Stress Analysis of Polyoxymethylene which Leads to Wear in Pin on Disc Configuration using Finite Element Method

Alen John[1], Dr. Y.V.K.S Rao[2], Jerin Sabu[3], Rajeev V. R[4]

P G Scholar[1], Department of Mechanical Engineering, Mar Baselios College of Engg & Technology, Trivandrum, India

Professor[2], Department of Mechanical Engineering, Mar Baselios College of Engg & Technology, Trivandrum, India

P G Scholars [3, 4], Department of Mechanical Engineering, Mar Baselios College of Engg & Technology, Trivandrum, India

Email: alenjohn1989@gmail.com[1]

Abstract-The wear is defined as the progressive loss of substances from the operating surface of a body occurring as a result of relative motion at the surface. The simulation of the stresses which leads to wear in pin on disc configuration having the pin made of polyoxymethylene having the NC010-150 designation in sliding contact with the rotating steel disc (AISI 304) was performed using finite element software ANSYS Workbench 11.0. The three dimensional geometry of the required configuration is made by using the modelling software CATIA. Using the boundary conditions the analysis are done. Rotation of 3000 rpm is provided for the disc. Then with analysis software we are analysing the wear by varying the pressure acted upto the tensile yield strength of the material and assuming that the region of the pin where maximum equivalent stress occurred is failing or wearing out, since we have no provision for finding out the wear analysis in the available software packages. Then we are removing the failed elements and again repeating the same process until the material of the pin fails completely, i.e. varying the length of the pin. Along with that the maximum deformation and the maximum equivalent stresses are noted. Then compare the results of deformation and the equivalent stress and concluded that when the length of the pin decreases (pin failed more) the deformation is decreasing and the equivalent stress is increasing. When the load exceeds tensile yield strength of the material then the material is failed under these circumstances.

Index Terms- wear, polyoxymethlene; pin on disc configuration; stress analysis; finite element method.

1. INTRODUCTION

Tribological failure is one of the most common problems that limit productivity and its reason is the wear. Wear is defined as the progressive loss of substances from the operating surface of a body occurring as a result of relative motion at the surface. The pin-on-disc configuration is commonly used for wear tests in laboratories because of its simple arrangement. It consists of a stationary pin under an applied load in contact with a rotating disc.

The pin can have any shape to simulate a specific contact, but spherical tips are often used to simplify the contact geometry. Coefficient of friction is determined by the ratio of the frictional force to the loading force on the pin. However, some phenomena which affect the test results. The wear rate of a stationary pin forced on a rotating disc is higher than that of a rotating pin sliding on a stationary disc when the normal load is over some level.

The simulation of wear is still in its early stages due to the complexity of consideration of additional phenomena occurring at the interface. And they are thermal effects, transfer of materials that are affected by not only the operating conditions but also contact geometry and materials properties [Benabdallah and Olender, (2006)]. Extensive research has been made in the development of a wear model that describes the wear process fully [Oqvist, (2001)]. Innovations in modern technology have pointed out the increase in the demand on advanced plastic materials. [Liu *et al.*, (2000)].

Polyoxymethlene (POM) is an engineering thermoplastic used in precision parts when high stiffness, low friction and excellent dimensional stability are required. POM is characterized by its high strength, hardness and rigidity to ~40 °C. POM can be used for Mechanical gears, sliding and guiding elements, housing parts, springs, chains. It is also used in electrical engineering applications like insulators, bobbins, connectors. For modelling railway parts and in the medical, furniture and construction field also this material is applicable.

The primary aim of this work was to simulate the pin on disc experimental setup for the case of an engineering polymer in the form of a flat-ended pin in contact with a metallic spinning disc. The analysis is expected to be completed in the finite element analysis software ANSYS Workbench 11.0 with the help of the model made by using CATIA V5R18. The main focus of this work is to study how the pin failure occurs when pressure is increased and to check which elements are failed out. The stress contours and the total deformation are also found out.

