K.V. Datar, Y.P. Reddy, Ravi Sarangapani / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com
Vol. 2, Issue 6, November- December 2012, pp.994-1001

Fatigue life analysis of partial hip endoprosthesis for an activity of brisk walking

K.V. Datar* Y.P. Reddy Ravi Sarangapani*****

* (Master of Engineering (Mechanical- Design Engineering)
Sinhgad College of Engineering, Vadgaon (Bk), Pune – 41 (Maharashtra)
** (Professor, Department of Mechanical Engineering,
Sinhgad College of Engineering, Vadgaon (Bk), Pune – 41 (Maharashtra)
*** (Vice President, Development and Quality Management
Adler Mediequip Pvt Ltd, Hinjewadi Phase II, Pune – 57 (Maharashtra))

ABSTRACT

Hip joint implant surgery is becoming common not only in old aged people, but also in comparatively younger persons. The load experienced by the hip implant is of fluctuating nature, and hence, the prediction of fatigue life of the hip implant is of utmost importance. This paper deals with the preparation of finite element model, its validation through physical testing, and prediction of fatigue life of the implant considered through finite element analysis. The validation of FEA model is done as per the guidelines of ISO 7106 (4). The orientation of implant and the loading is done as per ISO 7206 in both, FEA and physical testing. The activity considered for predicting fatigue life is brisk walking.

Key words: fatigue life, hip implant, partial hip endoprosthesis, fatigue analysis

1. Introduction

Human bones and joints are continuously acted upon by varying forces. The variation in boundary and human body conditions further increases the gravity of the problem due to which, the joint replacement surgeries are increasingly carried out. In these surgeries, human bone or joint is replaced by an artificial bone or joint, made up of bio-materials like stainless steel, titanium, composites and others.

In the earlier days, it was assumed that joint replacement surgery is to be performed only on older patients with very limited physical movements, and hence, life prediction of the joint was not given much importance. But in modern era, even younger patients are being treated with the joint replacement surgeries. It will be wrong to expect such patients to restrict their movements to merely walking for a few times in a day. This paves the way for formulating a validated finite element model of implant, with which, its life can be predicted for various activities of patient and loading conditions arising from those activities. If such practice of predicting fatigue life data is adapted to all types of implant variants available in the market, it would be easier for the surgeons, and the patients to select a proper implant as per the patient's physical needs. The surgeons will also come to know about the credentials of the product they are going to use, with the availability of the fatigue life data. This research work takes into consideration a typical physical activity of brisk walking to predict the fatigue life of the implant. The activity of brisk walking is specifically selected to find out the speed of walking which can give sufficient life of the implant. The brisk walking is considered to be with a speed of 2.1 m/s and the normal walking speed is considered to be 1.5 m/s.

This paper is based on the analysis of a hip joint implant under varying load conditions during brisk walking, and fatigue life prediction of the same joint. A typical type of hip implant called modular bipolar implant is selected for this analysis. The scope of the present work is

a) to prepare a computer aided drawing (CAD) model and Finite Element Analysis (FEA) model of implant
b) to find stresses developed at four critical locations on the implant through FEA, by applying loads as per ISO 7206.
c) to validate the FEA model through physical testing done as per ISO 7206, and
d) to calculate fatigue life of the implant for activity of brisk walking with a speed of 2.1 m/s.

2. Literature Review

Human body undergoes a number of dynamic forces as a result of various activities performed by the human. All these forces pass through the human bones and joints and are transmitted through entire human body. During this process, the human bones and joints have to carry significant amount of forces through them. Hip joint is one of such joints, which primarily transmits the forces developed due to a person's self weight, and some secondary forces like dead weights carried by the person. The force developed in the joint is in multiples of body weight, depending upon the activity being carried out. Normally the hip implant is strong enough to take these loads. But, accident,

injury and age can damage the joint, limiting its flexibility and making the movement painful. The hip joint is a large ball and socket joint composed of two parts, 1) Femur- the spherical head of the thighbone, and 2) Acetabulum- the concave spherical surface of a bone in the pelvis.

