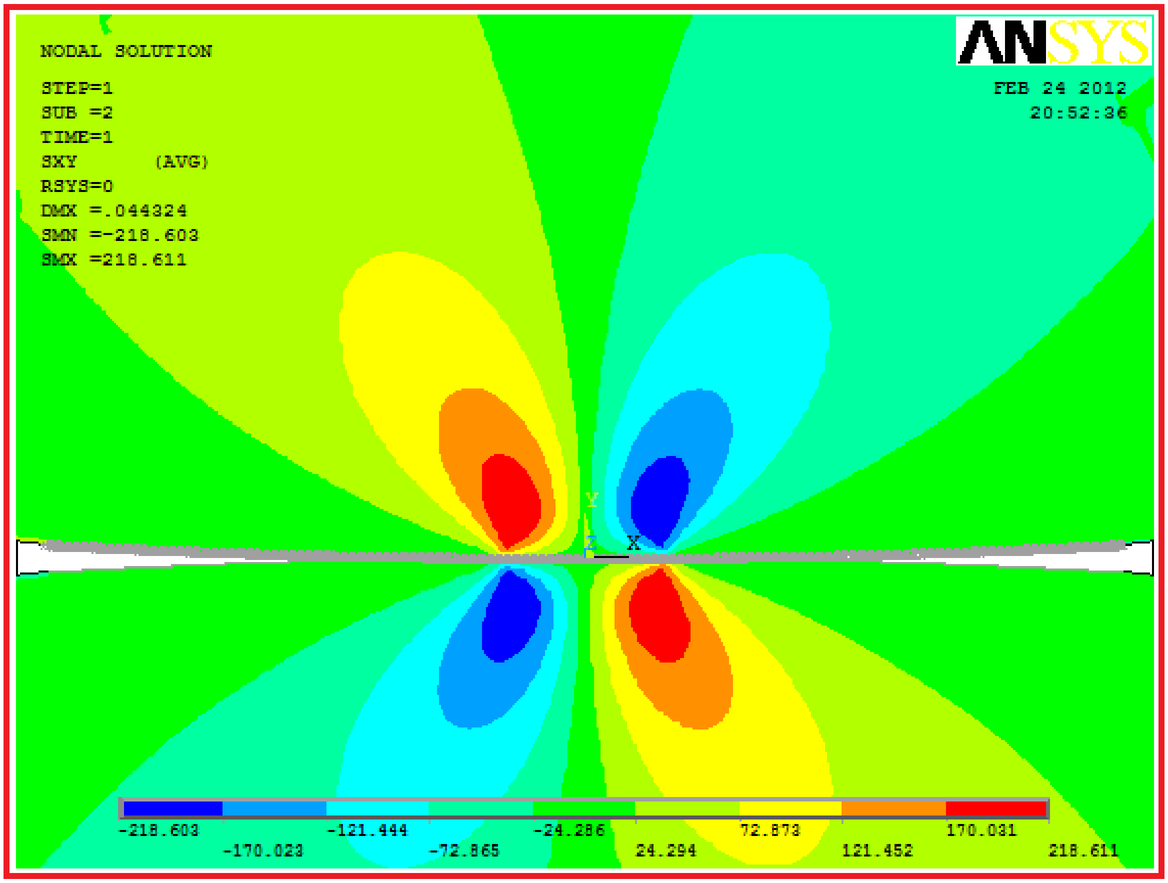


Fig 2.8 Maximum shear stress in contacts

2.8 Approach distances between the centers of cylinders:

The approach distance between the cylinders refers to when the two cylinders are in contact and the load is applied to the one cylinder it under goes some deformation which will be of elastic or plastic deformation this differs by the load applied and the material property. But in this case it is elastic deformation and the observation from the graph is when the compressive load of 1049 MPa is applied to the upper cylinder as the bottom cylinder is fully constrained the upper cylinder tries to penetrate inside the bottom cylinder and the bottom cylinder will resist the upper cylinder but still some elastic deformation of 0.044mm takes place in the cylinder.

As the result obtained is compared with the analytical solution the result obtained from the Ansys is 0.044mm and the result obtained by the analytical solution is 0.048 the variation in the result is of the accepted limit.

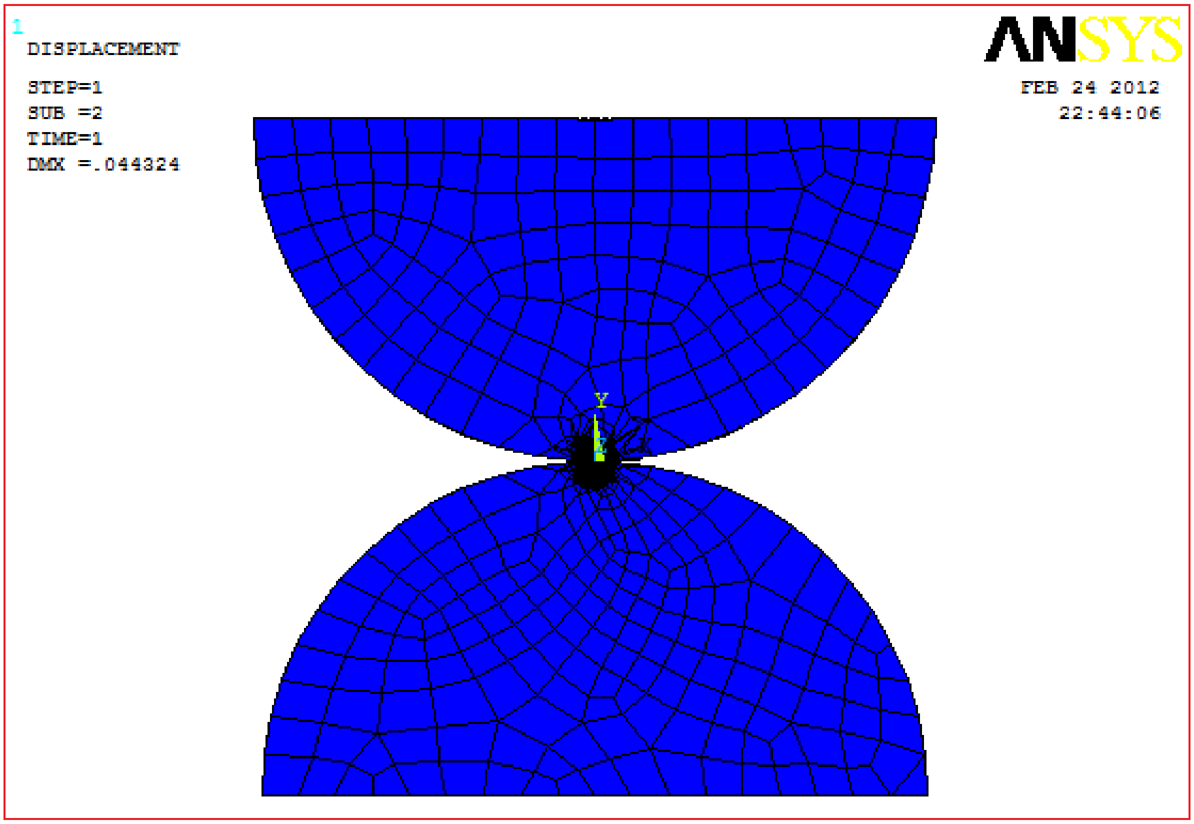


Fig 2. 9 Deformation in contacts

2.9 Distribution of normal stresses along the depth of the cylinders at the mid-point of the contact patch:

The distribution of normal stresses along its depth is plotted in X, Y & Z axes are shown in the Fig 2.10 in which X axis is plotted with distance and in Y axis the values of the stresses are plotted. Purple colour line in graph represents stresses in X axis. The observations are the compressive stress is 878.133MPa maximum in the surface which is in contact and gradually reduces to the surface away from contact but compared to other two axes it has very less stress generated at the distance of 1.22mm the stress generated is 28.95MPa. The stress acting in Y axis is shown in violet colour line in graph the observation are the stress at the contact surface is 878.133MPa the stress is gradually reducing in the surface which is away from the contact surface.