2. SOLID MODELLING

The solid modelling required for this work is done by using the software CATIA V5R18 and its isometric view is given in fig.1.

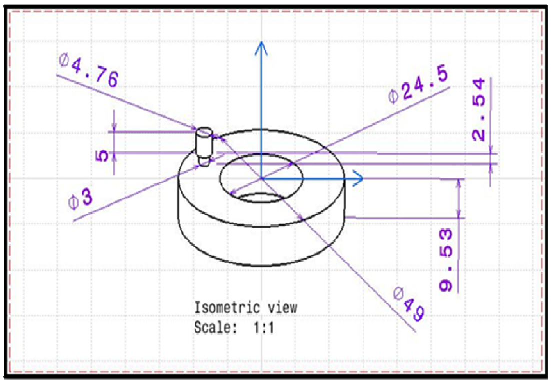

Fig. 1. Isometric view of the model

First each parts are made separately in the modelling software with the help of main menu. From main menu we go to 'mechanical design' and then to 'part design' option. Then select the required plane. For modelling the first part called hollow disc two circles are drawn by using the 'sketch' option. Then use 'exit workbench' command for exit. Use the 'pad' option for making these circles into the disc. The same procedure are followed for making the remaining portions of the model the differences are in the case of dimensions only. For completing this work five pins are modelled by decreasing its dimension to 0.54 mm in the second case, whereas from the third case to fifth case we reduce it to 0.50 mm.

For assemble these parts, main menu of the software is used and then to 'mechanical design' and then to 'assembly design'. Then select the 'existing component' option and click on the 'product' label. Then the required parts of the assembly are opened in the order from hollow disc, pin holder and pin. Using the 'manipulating' option, change the plane of disc. Arrange the pin into the center portion of the hollow disc as shown in the isometric view in the fig.1, with the help of 'offset' option. Then use the 'contact constraint' option and then select the surfaces to make them in contact. Again select the 'existing component' option and click on the 'product' label and select the holder. Arrange the holder into the top portion of the pin as shown in the isometric view with the help of 'offset' option. Then use the 'contact constraint' option and then select the surfaces to make them in contact. Then the required assembly is obtained as shown in the fig.2.

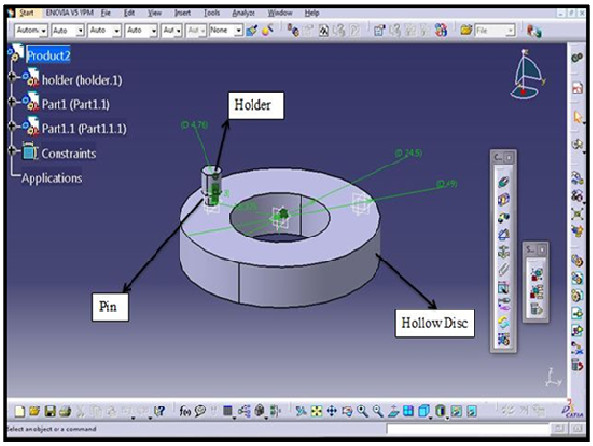

Fig:2. -3D view of the assembly

3. FINITE ELEMENT PROCEDURE

A computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects is called finite element analysis. The engineering simulation software developed by United States called ANSYS Workbench platform which is subsidiary part of ANSYS 11.0 is used here.

The model made by using the CATIA software is converted into 'igs' then it is opened in the ANSYS workbench. Open 'empty project' and then click on the 'Link to geometry file' and then browse, select the required model in 'igs' format. Click on 'new simulation' option. The required material properties are added into the software by using the tribo material properties provided in the Table 1.