The joint replacement surgery is one of the way to get rid of pains arising from damaged hip joint. The damaged portions of the hip are replaced with artificial parts having smooth and durable surfaces during hip joint replacement surgery which will allow the joint to function properly. There are two types of hip replacement surgeries – a) total hip replacement, in which, both parts of the hip joint are removed and replaced. b) partial hip replacement, in which, only the lower part of hip joint made up of femur head is replaced.

In partial hip replacement, depending on the position of centres of the femoral head, and acetabular spherical surface there are two types of endoprostheses:

i) Conventional metallic proximal femoral endoprostheses - In this type, the replaced femoral head, and the natural acetabular surface are deliberately kept concentric.

ii) Bipolar hip endoprostheses [1] - It has an eccentric acetabular surface with respect to the centre of the femoral head, when the outer head is in its neutral position.

The bipolar hip endoprostheses was developed in an attempt to eliminate the problems faced in conventional concentric models. The aim is to achieve low friction between metal and polyethylene inner bearing motion while decreasing shear stress across the acetabular cartilage. The compounded motion between inner and outer bearing surfaces also provided a greater range of hip joint movement, thereby reducing the likelihood of dislocation and loosening. However, problems of acetabular wear and erosion, outer head dislocation, and inner bearing separation or fracture still occurred. These problems were related to loading the outer head in a condition when the implant axis became more and more inclined with vertical. It is called varus position. It increases the bending moment acting on the implant. To eliminate this problem, a new design, called Uneversal Head (UHR) endoprostheses is introduced [1], which incorporates a small polar offset between the outer head centre and inner bearing centre. This design procedure is known as ‘anti varus head dynamics’. In the present research work, a bipolar implant similar to UHR with the centre of femur being two millimeters eccentric to the centre of the acetabular surface is used. A paper published at Royal Society of London describes the force actions transmitted by joints in the human body [2]. In this study, the angles at which the joint forces are maximum and the magnitude of these maximum forces as a multiple of body weight are determined. An analysis to determine the magnitude and direction of the resultant force transmitted between the femoral head and the acetabulum during stationary weight bearing on one foot is presented. It is found that the value of resultant force is 2.92 times the body weight. The author concludes through this research, that the hip joint force would be 4.5 times body weight during normal walking and 8 times the body weight during brisk walking.

The hip joint forces produced in two patients were compared under in-vivo conditions [3]. One of these two patients had undergone partial hip endoprostheses and the other complete hip endoprostheses. It was found that the joint forces increase for initial few days, then reduce and settle at a certain value. A research paper in development of fatigue lifetime predictive methods for hip implants was presented [4] with the aim of developing accelerated fatigue simulation testing procedures. This testing procedure is used to enhance the methodology of hip implant lifetime prediction. The combined effect of fatigue and wear on the biomaterials used for implants was analysed [5], and it was concluded that the forged steel components have better fatigue life than the cast components. A standard protocol is proposed [6] for pre clinical testing of the human implants. This protocol establishes a new method for testing of the implants.

The International Standards Organization has released norms for testing various types of hip joint implants. The ISO 7206 defines specific conditions for fatigue testing of the hip implant. The standards specifies that the fatigue testing is to be done by varying only the magnitude of force acting on the implant, by keeping the angle of force with stem axis constant. Hence for new generation fatigue implants that can be used for younger patients, the ISO 7206 remains necessary but not sufficient criteria to be satisfied. In the present research work, the ISO 7206 norms have been used to validate the FEA model.

In the present study, the ‘endofit’ hip implant is used for analysis. This stem implant, when used with the ‘moduloc’ femure head cup, forms a modular bipolar hip implant, and exhibits anti varus head dynamics properties. The stem and head are made of ‘high nitrogen stainless steel’ with a typical composition of ‘21Cr10Ni2.5MO’. This alloy is also termed as Ortron-90 or Rex 734. The material complies with the requirements of ASTM F-1586 and ISO 5832-9

3. Analysis and validation of FEA model

A three dimensional CAD model of the implant is prepared by using ‘Solidworks’ software. The model consists of two parts; the stem and the femur head, and later, assembled together. The actual implant is also manufactured and assembled in same way.

