International Journal of Engineering Technology and Applied Science

(ISSN: 2395 3853), Vol. 3 Issue 5May 2017

Finite Element Analysis of Hip Joint under Normal Contact Pressure

[1]Satish Prajapati, [2]Dr. S. S. Chouhan, [3]Ranjeet Kumar
[1]M-Tech Scholar, [2]Director, [3]Professor
[123]Department of Mechanical Engineering, VITS, Bhopal, India

Abstract- In order to analyze the Hip joint it is very important to analyses the different Stress, Strain and deformation on the joint. Here in this paper the following objective can be achieved. Develop the Finite element model of the hip joint.Analyzing the deformation on the hip joint at different frequency during the normal load condition. Calculate the different Stress on the joint.Study and Analysis of Three-Dimensional ball and socket hip Joint under normal contact pressure and different frequencies and results have been observed and optimize it. Also optimizing the different stress conditions.

KEYWORDS: Hip Joint, FEA, FEM, Contact Normal Pressure, Stress.

I Introduction

The ball and socket joint (or spheroidal joint) is a type of synovial joint in which the ball-shaped surface of one rounded bone fits into the cup-like depression of another bone. The distal bone is capable of motion around an indefinite number of axes, which have one common center. It enables the bone to move in many places (nearly all directions).

An enarthrosis is a special kind of spheroidal joint in which the socket covers the sphere beyond its equator.

Example: Examples of this form of articulation are found in the hip, where the rounded head of the femur (ball) rests in the cup-like acetabulum (socket) of the pelvis, and in the glenohumeral joint of the shoulder, where the rounded head of the humerus (ball) rests in the cup-like glenoid fossa (socket) of the shoulder blade. It should be noted that the shoulder includes a sternoclavicular articulation joint.

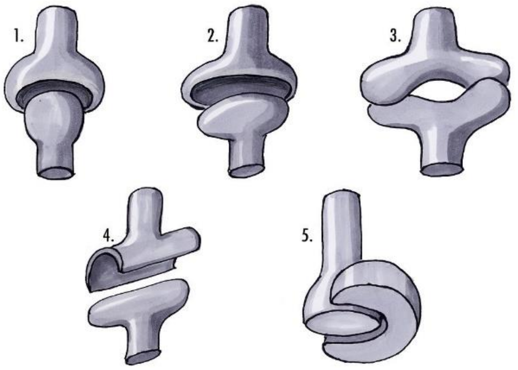

Figure 1:Joints 1. Ball-and-Socket-Joint 2. Ellipsoid Joint 3. Saddle Joint 4. Hinge Joint 5. Plane Joint

II Literature Survey

In 2016 advance in science and technology research journal Robert Karpinski et al. [1] proposed that the results of a preliminary study on the structural analysis of the hip joint, taking into account changes in the mechanical properties of the articular cartilage of the joint. Studies have been made due to the need to determine the tension distribution occurring in the cartilage of the human hip. These distribution are the starting point for designing custom made human hip prosthesis. Basic anatomy, biomechanical analysis of the hip joint and articular cartilage are introduced. The mechanical analysis of the hip joint model is conducted. Final results of analysis are presented. Main conclusions of the study are: the capability of absorbing loads by articular cartilage of the hip joint is preliminary determined as decreasing with increasing degenerations of the cartilage and with age of a patient. Without further information on changes of cartilage's mechanical parameters in time it is hard to determine the nature of relation between mentioned capability and these parameters.

In 2016 Tribology International Ehsan Askari et al. [2] proposed that the occurrence of audible squeaking in some patients with ceramic-on-ceramic (COC) hip prostheses is a cause for concern. Great effort has been dedicated to understand the mechanics of the hip squeaking to gain a deeper insight into factors contributing to sound emission from COC hip articulation. Disruption of fluid-film lubrication and friction were reported as the main potential cause, while patient and surgical factors, and design and material of hip implants, were also identified as leading factors. This article summarizes the recent available literature on this subject to provide a platform for future research and development. Moreover, high wear rates and ceramic liner fracture as viable consequences of hip squeaking are discussed.

In 2015 Mohammad Rabbani et al. [3] proposed that the stress distribution of a complete assembly of femur and hip prosthesis is investigated with realistic boundary conditions under nine routine activities using finite element analysis. In each activity, different forces of varying magnitude and orientation were applied on the prosthesis during a period of time to examine the critical points developed in the entire 3D model. This includes a full description of the geometry, material properties and the boundary conditions. The activities considered comprise slow walking, normal walking, fast walking, upstairs, down stairs, standing up, sitting down, and standing on 2-1-2 legs and knee bending. The findings of this study can be used to develop more optimized hip joint prosthesis by altering the prosthesis geometry to achieve a more balanced stress distribution.