As the load applied is in Y axis the stress is higher than other two axes ax 1.22mm distance away from the contact surface the stress obtained is around 460MPa. The stress along z axis is represented in red colour line in the graph. The stress is very less compared to other two axis at the contact surface were the stress of 538.46MPa is acting at the contact and gradually reduces to the surface away from the contact.

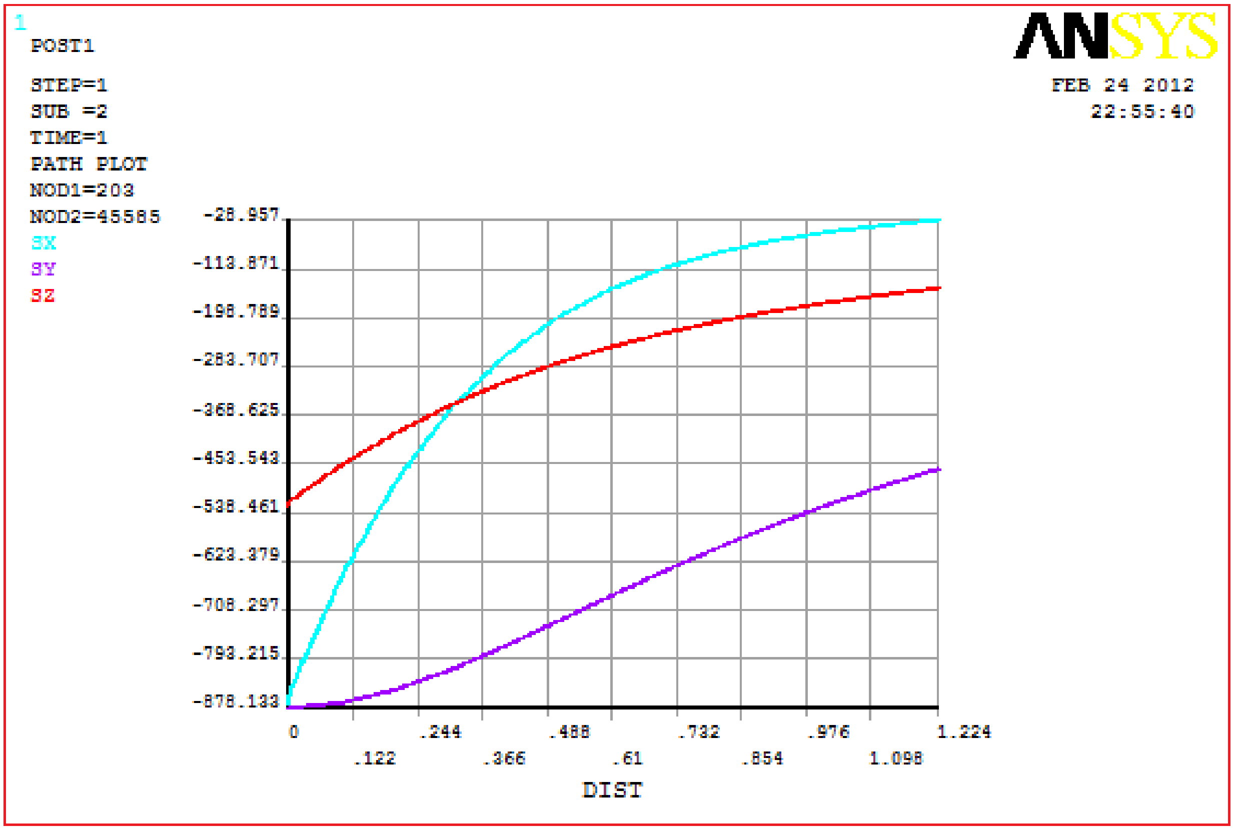


Fig 2. 10 Graph showing stresses in X,Y &Z axis

Comparison of analytical solution with Ansys solution:

Si.no	Result description	Analtical result	Result from Ansys	Variation
1	Patch width	1.502mm	1.541	0.039
2	Maximum Pressure	889.23	878.133	11.097
3	Stresses in X-axis	889.93MPa	878.133	11.797
4	Stresses in Y-axis	889.93MPa	878.133	11.797
5	Stresses in Z-axis	533.53MPa	522.137	11.393
6	Shear stress XY-axis	266.97	218.611	48.359
7	Approach distance	0.04884mm	0.044	0.004

Table 2. 1 Comparison of results of analytical solution with Ansys solution

PART-C

3.0 Introduction:

The Part-C is about the gear meshing problem which is to be solved with the knowledge obtained from the Part-B were Hertz contact stress was found between the rollers the problem can be considered as similar as the gear has involute profile and while gear meshing the contact takes place between the gear will be similar to the contact takes place between the cylinder. In Part-C by finite element analysis the gear profile is to be meshed and loads are to be applied and the results are to be verified.

3.1 The scheme used for creating the model:

The IGES gear file was provided for solving the Part-C assignment. The file is first imported to the Hyper mesh and meshing is carried by taking only two gear is in contact one profile of driver and one profile of driven as the result obtained for the one teeth will be same for all while the meshing, boundary condition and loads are same. To reduce the computing time this process is carried out after completing the meshing in hyper mesh the file is exported as .cdb which is the input for Ansys file.

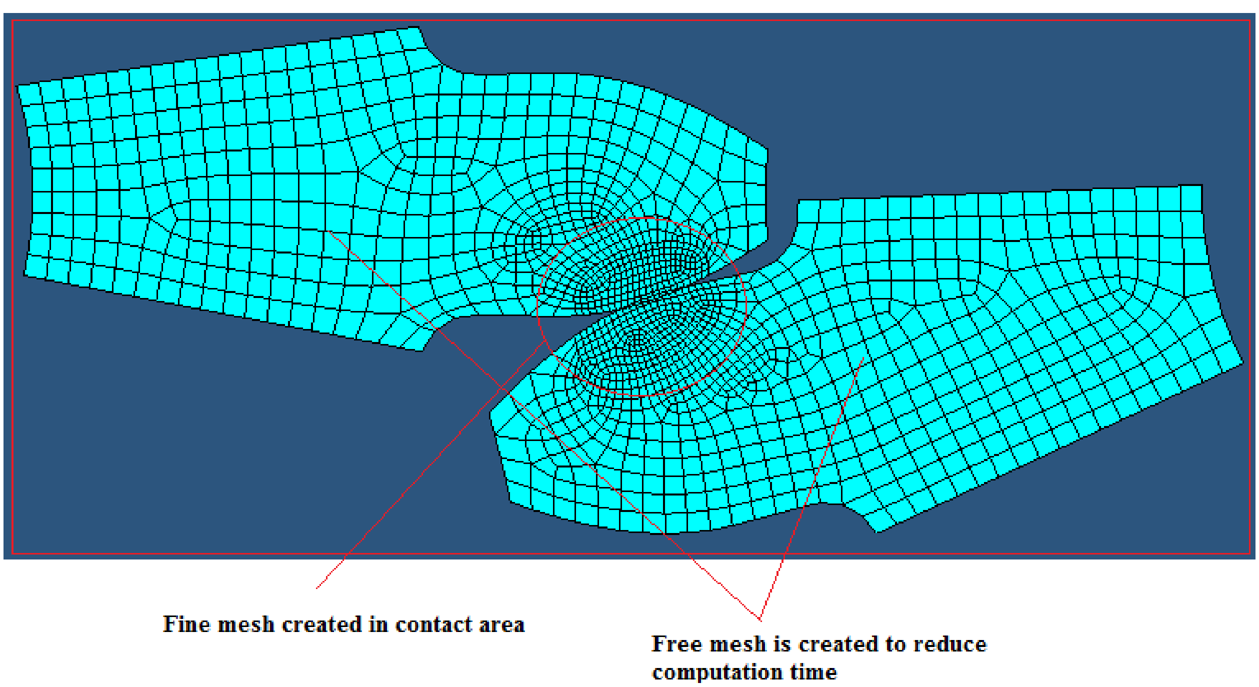


Fig 3. 1 Mesh created in hyper mesh

3.2 Element sizes:

In hyper mesh following mesh size of 0.09 4 node quad elements were used around the boundary created surrounding of the contact and the coarse mesh is created in the rest of the portions in the gear.

3.2.1 Element type:

Plane 82 quad element is used as it has 8 degrees of freedom and each element has 8 nodes and the result variation will be plotted as linear better result will be achieved. The element provides good result around the curved portion.