3.1 Finite element analysis

The STEP file of the CAD model is imported to Ansys Workbench V11 software to prepare a finite element model. The purpose of this model is to find out stresses and deflection in the implant when oriented and loaded as per ISO 7206 test protocol. Fig. 1 shows the FEA model of the hip implant with loading mechanism. The block number two acts as a plunger of the machine, imparting the force to the top of the femur head. The modulus of elasticity of the plunger is identical to that of the femur head material to avoid penetration of harder material into softer material. The block number one acts as a supporting frame to the plunger, so that the plunger can have only vertically downward motion with respect to the frame. It ensures that only vertical load is applied on the top of the femur head. The two blocks are considered to be a frictionless joint.

The contact between the head and the stem is perfectly rigid without any relative motion between them. The block number three, which is a stem holding frame, represents the solidified epoxy resin. It surrounds the stem, and grips it from outside. The modulus of elasticity of the stem holding frame at the stem surface is considered same as that of epoxy resin, holding the stem. The reference value of 6000 N/mm^2, mentioned in ISO 7206 is considered for this purpose.

The ISO 7206 gives test procedure to find out whether the implant being tested satisfies the norms to have sufficient life under varying loads. The FEA model of implant needs to be tilted in frontal and lateral planes to orient it as per the ISO 7206 norms.

The model of the implant is complicated in shape and consists of a number of curved surfaces. The geometry of the femur head and stem model is not mesh friendly during mapping because of its non standard shape. Hence, for meshing of this model, auto mesh is adopted. Solid tetrahedron elements with mid side nodes are selected, as this type is found to be more suitable for irregular and curved nature of the model. The mesh density is considered as 3 mm at the boundaries and 6mm at the core. The Fig. 2 shows the meshing of implant. A static vertically downward load of 2300 N is applied on the top surface of the femur head, after the meshing is completed. The solution is run in FEA solver after application of the load.

3.2 Results of finite element analysis

In the finite element analysis of implant model, four locations are found to be critical in terms of stress developed. The inner and outer surfaces of neck of the stem are two such heavily stressed areas, because cross sectional area in this region is the least. The inner and outer surfaces of the stem where it touches the surface of holding epoxy resin are the other two areas, since the bending moment produced by the applied load is maximum at these points. These locations are named as 'Surface Out', 'Surface In', 'Neck Out', and 'Neck In' as shown in Fig. 2. The maximum stress developed under a static load of 2300 N is 201 N/mm^2 at Surface In location of the stem, and it is compressive in nature.

The endurance strength S_e' of the implant material is 500 N/mm^2. However, the endurance strength of actual implant is less than that of standard test specimen due to the effect of size, surface finish, reliability, temperature, and load. The value of endurance limit of the actual implant calculated from S-N curve is found to be 307 N/mm^2 and the maximum stress developed at the surface, calculated by analytical method is 216.1 N/mm^2.

Since the maximum stress developed in the implant, calculated through FEA and through analytical method, is less than its endurance limit, the implant has infinite fatigue life under the loading mentioned in ISO 7206. The stress developed in the implant under static loading will be same as those developed under repeated loading.

4. Physical testing

The testing of the implant is done under static conditions, by orienting the implant as per ISO 7206 protocol. The stem of implant is held in epoxy resin with stem axis making 10^0 angle with vertical in frontal plane and 9^0 in the lateral plane. The liquid epoxy resin cannot hold the stem, and hence, a fixture is manufactured to hold the stem till epoxy gets solidified. The test set up used for physical testing of the hip implant consists of three parts namely strain gauge set up, data acquisition system and loading frame.

The strain gauges work as a sensor to capture the data of strain developed at critical locations of the implant during the loading process. The strain gauges are glued to the surface of the specimen. The locations for mounting strain gauges are fixed based on the results of finite element analysis. Four uniaxial foil type strain gauges are used to form quarter strain gauge bridges. A high speed multi channel data acquisition system with simultaneous sampling and hold capability to record the strains during the tests is used. This data acquisition system reads and represents the data generated by strain gauges in terms of strain and stress developed at a certain point of time. An electronically controlled Universal Testing Machine (UTM), with a capacity of 60kN is used as loading frame, to apply the load as per the ISO 7206 standards. Fig. 3 shows the data acquisition system and the implant with strain gauges loaded under the loading frame.

4.1 Results of physical testing

The compressive stress is developed at Neck In and Surface In locations of the implant. The tensile stress is developed at Neck Out and Surface Out locations of the implant. Fig. 4 shows a comparative chart of stresses developed at four different locations with respect to time.