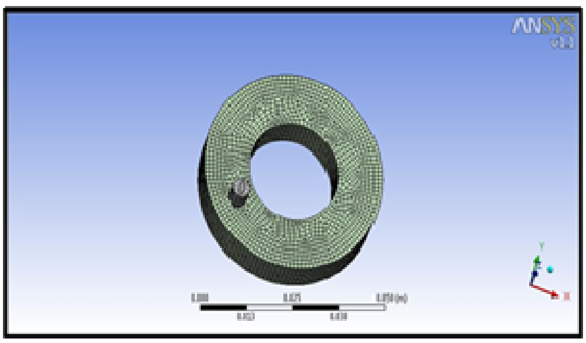

Fig: 3 -Meshed view of the model

Giving the required connections between the parts of the model is the next step. In between the holder and pin bonded connections is provided and given in

Table. 1: Tribo-materials properties

Material	Geometry	Elastic Modulus (GPa)	Tensile Yield Strength (MPa)	Poisson's Ratio	Density (kg/m^3)
POM	Pin	3.1	72	0.25	1460
Stainless steel	Disc	210	1109	0.3	7930
Structural steel	Holder	200	250	0.3	7850

between the top face of the pin and bottom face of the holder. Then the connection between the pin and disc is given as frictional contact, value of co-efficient of friction as 0.3 and it is provided in between the top face of the disc and bottom face of the pin.

Meshing is the next step, fine mesh is provided. Giving default element size resulting in 45435 nodes and 9754 elements to the entire geometries. Elements selected are solid185, conta174, targe170 & surf154. The meshed view of the model is given in the Fig: 3

3.1. *Boundary conditions and loads*

Here the displacement in the x and y direction of the outer round faces of the pin and holder is arrested, so given the value as zero. At the same time, z direction displacement given as free, for allowing the movement of pin and holder only in the z direction. Then displacement in the x and y direction of the outer round faces of the hollow disc made free At the same time, z direction displacement is arrested, so given the value as zero, for allowing the movement of hollow disc in the x and y direction.

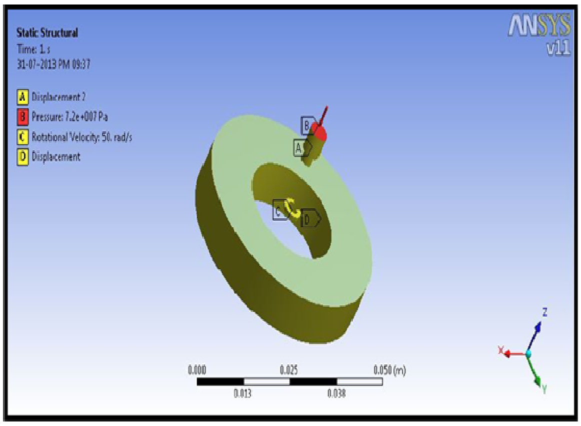

Fig: 4 -Boundary condition and load of the model

The pressure is applied in the top face of the pin holder, its value is varying from 10 MPa to 70 MPa since the tensile yield strength of the POM is 72 MPa. Along with that a rotational velocity of 50 rad/sec (3000rpm) is provided for the hollow disc in the z direction, for the rotation of disc and this is clearly given in Fig.4

3.2. *Analysis*

When the pressure is applied in the top face of the pin holder, then the equivalent (von-Mises) stress is finding out, then the maximum and the minimum value is noted. The portion where maximum and minimum value obtained is also noted. After this, the total deformation stress is finding out, then the maximum and the minimum value is noted. The portion where maximum and minimum value obtained is also noted.

It is repeated for the same model from 10 MPa to 70 MPa in steps of 10 MPa, it results in 7 analysis. When the initial setup is analysed, we got from the analysis that some portion of the pin showing the maximum equivalent stress. We assume that portion is worn out. Then removing that elements. We have no provision for measuring the length of the failed elements of pin by this software. So it is necessary to make 5 models, then all these results in 35 experiments.

4. RESULTS AND DISCUSSIONS

Using the ANSYS Workbench software, along with the boundary conditions, 10 MPa pressure is given then we got a value for this pressure, the values equivalent (von-Mises) stress and deformation. Von Mises stress is widely used by designers, to check whether their design will withstand given load condition and it arises from distortion energy failure theory[Alen John and Nidhi,(2014)].Then the maximum and the minimum value is noted and the portion where it's value is obtained, i.e. some portion of the pin showing the maximum equivalent stress. We assume that portion is worn out.