In 2016 IJRASET Tushar V Kavatkar et al. [4] proposed that the Total hip replacement (THR) is one of the most successful applications of biomaterials in the medical industry. In THR, a spherical head connected to the femoral stem articulates against a spherical cup/liner attached to the pelvic bone. The tribological performance of artificial hip joints is a critical issue for their success, because adverse tissue reaction to wear debris causes loosening and failure. The wear of the bearing surfaces of hip joint prostheses is a key problem causing their primary failure. Many studies on wear of hip prostheses have been published in the last 10 years. Theoretical/ numerical models have been proposed for investigating geometrical and material parameters also.

III Development of Solid model of Hip Joint

Finite element modeling of Hip Joint, The Figure 2 represent Hip joint model developed using CATIA modeling package. The stem, cap and femur are separately built and assembled using CATIA assembly options. Standardized data based on experimental results as suggested is considered to developed Hip joint model.

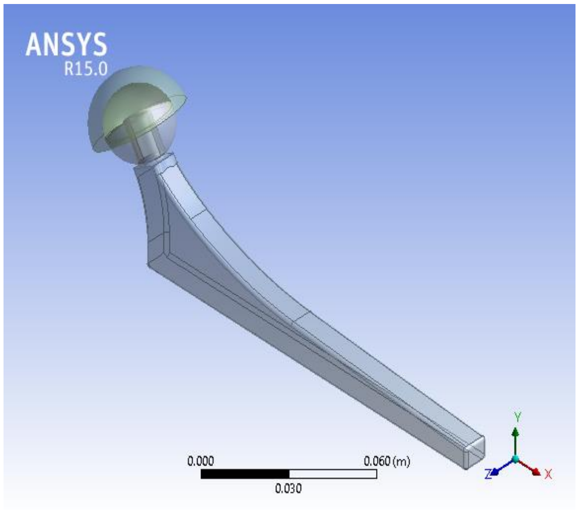

Figure 2: Solid Model

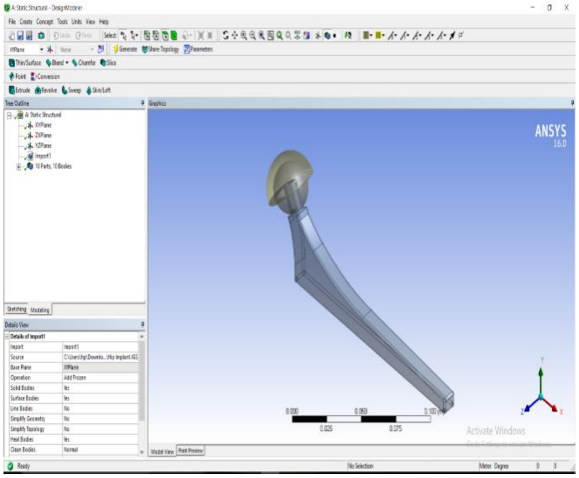

Figure 3: Showing the Import Geometry in the Ansys

IV Meshing

The model is exported to hyper mesh for meshing in step- file format and meshed using solid meshing options. The meshed view of the modeled is shown in Figure 4. The structure is Tetra meshed due to complicated geometry with internal Cancellous and cortical bones type geometries. Solid 45 is a 4 nodded element with three degree of freedom at each node. Contact Elements TARGE169 is used to represent various 2-D "target" surfaces for the associated contact elements. Conta171, Conta172 and conta175 are used to represent various 3-D solid elements. Contact of elements takes place when surface element penetrates the target segment element. Contact condition of completely bonded type is selected for contact.