3.2.2 Contact175:

This element is used to represent the contact and sliding between the two surfaces between node and surface.

3.2.3 Targe169:

This element is used to represent target surface for contact elements describing the boundary of deformable body which are in contact with target surface.

NOTE: The elements used are as same used for Part-B as the results obtained from the gear problem are verified with the Part-B.

3.3 Loading and boundary conditions:

Finding the tangential load substituting the given data:

Given data:

i) Pitch Circle Diameter : 98.34mm
ii) Angle (ϴ) : 21.75^0
iii) Torque (T) : 987 N-mm

Tangential stress:

Tangential Load $= \dfrac{2\ x\ T}{P.C.D} = \dfrac{2\ x\ 987}{98.34}$

Tangential Load = 20.07 N

Normal stress:

Normal Load $= \dfrac{F_t}{\cos\theta} = \dfrac{20.07}{\cos 21.75}$

= 21.61 N

3.3.1 Loading and boundary condition for single gear:

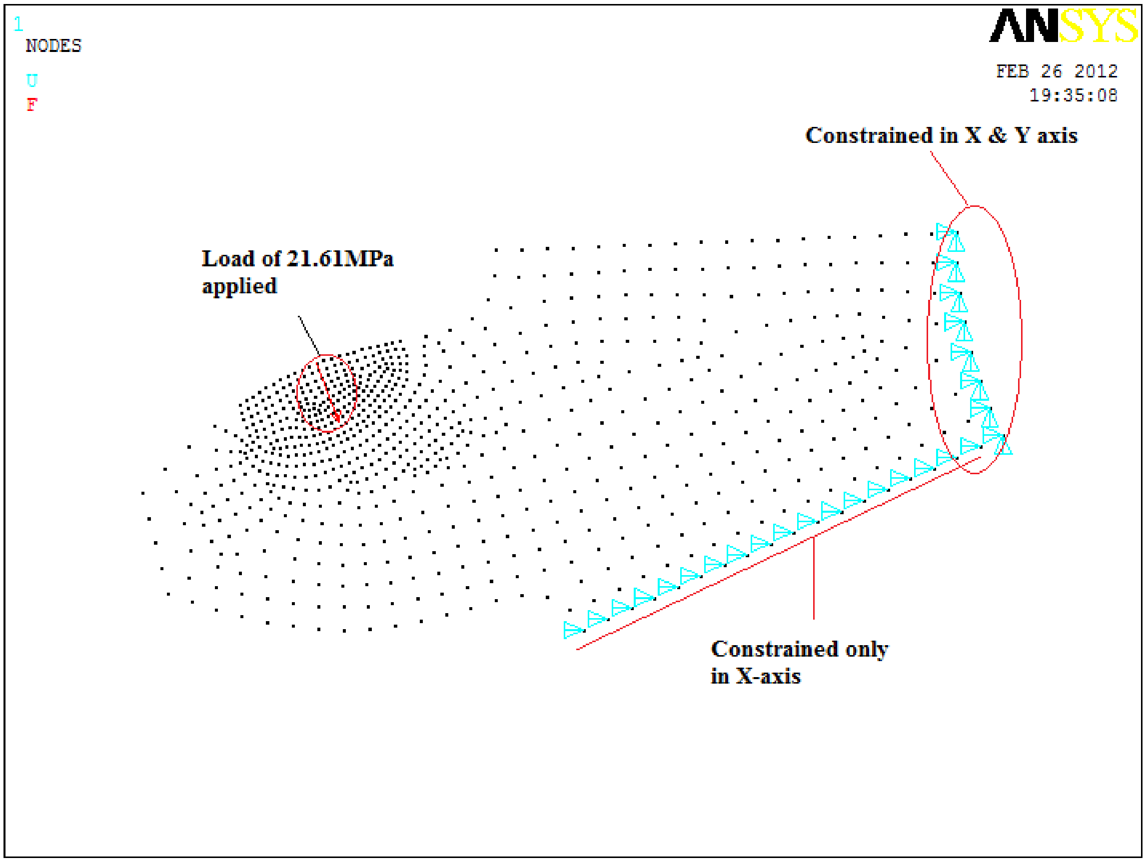


Fig 3. 2 Boundary condition for single gear

The boundary conditions applied to the single gear analysis are the XY axis is fully constrained as the gear should not move and in another condition the nodes of X-axis is alone constrained as when the load is applied in Y-axis it is free to move in y-axis. The compressive load of 21.61MPa is applied in Y- axis on the mid node of the gear as shown in the Fig 3.2.

3.3.2 Loading and boundary condition for gear in contact:

The loading and boundary condition of gears in contact are shown were in upper driver gear were load is applied nodes are constrained only in x-axis as the gear has to move inside the bottom gear when load is applied. In the driven bottom gear both X & Y is constrained as the gear should not move when load is applied. In order to avoid the movement along the X-axis is constrained.