4.2 Comparison of results

The comparison of results of FEA and physical testing is shown in Table 1. It can be concluded from the comparison, that the results are fairly matching to each other. The stresses developed at all four critical locations of implant during physical testing are within 10 percent variation from the values obtained in FEA. This is an acceptable limit of variation for FEA. The magnitude of stress at the 'Surface In' location, evaluated by analytical calculations also matches with the stress obtained by FEA and physical testing. It can be safely concluded that the FEA model is validated, because the results of FEA and physical testing are matching with each other within the acceptable limits.

5. Fatigue life analysis

Fatigue is the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. Fatigue life is the number of applied repeated stress cycles a material can endure before failure. The loading considered for the fatigue life prediction is the load sustained by the implant when a person takes a brisk walk at a speed of 2.1 m/s. The maximum force acting on the hip joint is a function of speed. If the speed is decreased from 2.1 m/s to 1.5 m/s, the maximum joint force decreases by half. If the stresses developed in FEA of the hip joint map with the fluctuating load during walking, this analysis can be directly used to find the stresses induced at less walking speeds. This is done by considering the ratio of the loads during two different speeds to be

The following procedure is adopted for fatigue life analysis of the implant-

1) Analysis of load acting on the hip joint during a brisk walk.
2) Analysis of angle made by loads in frontal and lateral plane during brisk walking.
3) Application of these forces and angles in the FEA model which has been already prepared and validated.
4) To find maximum and minimum principal stresses at four critical locations on the implant.
5) To find the mean and the amplitude stresses, equivalent stresses and the fatigue life based on the maximum equivalent stress.

A test subject of 70 kg is considered for this analysis. The load is considered to be varying in a cyclic manner with respect to the percentage of cycle time [2] and the load is shown as a multiple of body weight. A cycle comprises of two steps, starting from one foot hitting the road till the time it again hits the road. During one cycle, each of the two foots are in contact with the surface for only 50% of the time. Hence the load curve shows the load being carried by one leg only for 50% of the cycle time.

The maximum load considered here is 8 times the body weight. In the present analysis, the maximum force is considered to be 5494 N which is equivalent to 560 kg. The force exerted on the hip joint is found to be maximum at 7% and 47% of cycle time. The force acting on the joint and the angle made by this force with the stem axis vary with respect to the percentage of cycle. The stem axis is defined as the line joining the centre of hip joint and knee joint. The cycle time varies from zero to fifty percent for every step. This time period is further divided into fifteen sub steps for analysis and evaluation of stresses developed, as given in Table 2. The FEA analysis is run for each combination of the force, angle in the frontal plane and angle in the lateral plane. The FEA is done only for thirteen load cases since the load acting at zero and fifty percent of cycle time is zero. The maximum and minimum principal stresses are found out through FEA for all the thirteen load cases.

6. Results

The principal stresses developed at the four critical locations are evaluated through FEA. The locations chosen are same as those selected during validation phase of the model. The nodes selected for four chosen locations are same in all the sub steps. The Table 3 shows maximum and minimum principal stresses developed during finite element analysis for all the fifteen load cases. The maximum principal stresses at the outer locations, Surface Out and Neck Out have large positive magnitudes and the minimum principal stresses at these locations have marginal values. However, the minimum principal stresses have large negative values on both the inner surfaces, Surface In and Neck In, and marginal values at both the outer surfaces. The absolute stresses for each of the fifteen sub steps are calculated based on the results of maximum and minimum principal stresses. The maximum tensile stress is found to be 492.06 N/mm^2 at Neck Out location, and the maximum compressive stress is found to be 458.09 N/mm^2 at Neck In location, as given in the Table 4. The amplitude and the mean stresses calculated for all the four critical locations, considering the maximum and minimum values of the absolute stresses developed at these four locations are given in the Table 5. The induced stresses at each of the four locations are calculated

based on the values of the ultimate tensile strength, mean stress and amplitude stress, with the help of Goodman's equation. The maximum induced stress is found to be 326.31 N/mm^2, acting at Neck Out location, as given in the Table 5.