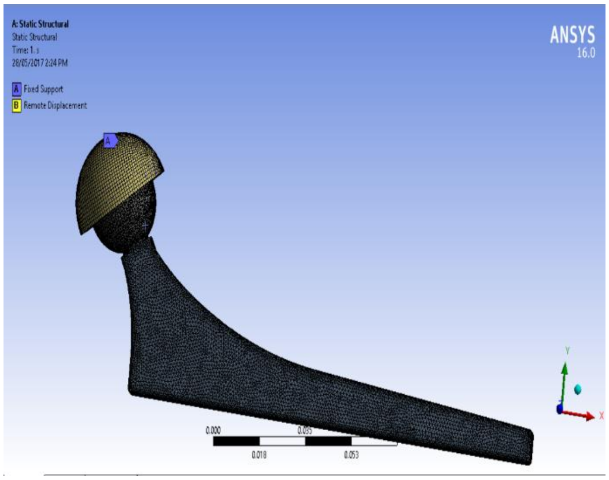

Figure 4: Showing the meshing of the hip joint assembly

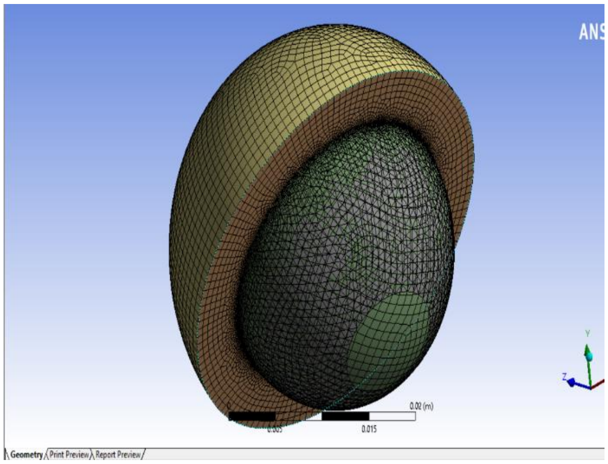

Figure 5: Showing the Mesh

V Material Properties for the Femur Head and Stem Neck

Two different materials are used in the present finite element simulation: Cobalt Chromium (Co Cr Mo) alloys fall under two main categories: cast alloys (ISO 5832- 4) and wrought alloys (ISO 5832-12). Cast Co-Cr-Mo exhibits elevated mechanical properties and optimal corrosion resistance under friction condition. Its main drawbacks are related to their poor fatigue resistance and their high cost. Wrought Co-Cr-Mo is even more expensive than cast material, but the higher cost can be justified by the enhanced corrosion and fatigue resistance. In the presence study wrought alloy is used to simulate a prosthetic metallic head material due to their high strength and sufficient biocompatibility in clinical conditions.

Table 1: Material Property of Co-Cr-Mo and Ti6Al4V

Material Properties	**Cobalt-Chromium-Molybdenum Alloy (Co-Cr-Mo)**	**Titanium Alloy (Ti6Al4V)**
Young's Modulus (Gpa)	230	114
Tensile Strength (Mpa)	530	850
Ultimate Tensile Strength (Mpa)	890	960
Density (Kg/m^3)	8300	4420
Expansion (m/m.c^o)	13.6x10^{-6}	9x10^{-6}
Poisson ratio	0.3	0.35

VI FEM Simulation

It is well comprehended that contact pressure in the hip joint is closely related to the three dimensional coverage of the socket of the hip bone, and the mechanical stress inside the cartilage increases as contact pressure rises. Finite Element Analysis (FEA) or Finite Element Method (FEM) is a relatively new method for solving complex engineering and mathematical problems. Since the 1940s, this method has evolved into the method of choice for computational analysis. In the early years, the finite element method was limited to the manpower available to solve large matrices. However, as technology has evolved FEM has evolved into a computational jug naught. This process is now only limited by the capabilities of the available hardware to solve matrices that can go out to machine epsilon. FEM is a part of many engineering applications such as structural mechanics, heat transfer, fluid flow, electromagnetic, blade design in orthopedic design for implants and prosthetics. It has become a key part of the design and refinement processes in engineering.