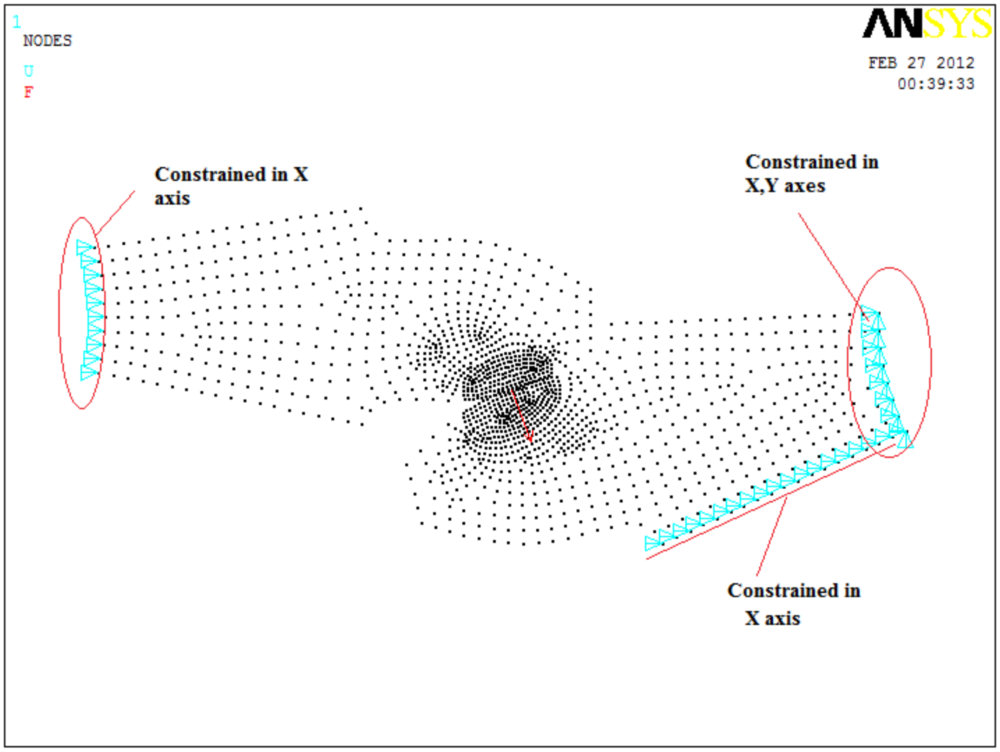


Fig 3. 3 Boundary condition for gear in contact

3.3.3 Stress for single gear in X-axis:

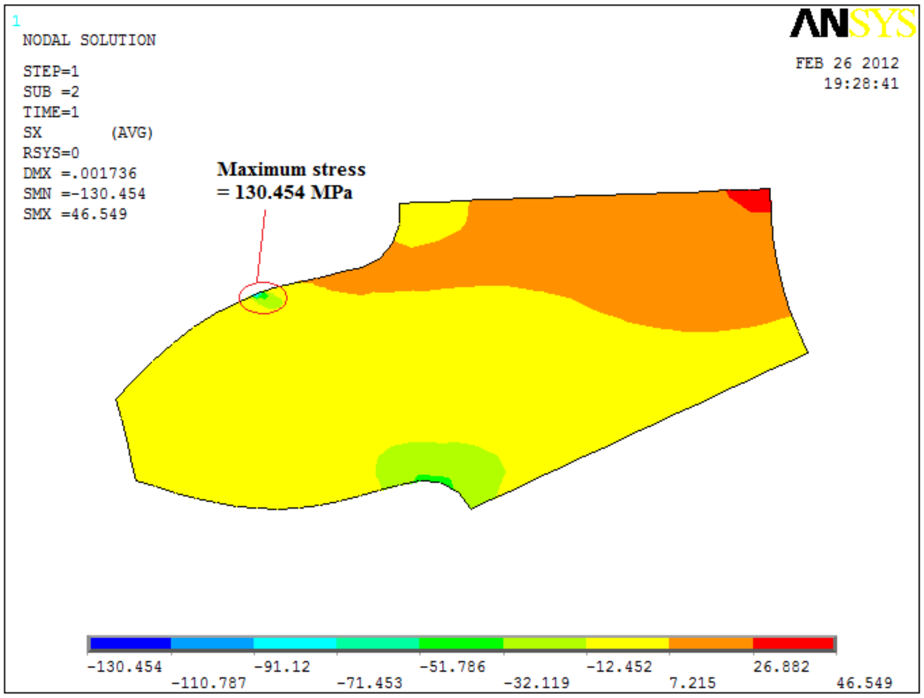


Fig 3. 4 Stress in X-axis for single gear

For the boundary conditions for single gear tooth when load is applied the stress in X-axis of 130.454MPa is obtained which is maximum at the point of load applied.

3.3.4 The contact stress for gear tooth in contact at X-axis:

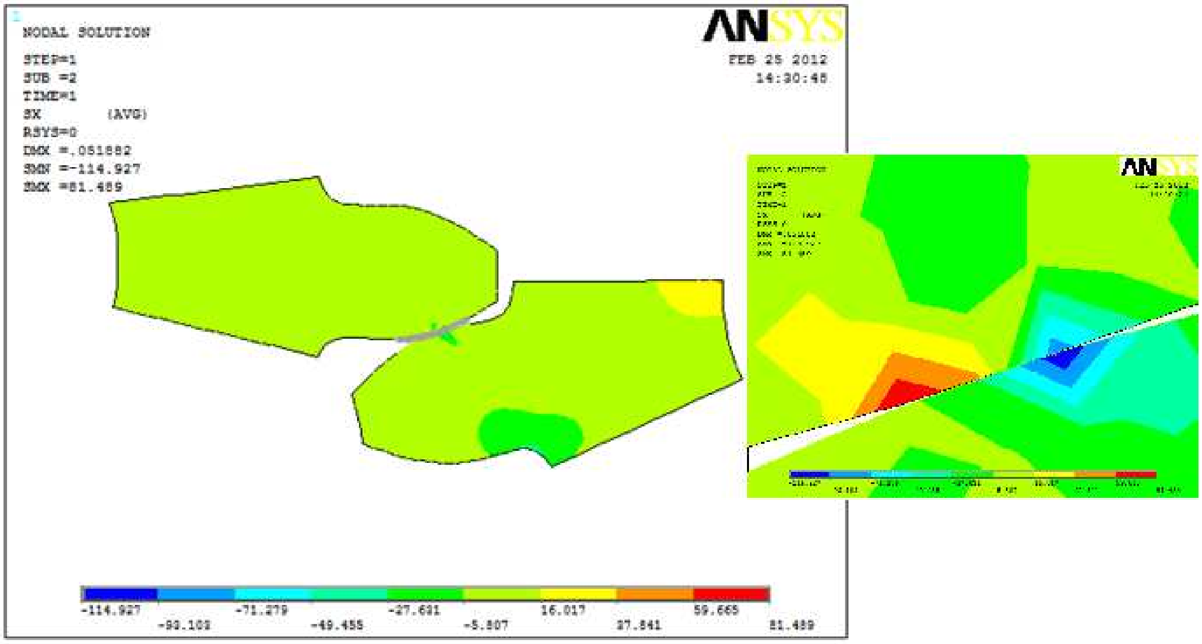


Fig 3. 5 Stress in X-axis for contact gear

For the boundary conditions for contact gear tooth when load is applied the stress in X-axis of 114.927MPa is obtained which is maximum at the point of load applied.

The result obtained for single gear is 130.454MPa and for contact gear is 114.927MPa. The variation in result is 15.527MPa.

3.3.5 Stress for single gear in Y-axis:

For the above boundary conditions for single gear tooth when load is applied the stress in y-axis of 245.454MPa is obtained which is maximum at the point of load applied as shown in the Fig-3.6