Then the stress contour & deformation obtained at length of pen (LOP)-2.54mm, 70MPa, and 50 rad/sec is given in the figures, fig. 5(a), 5(b) and 5(c). At the tip of the pin is having the maximum stress value and the maximum deformation is obtained at the portion where the pin holder and the pin is bonded and which is especially in the face of the pin where the pin holder and the pin is bonded. At the maximum pressure of 70 MPa, the value of maximum equivalent stress as 171.89MPa and maximum deformation as 35.45μm.

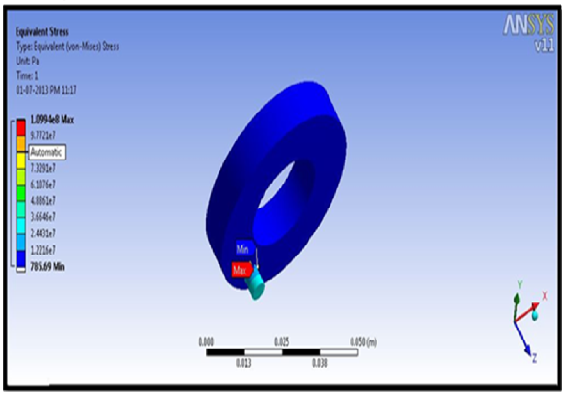

Fig: 5(a) - Equivalent stress at LOP-2.54mm, 70MPa, 50 rad/sec

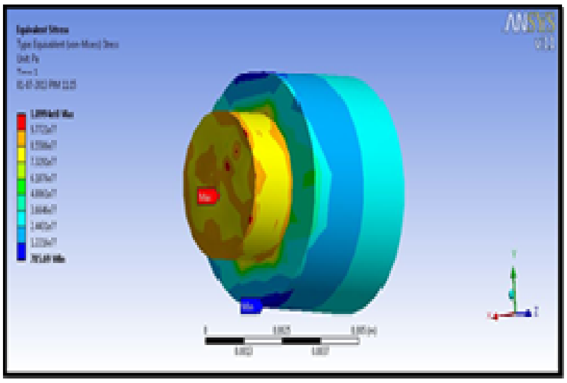

Fig: 5(b) – Deformation at LOP-2.54mm, 70MPa, 50 rad/sec

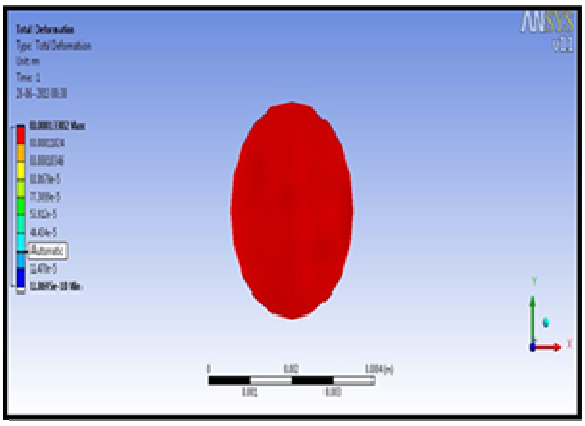

Fig: 5(c) – Deformation obtained for pin on top face at LOP-2.54mm, 70MPa, 50 rad/sec

Along with the boundary conditions, 10 MPa pressure is given then we got a value for this pressure, the values are noted. Assumed that 0.5 mm of the length of the pin is worn out, and the model used here is having the length of the pin as 2.00mm. The value means the equivalent (von-Mises) stress and deformation. Then the maximum and the minimum value is noted and the portion where it's value is obtained, like that 1.50mm,1.00mm and 0.5mm length was given as the length of the pin and the values of these 35 analyses are given the table.2

While considering that, about 2.00 mm of the length of the pin is worn out, and the model used here is having the length of the pin as 0.50 mm. The values means the equivalent (von-Mises) stress and deformation. Then the maximum and the minimum