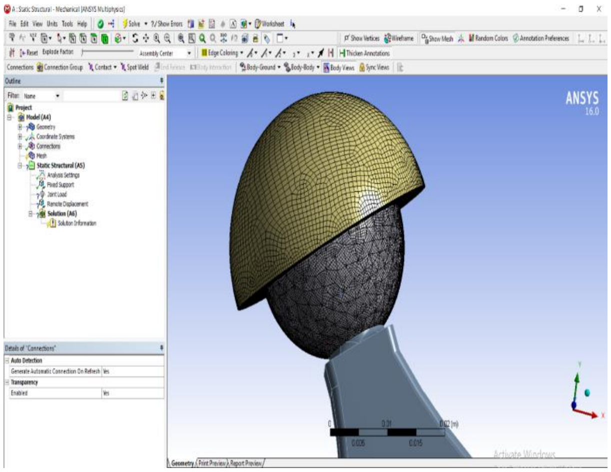

Figure 6:Showing the analysis of joint in Ansys

VII Fem Simulation Flow Chart

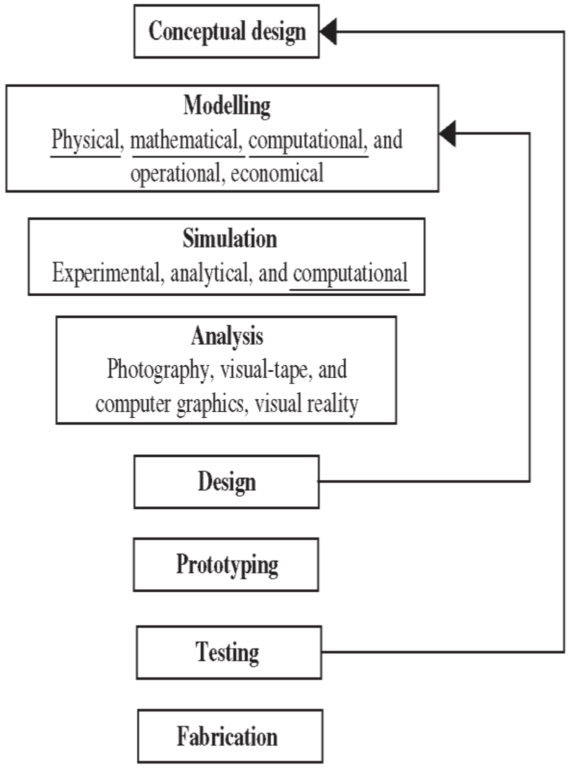

Figure 7: Flow chart illustrating the FEM procedure

The analysis has been done and shown at different interval of time the frame by frame simulation at total deformation shown in figure 8.

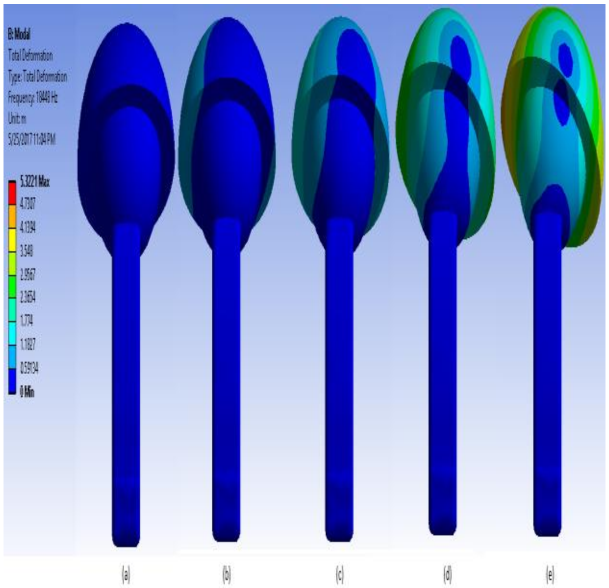

Figure 8: Frame by Frame deformation FE simulated results

VIII Deformation under different frequencies

In the present work the FEM simulation shows that deformation under different frequency the contours help to understand the effects during the load applied on the ball socket joint the red colour indicates the higher value and the blue color show the lower value of the deformation under Frequency 18789 Hz, 22157 Hz, 18448 Hz, 50647 in Fig 9, Fig 10, Fig 11, and Fig 12.