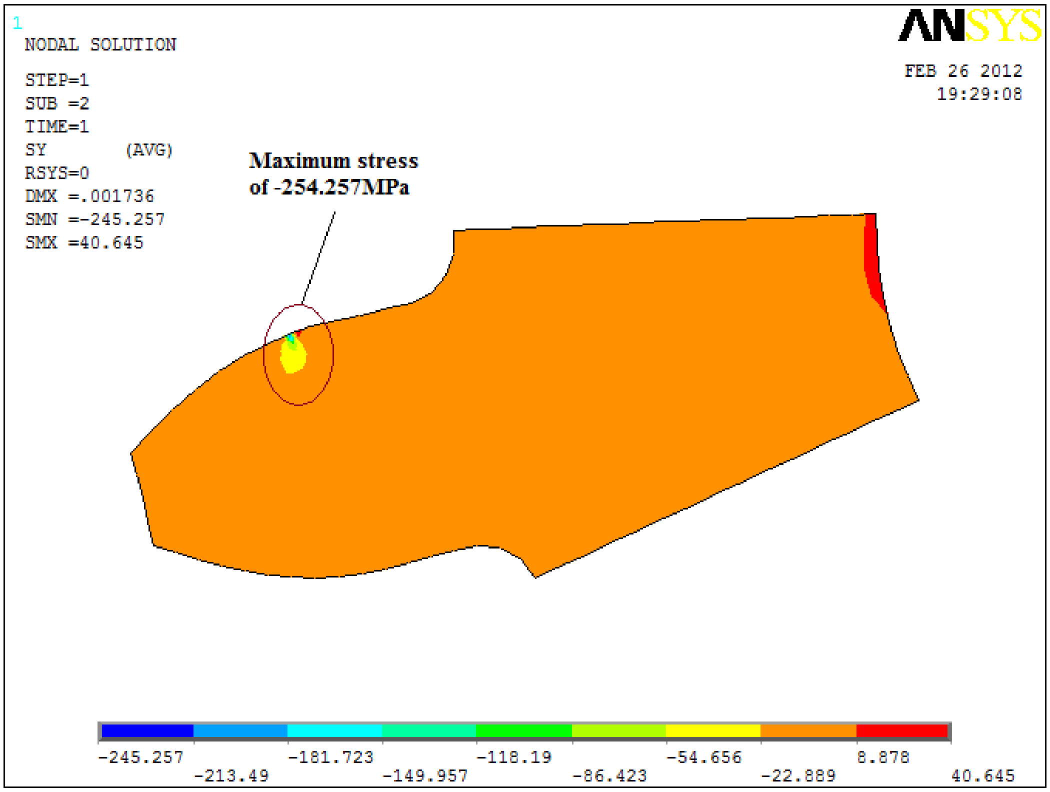


Fig 3. 6 Stress in Y-axis for single gear

3.3.6 The contact stress for gear tooth in contact at Y-axis:

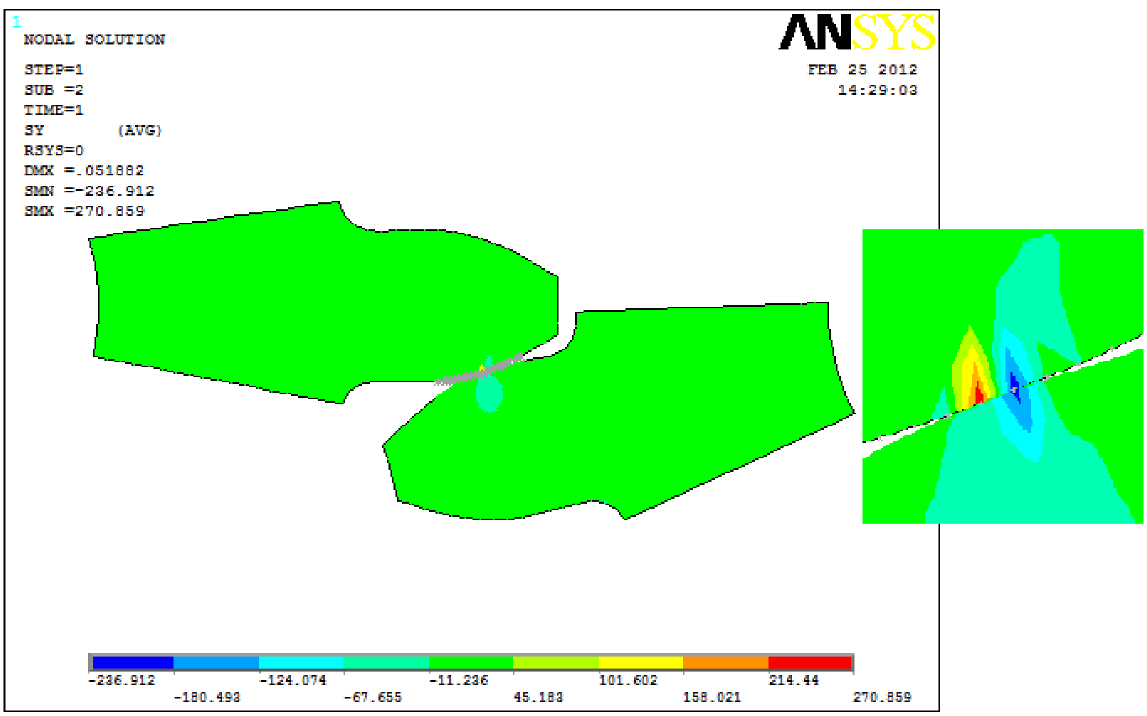


Fig 3. 7 Stress in Y-axis for contact gear

For the boundary conditions for contact gear tooth when load is applied the stress in Y-axis of 270.879MPa is obtained which is maximum at the point of load applied.

The result obtained for single gear is 254.257MPa and for contact gear is 270.879MPa. The variation in result is 16.602MPa

3.3.7 Shear stress for single gear in XY-axis:

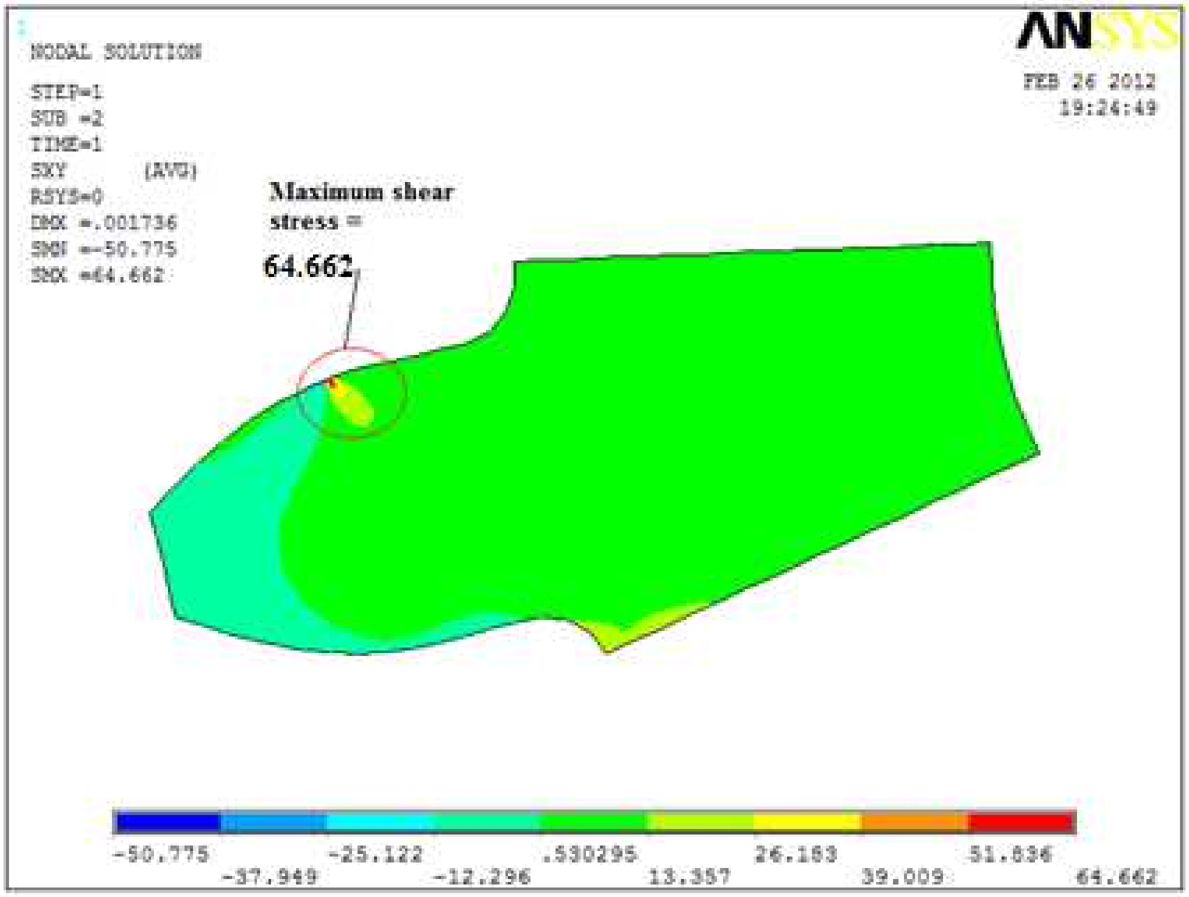


Fig 3. 8 Shear stress for single gear

For the above boundary conditions for single gear tooth when load is applied the shear stress in XY-axis of 50.775MPa is obtained which is maximum at the point of load applied

3.3.8 The shear stress for gear tooth in contact at XY-axis:

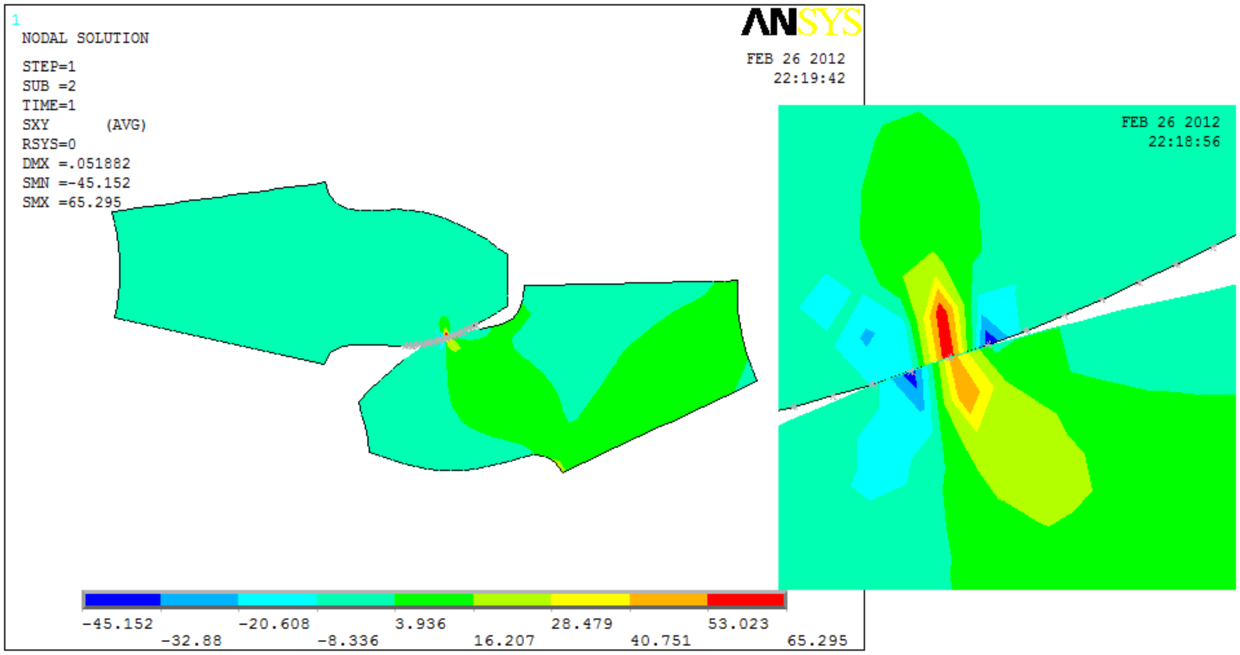


Fig 3. 9 Shear stress of contact gears

For the boundary conditions for contact gear tooth when load is applied the shear stress in XY-axis of 66.662MPa is obtained which is maximum at the point of load applied.