The absolute stresses at all four locations are mapped on the force versus percentage of cycle time graph to check the relationship between applied load and developed stresses, as shown in the Fig. 5. The pattern of the load and stresses shows that stresses are mapping the pattern of load. The stresses developed vary in the same pattern with which the load varies. If the load is increased or decreased with a certain percent, the stress pattern will also exhibit the same change in its magnitudes.

The calculated fatigue life of the implant based on the S-N curve and the maximum induced stress is 6,80,000 cycles. This is the expected life of implant when the test subject walks continuously at a constant speed as high as 2.1 m/s. If the test subject is considered to walk 2 km per day, the life of the implant would be 225 days. The average speed of walking of a common man is around 1.5 m/s. The stress developed at this speed will be fifty percent of the stresses developed at a speed of 2.1 m/s. Hence, the stress on most severely affected location in the implant with a speed of 1.5 m/s will be approximately 163 N/mm^2. It is less than the endurance limit of the implant.

7. Conclusions

This paper presents a procedure for analysing the fatigue life of hip implant for a physical activity of brisk walking. The FEA modelling and physical testing is carried out for this purpose as per ISO 7206 norms. The comparative results are within the acceptable limits. This shows that the implant satisfies the conditions set by ISO 7206. The induced stresses developed at four different critical locations during brisk walking at a speed of 2.1 m/s are found out from finite element analysis. The comparison of FEA results with the endurance strength of the material suggests that on three of the four critical locations selected for testing, the stress induced is less than the endurance limit. The location 'Neck Out' is the only point where the stress induced is greater than the endurance strength. Hence the fatigue life of implant is defined by this most heavily stressed location on the implant surface. The remaining three locations exhibit infinite life in fatigue. But, due to finite life of the most critically stressed location, overall life of the implant is also finite. It is found from this work that the life of the hip implant is 680000 cycles or 225 days for brisk walking at a speed of 2.1 m/s. The life of hip implant becomes infinite if the test subject walks at a speed of 1.5 m/s.

References

1. S. Krein, and E Chao, Biomechanics of bipolar hip endoprostheses, *Journal of Orthopaedic Research*, *Vol.2*, 1984, 356-368
2. J. Paul, Force actions transmitted by joints in the human body, *Proceedings of the Royal Society of London series B, Biological Sciences*, *vol.192*, 1976, 163-172
3. G. Bergmann, and F. Graichen, Hip joint loading during walking and running measured in two patients, *Journal of Biomechanics*, *Vol.26*, 1993, 969-990
4. C. Styles, S. Evans, and P. Gregson, Development of fatigue lifetime predictive test methods for hip implants, Part I Test Methodology, *Biomaterials*, *vol.19*, 1998, 1057-1065.
5. S. Teoh, Fatigue of biomaterials - a review, *International journal of fatigue*, *vol. 22*, 2000, 825-837.
6. P. Prendergast, and S. Maher, Issues in pre clinical testing of implants, *Journal of materials processing technology*, *vol. 118*, 2001, 337-342.