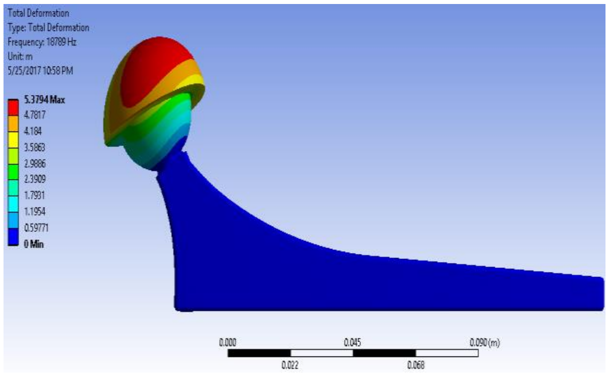

Figure 9:Showing the deformation at 18789 Hz

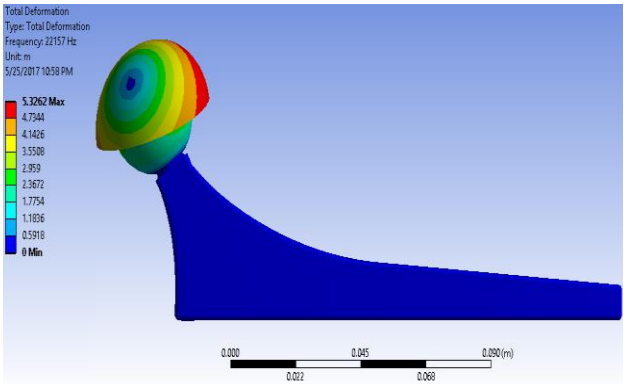

Figure 10:Showing the deformation at 22157 Hz

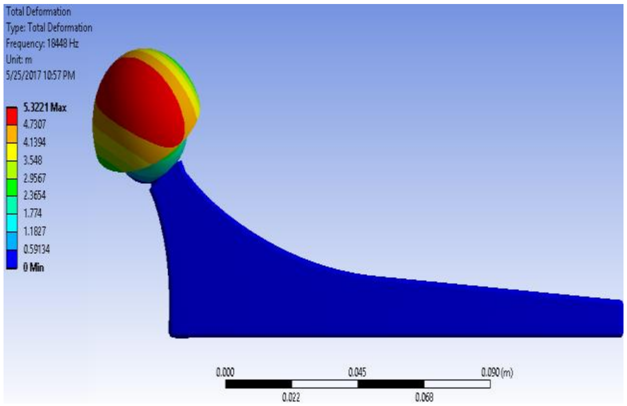

Figure 11: Showing the deformation at 18448 Hz

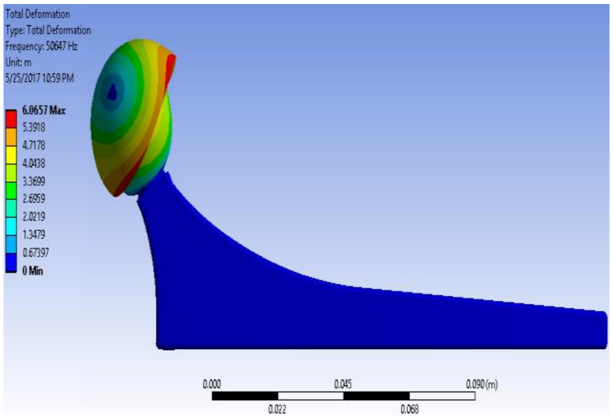

Figure 12: Showing the deformation at 50647 Hz

Here we uses the different Frequencies in order to analyze the deformation of the hip joint. Here in this analysis the deformation is varying over the joint. So in order to increase the life of the joint the variation of deformation over the joint must be less as much as possible. So in order to find the optimal condition, here it analyzed the deformation condition at different frequency at normal pressure

Table 2 Showing the Deformation at the joint under different frequency at normal pressure

S.N	Frequency (Hz)	Minimum Deformation at the joint (m)	Maximum deformation at the joint (m)	Mean deformation at the joint (m)
1	18789	0.59771	5.3794	4.184
2	22157	0.5618	5.3262	4.1426
3	18448	0.59134	5.3221	4.1394
4	50647	0.67397	6.0657	3.3699

After analyzing the hip join at different frequencies under normal pressure condition it is shown that at 50647 Hz the deformation over the hip joint is mean while less as compared to the other frequencies during normal pressure condition. Therefore 50647 Hz was the best frequency during the normal pressure condition in order to increase the life of the joint.

IX Strain Energy

The strain energy at the three critical cross-sections of femur induced by the applied forces was computed using in-house developed MATLAB codes and the data extracted by APDL codes from the obtained finite element solutions. The plane boundaries of the three critical cross-sections, extracted from the finite element mesh, were imported to MATALB to generate a 2D mesh for calculating the cross-section strain energy. With the help of FEM simulation we can predict the Strain energy values for ball- socket hip joint results dhows the variation strain energy at normal pressure (75kg/cm^2) in Figure 14.

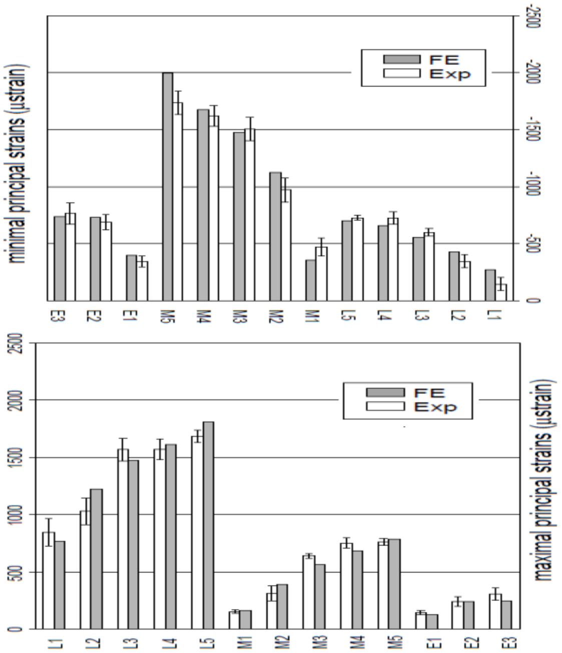

Figure 13: Titanium aluminum alloy (Strain)