The result obtained for single gear is 66.662MPa and for contact gear is 65.296MPa. The variation in result is 0.633MPa

3.3.9 Root stress for single gear:

While bending the outer layer experiences the compressive stresses and inner layer experiences the tensile stress as shown in the Fig 3.10.

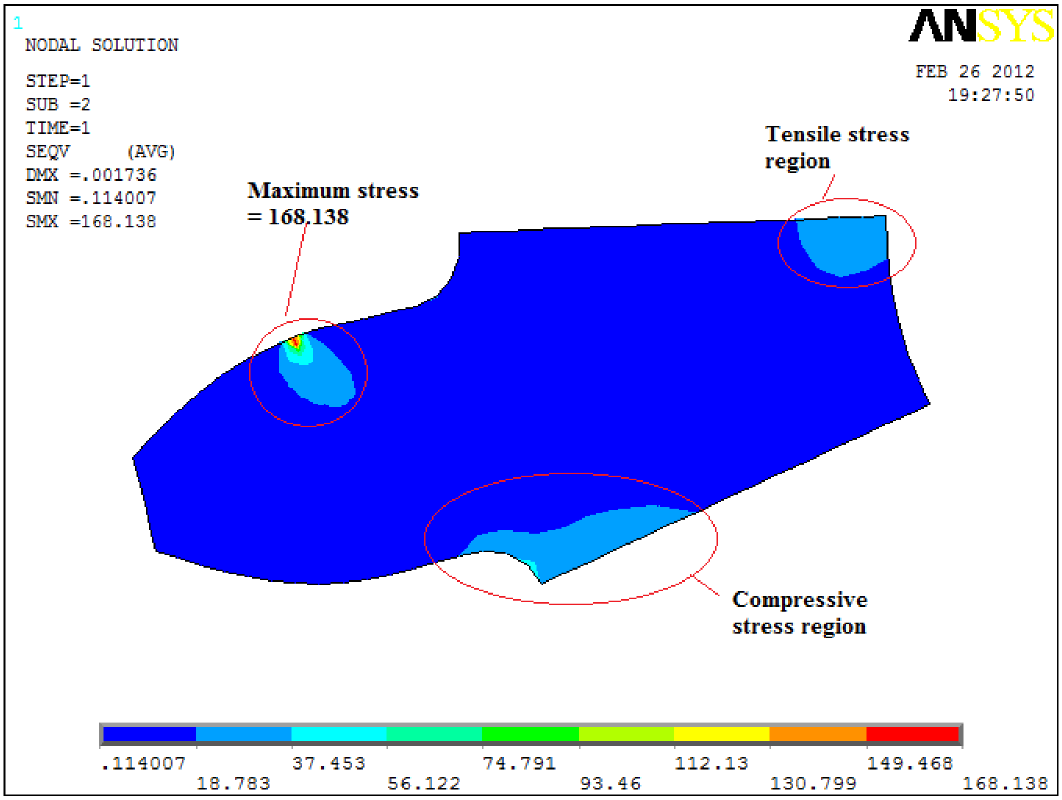


Fig 3.10 Root stress for single gear

3.3.10 Root stress:

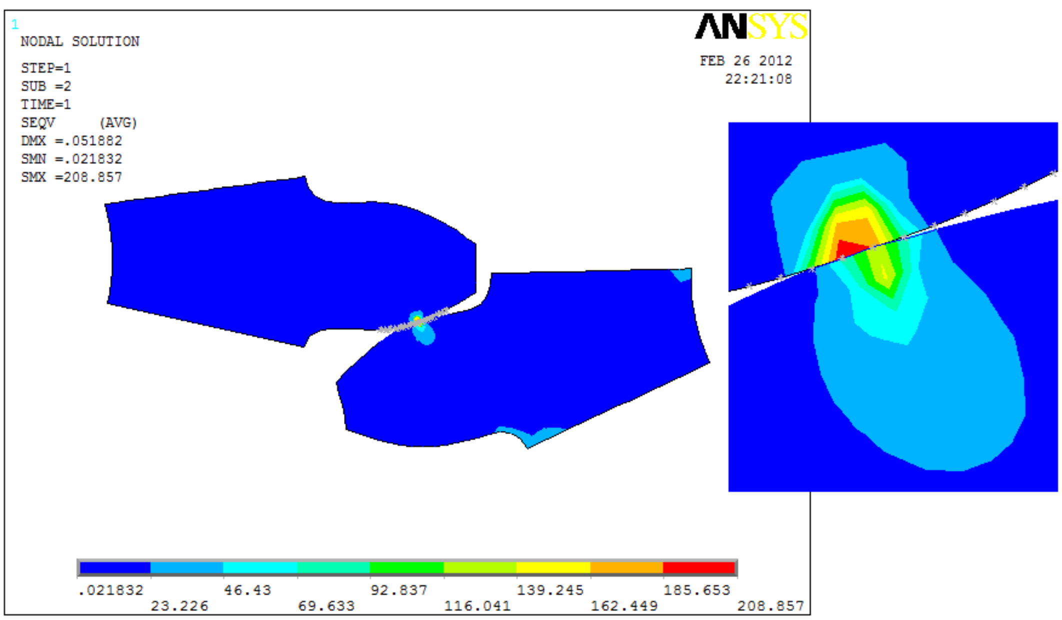


Fig 3.11 Root stress for contact gears

For the boundary conditions applied to contact gear tooth when load is applied the root stress are obtained from von-mices condition in the figure the load applied side experiences the tension and the opposite side experiences the compression the value of maximum stress is 208.857MPa is obtained which is maximum at the point of load applied.

3.4 Comparison of result of single and contact pair gears:

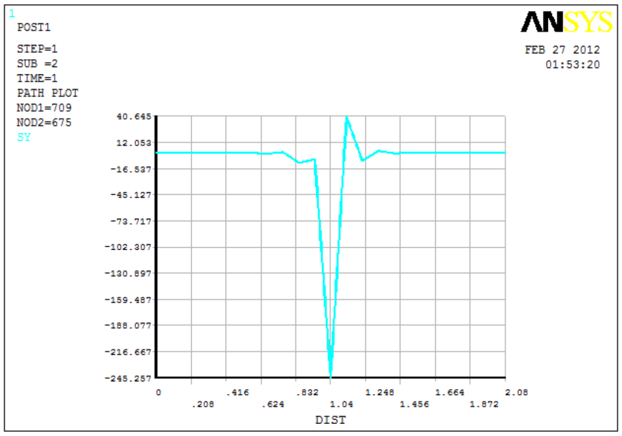


Fig 3.12 Stress developed in single gear

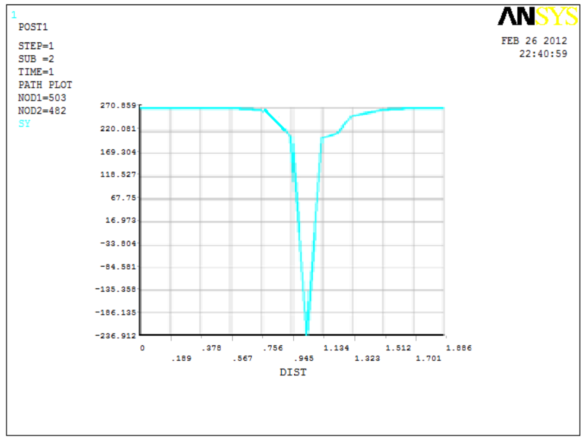


Fig 3.13 Stress developed in single gear

Si. no	Result description	Result of single gear teeth	Result of contact mesh gear teeth	Difference in value
1	Stress in X-axis	130.454	114.927	15.527
2	Stress in Y-axis	254.257	270.859	16.602
3	Shear Stress in XY-axis	64.662	65.295	0.633

Table 3. 1 Comparison of results of single gear and contact gear

The Fig 3.12 shows the stress distribution in Y-axis in single gear and Fig 3.13 shows the stress distribution in meshing gear the pattern of the stress distribution in the both the cases remains alike and the Table 3.1 shows the comparison of results obtained using the single gear analysis and meshing gear analysis were the error percentage is within the accepted limit.