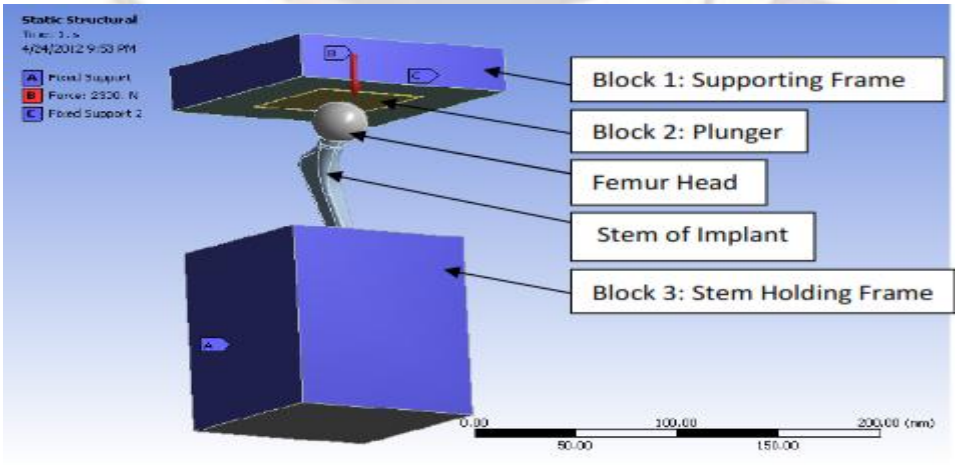

Fig 1: FEA model of the hip implant

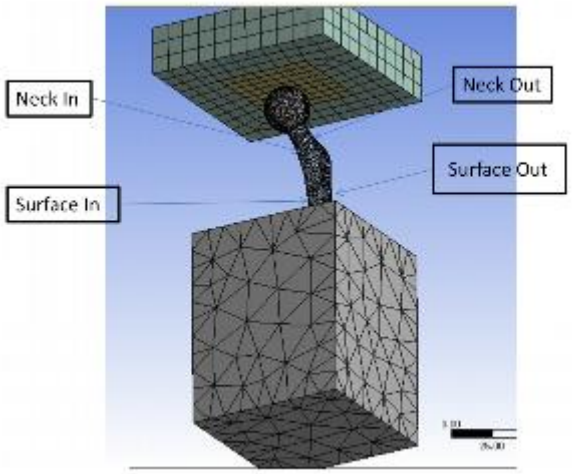

Fig 2: Mesh model in FEA

Fig 3: Data acquisition system and loading frame

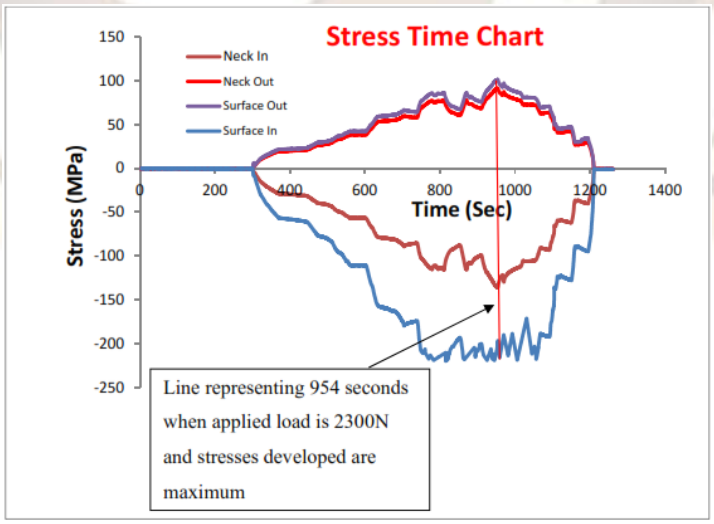

Fig 4: Stresses developed at four critical locations of the implant.

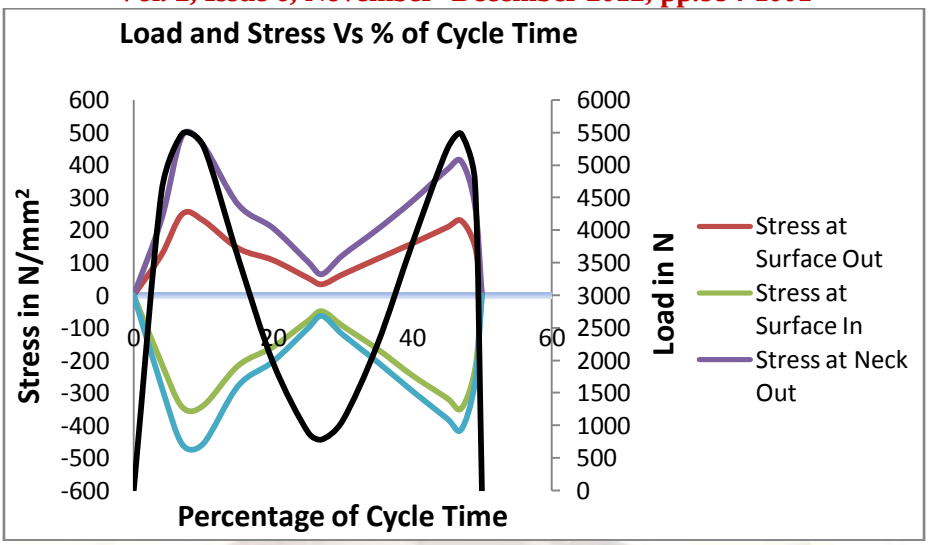

Fig 5: Mapping of stress and load graphs versus cycle time

Table 1: Comparison chart of FEA and physical testing results

Sr No	Position	Magnitude of Stress (MPa)		Nature of Stress	Error with Reason
		FEA	Physical Testing		
1	Neck Inner	127.28	135	Compressive	Within acceptable limits of +/- 10%
2	Neck Outer	87.198	91	Tensile	
3	Epoxy Surface Inner	201.09	217	Compressive	
4	Epoxy Surface Outer	98.431	101	Tensile	