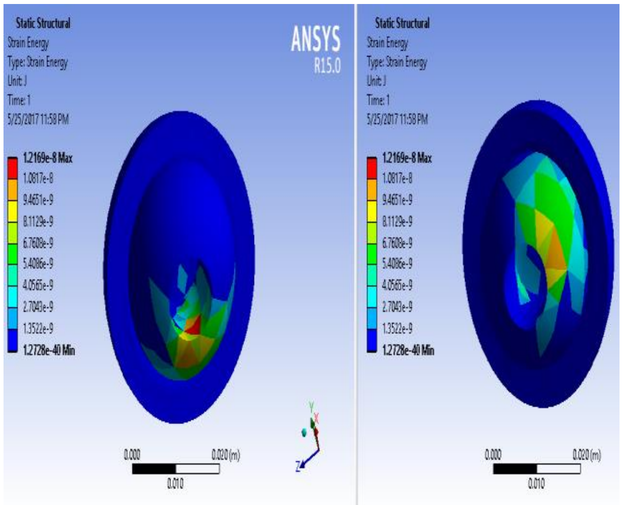

Figure 14: Showing the values of strain energy at normal pressure

X Response Surface Methodology:

Mainly response surface methodology (RSM) was carried out in this study. Usually, the correlation between the dependent variables and independent variables is either extremely complex. However, RSM gives a procedure which solves this problem. Assume that the decision maker is concerned with a system involving a dependent variable Y, which effects on the independent variable xj. It is also taken that xj is continuous and convenient. With RSM, the functional relationship between the output y and the levels of n input parameters can be written as:

$$y = f(x_1, x_2, x_3, x_4, \ldots .x_n) \tag{1}$$

A mathematical model for such a relationship does not necessarily exist. Thus, the first step in RSM is to get a suitable approximation for $y = f(x_1, x_2, x_3, x_4, \ldots .x_n$ using a low order polynomial in some section of the independent variables. If the approximated function has linear variables, a first-order polynomial can be used and written in terms of the independent variables

Axial stress:

The regression analysis is carried out for the given output responses. First the regression table axial stress is shown in Table no. 3 in which pressure, the square term of pressure and the interaction between volume fraction and pressure have significant value as their values are less than p=0.05. Here R-square value comes 99.94 % which is acceptable. Then Analysis of Variance (ANOVA) analysis for axial stress has been done in Table no. 4 in which the total degree of freedom of input parameters is 14.

Table 3: Regression table axial stress

TERM	COEFFICIENT	SE COEFFICIENT	t	p
CONSTANT	-28.52	11.87	-2.263	0.0689
PRESSURE	1.9	0.12	15.019	0.00
FREQUENCY	158	164.35	0.953	0.378
DEFORMATION	0.25	0.48	0.529	0.621
PRESSURE* PRESSURE	-0.03	0.00	-17.83	0.00
FREQUENCY* FREQUENCY	-1545.7	2002.47	-0.772	0.480
DEFORMATION* DEFORMATION	-0.00	0.01	0.722	0.503
PRESSURE* DEFORMATION	0.01	0.00	2.609	.409
FREQUENCY* DEFORMATION	-4.6	3.13	-1.457	0.205
PRESSURE* FREQUENCY	2.39	1.34	2.025	0.010

S = 0.246214 PRESS = 3.52163
R-Sq. = 99.93%, R-Sq. (pred) = 99.29%, R-Sq. (adj) = 99.65%

Table 4: Analysis of Variance for Stress

SOURCE	**DF**	**SEQ SS**	**Adj SS**	**Adj MS**	**F**
regression	9	548.117	547.88	60.89	1005.87
linear	3	523.592	15.1	4.6654	77.16
deformation	1	1.457	0.017	0.01474	0.29
pressure	1	525.1	13.770	13.5514	225.60
frequency	1	0.125	0.054	0.05548	0.91
square	3	20.304	20.415	6.9140	112.27
pressure* pressure	1	0.026	0.032	0.316	0.54
frequency* frequency	1	20.373	20.40	20.3654	335.198
deformation* deformation	1	0.045	0.039	0.4135	0.61
interaction	3	0.789	0.80	0.1307	4.45
pressure* frequency	1	0.422	0.425	0.2487	6.91
pressure* deformation	1	0.129	0.135	0.4122	2.16
frequency* deformation	1	0.250	0.254	0.1304	4.10
residual error	5	0.328	0.311	0.2481	-
lack of fit	3	0.204	0.214	0.0605	1.50
pure error	2	0.098	0.095	0.0490	-
total	14				

The first order quadratic equation is generated which is given below:
STRESS = 4.42000 + 0.08570* A + 0.64800*B + 20.50000*C

Residual plots:
Stress:

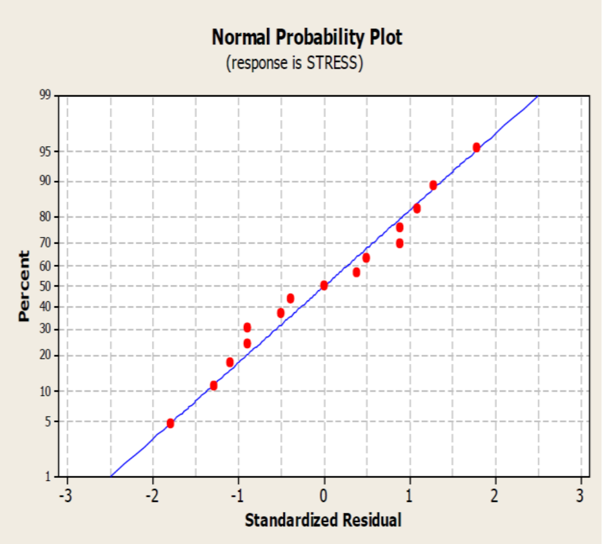

Figure 15: Showing the plot of residual Stress inside the hip joint during the normal pressure condition

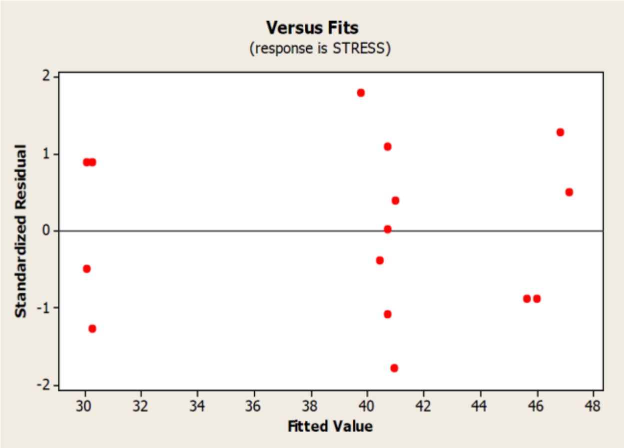

Figure 16: Showing the plot of Fitted value Vs Standardization Residual Stress

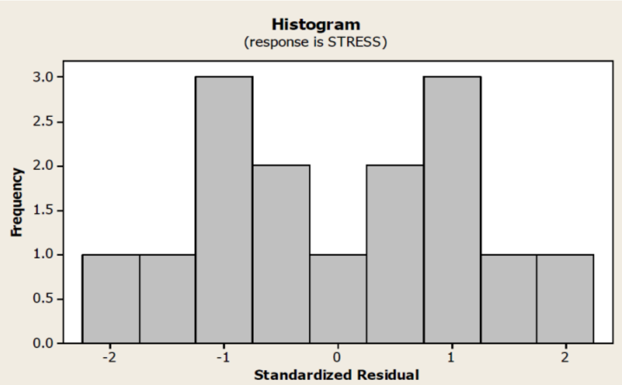

Figure 17: Showing the values of Residual Stress at different frequencies

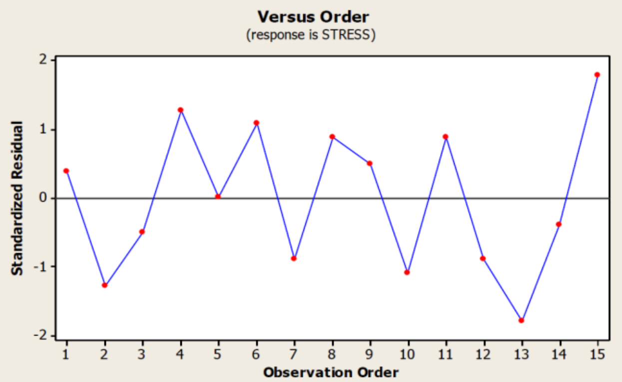

Figure 18: Showing the normal Probability plot of output response of axial stress

The normal probability plot of output response of axial stress is plotted in Figure 15 in which almost all the points are situated on the straight line. Thus, the results which are obtained for stress are correct. Then fitted value vs. standardized residual value plot is given in Figure 16. In this graph no pattern of these points is formed. So, the input parameters are fitted well in 95 % confidence interval. The histogram plot of stress is shown in Figure 17. in which all the columns are formed into normal probability distribution. Therefore, the present result indicates the realistic analysis of abrasive flow machining which is carried out in a successful manner. The plot between standardized order and residual values is presented in Figure 18. where minimum stress is occurred in the 13th run and similarly at the 15th run the maximum value is obtained.