3.5 Validation of result using benchmark:

The validation of the results are based on two methods one is of Hertz contact theory between the rollers and the root stress are validated using the modified photo elastic model which is the modified Lewis theory.

3.5.1 Justifications for Hertz contact theory validation:

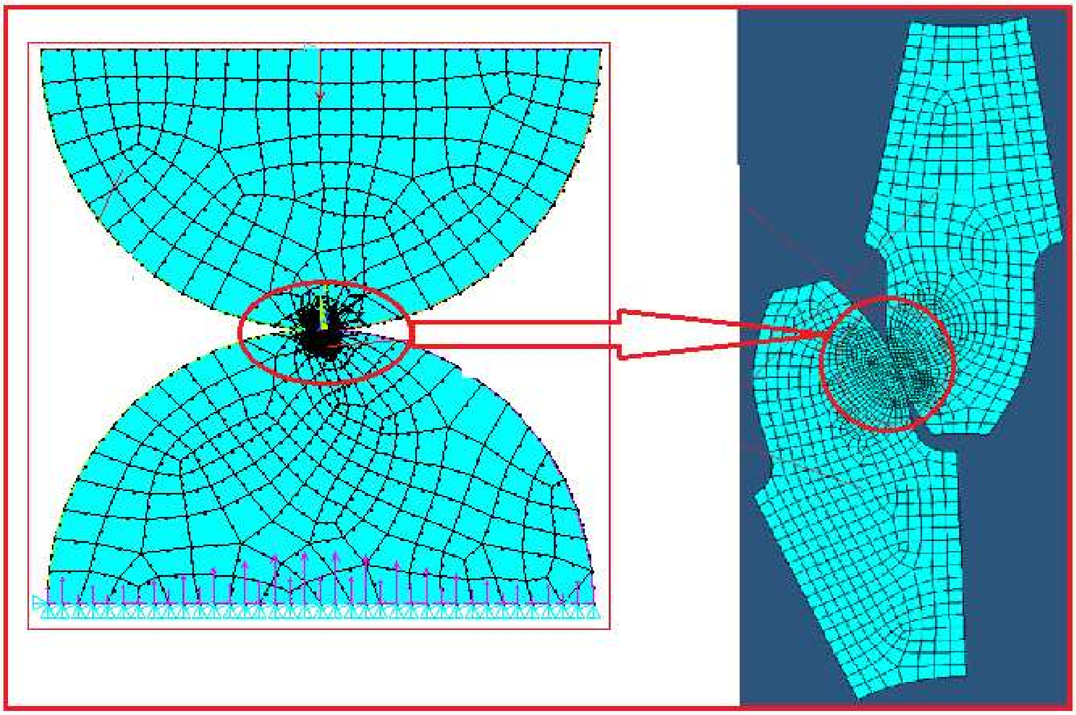


Fig 3. 14 Relation with Hertz contact and gear contact

As the gear tooth has involute profile while the gears are meshing the contact exist between the gears are similar to the contact exist between the cylinders. The Hertz contact theory is taken as the bench mark modal as per the Hertz contact theory the stresses induced at the contacts will have elliptical distribution along the contact zone.

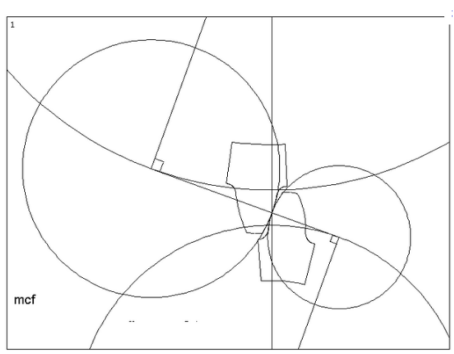


Fig 3. 15 Contacts of gears

Validating the result obtained from the Part-B Hertz contact with the contact obtained in between the meshing gear are shown below in the Fig 3.16. The stresses are maximum at the contact point and gradually decreasing at the distance moves away from the contact point. Which is similar in both the cases shown in the below figure. The contact patch width is obtained at the top of the graph were gradual diverging takes place in the graph.

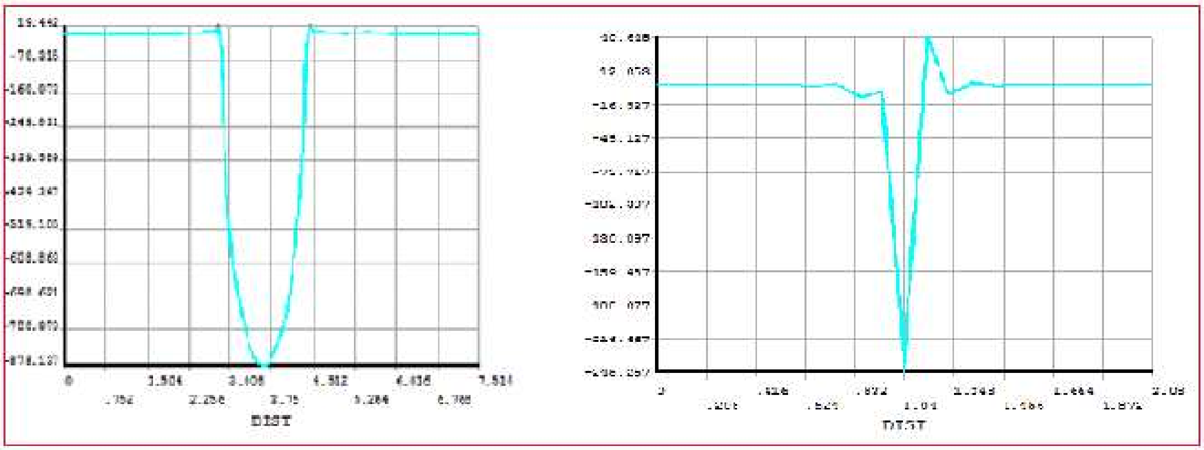


Fig 3. 16 Stress distribution in Hertz contact and gear contact

2.5.2 Comparing the stresses on the teeth:

As per the Hertz theory the stresses induced at the contacts will have elliptical distribution along the contact zone. Which is shown in the Fig 3.17 were the stresses are distributed in the elliptical manner.

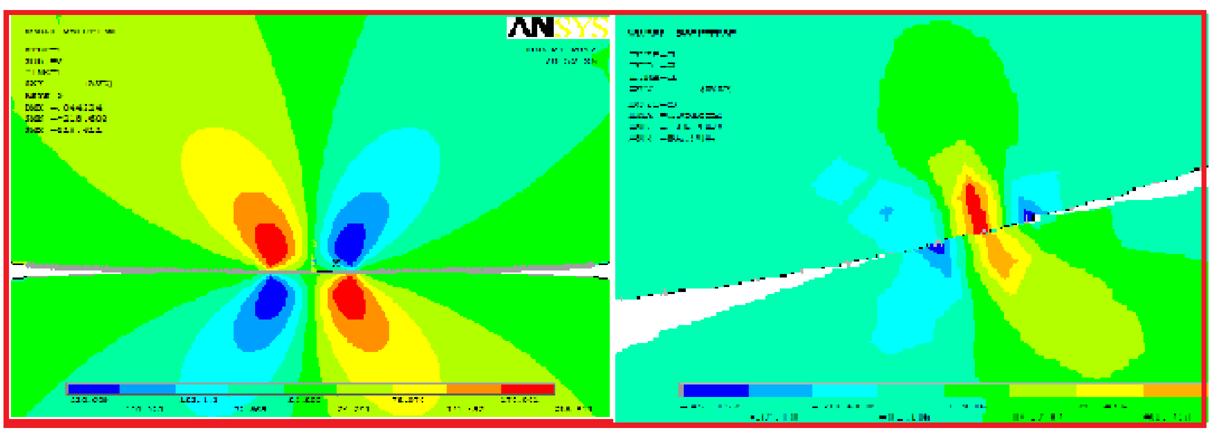


Fig 3. 17 Elliptical patch obtained in Hertz contact and gear contact

3.6 Root stress bench mark solution:

The photo elastic modal of gear theory is taken as the bench mark modal of the gears which was proposed by Dolan and Broghammer based on which the loading condition is done to obtain the results in single gear analysis. As there is a draw back in Lewis equation " *The greatest force exerted at the tip of the tooth is not true as the load is shared by teeth. It is exerted much below the tip when single pair contact occurs"* to overcome this draw back AGMA (American Gear Manufacturing Association) came up with modified Lewis theory based on this load is applied at the center and the result is obtained in the bench mark modal.