Table. 2: Maximum equivalent stresses and deformations obtained for the model when length of pin and pressure varied from 2.54mm to 0.50mm and 10 to 70 MPa respectively at 50 rad/sec

LOP(mm)	**Properties**	**Pressure(MPa)**						
		10	**20**	**30**	**40**	**50**	**60**	**70**
2.54	**Max Equivalent stress (MPa)**	24.67	49.18	73.69	98.22	122.76	147.32	171.89
	Max Deformation (μm)	4.98	9.95	14.76	19.82	24.72	29.61	35.45
2	**Max Equivalent stress (MPa)**	21.85	43.69	65.53	87.37	109.21	131.04	152.86
	Max Deformation (μm)	8.03	16.03	23.99	31.91	39.69	47.64	56.34
1.5	**Max Equivalent stress (MPa)**	20.17	38.92	56.02	77.16	95.24	114.25	143.68
	Max Deformation (μm)	12.12	23.06	34.01	45.09	56.91	65.61	75.06
1	**Max Equivalent stress (MPa)**	18.23	36.87	55.31	73.73	92.16	110.6	129.03
	Max Deformation (μm)	14.92	29.84	44.76	59.68	74.62	89.52	104.4
0.5	**Max Equivalent stress (MPa)**	16.65	34.89	42.19	60.76	78.11	96.22	109.44
	Max Deformation (μm)	18.095	36.19	54.28	72.38	95.01	114.02	133.02

value is noted and the portion where it's value is obtained, i.e. some portion of the pin showing the maximum equivalent stress.

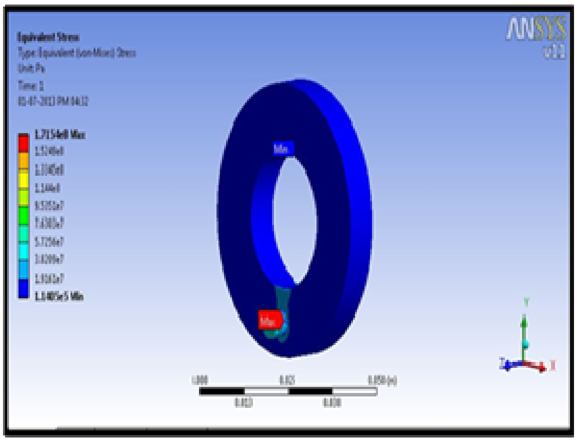

Fig: 6(a) - Equivalent stress at LOP-0.50mm, 70MPa, 50 rad/sec

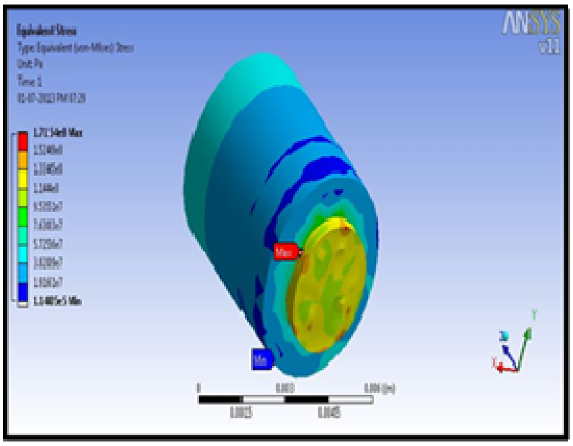

Fig: 6(b) – Deformation at LOP--0.50mm, 70MPa, 50 rad/sec

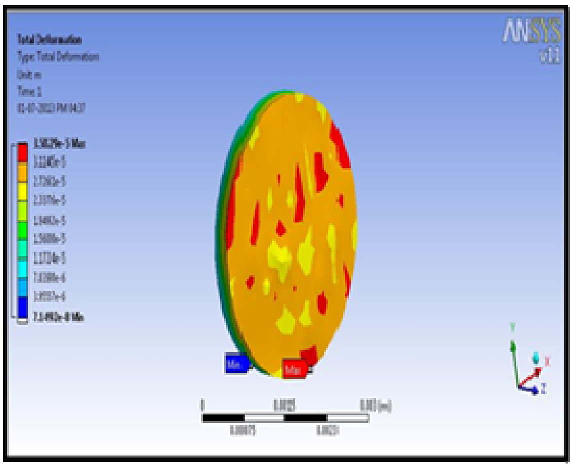

Fig: 6(c) – Deformation obtained for pin on top face at LOP--0.50mm, 70MPa, 50 rad/sec