Residual Plots

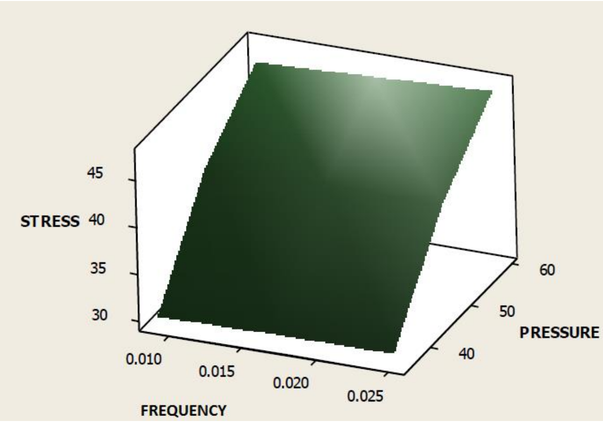

Figure 19: Surface plot of stress vs pressure, frequency

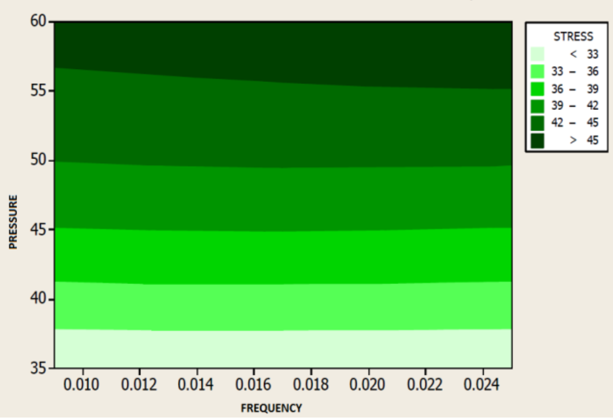

Figure 20: surface plot of pressure vs frequency, Stress

After analyzing the different condition it is found that for each frequency as the pressure increases the value of the stress is also increases. But in order to increase the reliability and life of the joint the value of the applied pressure must be in the range of 40 – 45 kPa.

XI CONCLUSION

After analyzing the hip join at different frequencies under normal pressure condition it is shown that at 50647 Hz the deformation over the hip joint is mean while less as compared to the other frequencies during normal pressure condition. Therefore 50647 Hz was the best frequency during the normal pressure condition in order to increase the life of the joint. A reliable methodology to assess hip fracture risk in individuals is crucially important for preventing hip fracture and initiating a repair work. The purpose of this study is to propose a more effective hip failure risk index that is based on the strain energy failure criterion, and it is able to better describe joint failure mechanism. The proposed failure risk index can predict not only the failure risk level, but also the potential failure location. The results of this study showed that there is a very low hip failure risk at optimum frequency and at optimum stress, while, during the, there is a high Failure risk at the femoral neck and the intertrochanteric region, compared to the sub trochanteric region.

REFERENCES

1) Robert Karpinski, Lukasz Jaworski, JaroslawZubrzycki, “Structural Analysis of Articular Cartilage of The Hip Joint Using Finite Element Method”, Advances in Science and Technology Research Journal 2016.
2) Ehsan Askari, Paulo Flores, “A review of squeaking in ceramic total hip prostheses”, Tribology International 2016.
3) Mohammad Rabbani and Hossein Saidpour, “Stress Analysis of a Total Hip Replacement Subjected to Realistic Loading Conditions”, Verizona Publisher 2015.
4) Tushar V Kavatkar, Milind S Kirkire, Harshal Salvi, DipakPatil, “Wear Analysis of Hip Joint Prosthesis”, IJRASET 2016.
5) C. Desai, H. Hirani, A. Chawla, “Life Estimation of Hip Joint Prosthesis”, J. Inst. Eng. India Ser. C 2014.
6) EkoSaputra, IwanBudiwan Anwar, J. Jamari, Emile van der Heide, “Finite Element Analysis of Artificial Hip Joint Movement during Human Activities”, Science Direct 2013.
7) RajiNareliya et al. Biomechanical analysis of Human femur Bone: International Journal of Engineering Science and Technology (IJEST), (2011).
8) Eng. Radu RACA'AN, Contributions Regarding the use of Modern Techniques for Measuring and Modeling Complex Surfaces in the Wear Evaluation of Total Hip Replacements, (2011).
9) JIANG Hai-bo, LIU Hong-tao, HAN Shu-yang, LIU Fen, Biomechanics Characteristics of New Type Artificial Hip Joint, Advances in Natural Science Vol. 3, No. 2, 2010, pp. 258-262.
10) J. Lord, T. Joyce, Analysis of failed metal-on-metal hip prostheses,
11) Dianne-Anand, Wear Analysis of Acetabular Components in a Total Hip Replacement, pp:1.
12) D Dowson, New joints for the Millennium: Wear control in total replacement hip joints, Journal of Engineering in Medicine 215: 335.