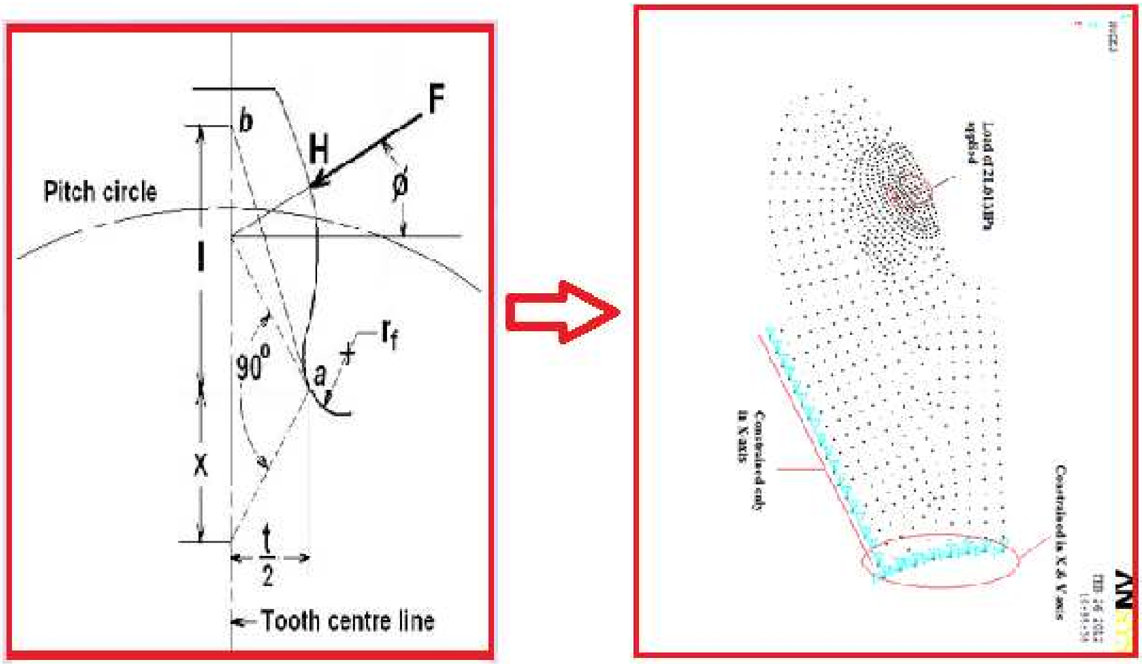


Fig 3. 18 Photo elastic theory idealization for single gear

4.0 Learning outcome:

The FEA module gives the idea of importance of analysis and the approach for solving the problem like idealization creating the mathematical model etc, the concept of discretisation and how it simplifies the problem was learned from the module. The various concepts of mechanics of materials like plane stress, plane strain etc, and its application in application Ansys software was learned and the concept of element formulation and various shape functions of different elements and solving the vibration and dynamic problems and idealization for these problem was learned. Thermal analysis and the non linear problems and solution is obtained using the various iterative methods was learned. In the lab session Hypermesh and Ansys software was covered were in hypremesh the selection of convergence and the selection of elements which is theoretically learned is applied in software and validating the mesh was learned modeling and applying boundary conditions and approach to the solution in Ansys was learned. The lab session can be made still effective by creating the standard exercise problems for Hypermesh and Ansys which are to be practiced during our lab session and should be evaluated before the start of assignment which will make us to our assignment in still better way.

References

[1] Kurowski, Paul M. (2004) "*Finite Element Analysis for Design Engineers*", Society of Automotive Engineers, USA.

[2]Saeed moaveni.(2011) "*finite element analysis*", 3rd edision dorling Kindersley India pvt.ltd.

[3] Prof gopinath, Gear failure, http//www.nptel.iitm.ac.in./courses/iit-MADRAS/Machine _design …/2_6.pdf retrieved on 24/02 /2012.

[4] Zeping wei http://library.usask.ca/theses/available/etd-10272004-163322/unrestricted/ ThesisFinalVersion .pdf retrieved on 26/01/2012.

[5] Abdullah alsahly, Adaptive finite elements http://www.sd.rub.de/downloads/adaptive_finite elements .pdf Retrived on 26/02/2012.

[6] Unknown, on target elements http:// mostreal.sk/html/elem_55/chapter4/ES4-169.htm retrived on 14/02/2012.

Appendix -1 (Analytical solution for part-B)

Analtical solution
For Part - B

Given data:-

$R_1 = 94.06$ mm $\quad D_1 = 188.12$

$R_2 = 91.89$ mm $\quad D_2 = 183.78$

$F = 1049$ N

Finding the patch width:

$$b = \sqrt{\frac{2F}{\pi L} \times \frac{D_1 D_2}{D_1 + D_2} \cdot \left(\frac{1-\nu^2}{E_1} + \frac{1-\nu^2}{E}\right)^2}$$

$$= \sqrt{\frac{2 \times 1049}{\pi} \times \frac{188.12 \times 183.78}{(188.12 + 183.78)} \left(\frac{1-(0.3)^2}{2 \times 10^5} + \frac{1-(0.3)^2}{2 \times 10^5}\right.}$$

$$= \sqrt{667.81 \times \left(\frac{34572.69}{371.9}\right) \times 9.1 \times 10^{-6}}$$

$$= 0.751 \text{ mm}$$

$$\text{Patch width} = 2b = 2 \times 0.751 = 1.502 \text{ mm}$$

Maximum Pressure:-

$$P_{max} = \frac{2F}{\pi b L} = \frac{2 \times 1049}{\pi \times 0.751} = 889.23 \text{ MPa.}$$

The stress induced in the cylinder along the z-axi given by.

$$\sigma_z = -2 P_{max} \cdot \nu \left[\sqrt{1 + \left(\frac{y}{b}\right)^2} - y/b\right]$$

$$= -2 \times 889.23 \times 0.3$$

$$\sigma_z = -533.53 \text{ MPa.}$$

stresses induced in x-axis is given by :-

$$\sigma_x = -P_{max}\left[\sqrt{1+\left(\frac{y}{b}\right)^2}\times\left(2-\left(1+\frac{y}{b}\right)^2\right)^{-1}\right]$$

$$= -889.23(1) \text{ at } y=0$$

$$\sigma_x \simeq -889.93 \text{ MPa.}$$

stresses induced in y-axis is given by

$$\sigma_y = -P_{max}\left(1+\left(\frac{y}{b}\right)^2\right)^{-1/2}$$

$$\sigma_y \simeq -889.93 \text{ MPa}$$

The approch or center between the two cylinder is given by.

$$\Delta = \frac{2F(1-\nu^2)}{\pi E L}\left(\frac{2}{3}+\ln\frac{D_1}{b}+\frac{D_2}{b}\right)$$

$$= \frac{2\times 1049\,(1-(0.3)^2)}{\pi\times 2.1\times 10^5}\left(\frac{2}{3}+\ln\frac{188.12}{0.751}+\ln\frac{183.78}{0.751}\right)$$

$$= 3.03\times 10^{-3}(0.66+5.52+10.10)$$

$$\therefore \Delta = 0.04884 \text{ mm}$$

Shear stress is calculated by

$$\tau_{xy} = 0.3\times P_{max}$$

$$= 0.3\times 889.93$$

$$= 266.97 \text{ MPa}$$