Table 2 Variation in force and angles with respect to % of cycle time

Stage No	Percentage of Cycle Time	Force (N)	Angle in Front Plane in Degrees	Angle in Lateral Plane In Degrees
1	0	0	29.49	15.33
2	4	4600	28.64	13.43
3	7	5494	28.00	12.00
4	10	5300	27.36	10.58
5	15	3580	26.30	8.20
6	20	1970	25.24	5.83
7	25	900	24.18	3.45
8	27	785	23.75	2.50
9	30	1075	23.11	1.08
10	35	2250	22.05	-1.30
11	40	3800	20.99	-3.68
12	45	5250	19.93	-6.05
13	47	5494	19.50	-7.00
14	49	4750	19.07	-7.95
15	50	0	18.86	-8.43

Table 3 Maximum and minimum principal stresses at critical locations

Stage No	Stress in N/mm2							
	Max Principal Stress				Min Principal Stress			
	Surface Out	Surface In	Neck Out	Neck In	Surface Out	Surface In	Neck Out	Neck In
1	0	0	0	0	0	0	0	0
2	125.1	0.158	234.32	-0.628	-3.197	-209.09	1.055	-279.63
3	250.82	-0.761	492.06	-0.999	-4.448	-344.14	2.032	-458.09
4	230.15	-0.663	460.1	-0.899	-5.571	-338.85	2.018	-454.62
5	144.77	-0.202	278.68	-0.571	-3.111	-216.13	1.212	-277.5
6	108.36	-0.282	206.79	-0.429	-2.657	-158.78	1.089	-203.16
7	53.609	-0.118	103.62	-0.221	-1.326	-79.01	0.521	-101.71
8	33.738	-0.125	64.745	-0.15	-0.784	-48.9	0.508	-63.345
9	64.168	0.181	123.01	-0.243	-1.538	-94.279	0.605	-120.48
10	112.24	0.11	202.59	-0.414	-2.659	-165.26	1.072	-204.47
11	159.79	0.024	289.42	-0.638	-3.988	-246.13	1.544	-293.34
12	209.43	-1.616	386.02	-0.792	-5.149	-316.07	2.275	-378.04
13	230.2	0.539	413.08	-1.122	-5.57	-348.37	2.3	-412.38
14	151.1	-0.092	277.13	-0.527	-3.01	-229.23	1.541	-268.04
15	0	0	0	0	0	0	0	0

Table 4 Absolute stresses developed at critical locations

Stage No	Percentage of Cycle Time	Absolute Stress Value N/mm2			
		Surface Out (Tensile)	Surface In (Compressive)	Neck Out (Tensile)	Neck In (Compressive)
1	0	0	0	0	0
2	4	125.1	209.09	234.32	279.63
3	7	250.82	344.14	492.06	458.09
4	10	230.15	338.85	460.1	454.62
5	15	144.77	216.13	278.68	277.5
6	20	108.36	158.78	206.79	203.16
7	25	53.609	79.01	103.62	101.71
8	27	33.738	48.9	64.745	63.345
9	30	64.168	94.279	123.01	120.48
10	35	112.24	165.26	202.59	204.47
11	40	159.79	246.13	289.42	293.34
12	45	209.43	316.07	386.02	378.04
13	47	230.2	348.37	413.08	412.38
14	49	151.1	229.23	277.13	268.04
15	50	0	0	0	0

Table 5 Amplitude, Mean and Induced stresses at critical locations

Sr No	Location	Amplitude Stress (N/mm^2)	Mean Stress (N/mm^2)	Induced Stress (N/mm^2)
1	Surface Out	125.41	125.41	143.393
2	Surface In	174.185	-174.185	184.345
3	Neck Out	246.03	246.03	326.313
4	Neck In	229.045	-229.045	186.360