Then the stress contour stress & deformation obtained at LOP-0.50m, 70MPa, and 50 rad/sec are given in the figures, fig. 6(a), 6(b) and 6(c). Here also the tip of the pin is having the maximum stress value and the maximum deformation is obtained at the portion where the pin holder and the pin is bonded and especially in the face of the pin where pin holder and the pin is bonded as earlier. At maximum pressure of 70 MPa, we got the value of maximum equivalent stress as 109.44 MPa and maximum deformation as 133.02 µm. At this stage, we are considering that the entire length of the pin is worn out after this consideration.

5. CONCLUSION

The experimental setups are modelled in CATIA which are very essential for these analyses. It is successfully imported into ANSYS Workbench. When the pressure increases from a value of 10 MPa to 70 MPa, the corresponding maximum equivalent stresses are increased, at the same time some portion of the bottom face of the pin results the maximum value. Found out that some elements of the pin will fail first, since the material used for making this component have low tensile yield strength.

When the pressure increases from a value of 10 MPa to 70 MPa, the corresponding maximum deformations are increased, at the same time some portion of the top face of the pin results the maximum value. When the pin starts wearing out, then the maximum value of the equivalent stresses are increased, it may due to the decrease in area. When the pin starts wearing out, then the maximum value of the deformation are decreased, it may due to the decrease in length.

In the future he same work by changing the material of the components. The changing may be of two types. One is considering the pin and disc having the same material and the other one is the work by using different combinations of the materials used and the modifications of their geometry also. Various materials which can be used for the tribological purposes can also be analysed with these type of analyses.

Acknowledgments

The authors would like to thank GOD ALMIGHTY for His unseen hand, which guided and strengthened me even at times when we felt we were not capable of doing it ourselves. Finally, we express our sincere thanks to our parents, teachers especially to the guide Dr. Y.V.K.S Rao, Professor, Dept. of Mechanical Engineering, Mar Baselios College of Engineering & Technology, then to our friends and well-wishers for their whole hearted support and to all those who have helped us directly or indirectly for doing this work and then to transformed it into a journal paper.

REFERENCES

[1] Alen John; Nidhi M.B. (2014): *Modelling and finite element analysis of automotive bumper under static loading,* National conference on advances in mechanical engineering-**2014,** pp. 09-13.
[2] Bellow D.G; Viswanath N.S. (1993): *An analysis of the wear of polymers,* Wear **162**, pp. 1048–1053.
[3] Benabdallah H; Olender D. (2006): *Finite element simulation of the wear of polyoxymethylene in pin-on-disc configuration,* Wear ,**261**, pp.1213–1224.
[4] Chen Q; Li D.Y; (2003*): Computer simulation of solid particle erosion*, Wear, 254, pp.203–210.
[5] Franklin F.J; Widiyarta L; Kapoor A. (2001): *Computer simulation of wear and rolling contact fatigue,* Wear, 251, pp. 949–955.
[6] Guicciardi S; Melandri C; Lucchini F; De Portu, G. (2002*): On data dispersion in pin-on-disk wear tests,* Wear, 252, pp.1001–1006.
[7] Hegatekatte V; Huber N; Kraft O.(2005): *Finite element based simulation of dry sliding wear,* Modelling & Simulation in material science and engineering, 13 ,pp. 1357-75.
[8] McColl I.R.; Ding J; Leen S.B. (2004*): Finite element simulation and experimental validation of fretting wear*, Wear, 256, pp.1114–1127.