IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN(e) : 2278-1684, *ISSN(p) :* 2320–334X, *PP : 07-14*
www.iosrjournals.org

Failure Analysis of Blister Packaging Machine Cam Shaft

J. D. Chougule[1], R. G. Todkar[2]
[1]*(Department of Mechanical Engineering, Rajaram Bapu Institute of Technology, Sakharale, India)*
[2]*(Department of Mechanical Engineering, A. D. College of Engineering and Technology, Ashta, India)*

ABSTRACT : *This paper presents a failure analysis of a cam shaft in the transmission system of a blister packaging machine used for packaging of tablets. It is observed that the cam shaft fails due fatigue loading into two pieces during operation. Visual examination of the fractured surface clearly shows cracks initiated from the roots of shaft step and the number of cycles till failure is shown on the machine PLC screen. To find out the cause of failure, a finite element analysis is carried out. Results of stress analysis reveal that the highest stressed area coincides with the fractured regions of the failure of the shaft. The theoretical stress fairly matches with the sub-model stress values. The failure analysis shows that the fatigue failure of the shaft is due to weak section at the step provided for cam shaft mount. To enhance service durability of the transmission system of Blister Packing Machine few suggestions are presented.*

Keywords – *Blister Packaging Machine, Cam shaft, Failure Analysis, shaft failure.*

I. INTRODUCTION

Tablets and ampoules are lifeline of human beings, and obviously even our pets too. Blister packaging machines are used in pharmaceutical industries for packing of tablets. These machines are multiple inputs, single output. Basically this machine constitutes of forming, feeding, sealing, perforation and punching stations. These stations are driven by a motor, One common driveshaft transfers rotary motion from motors to camshafts of the stations through Bevel gear attachment. Then cam and cam-follower operates the respective tools required for various operations like forming, sealing, and perforation and punching. In these stations of the machine, camshaft rotates and tool moves up and down to perform above said operations. These power transmission shafts (camshaft) continuously subjected to alternating tension and compression due to bending. Those cyclic stresses acting on shaft are much more severe than static stress of the same magnitude. The working life cycle of Blister packaging machine is affected due to these cyclic stresses acting on camshaft.

The concepts of possible reasons of automobile diesel engine crank shaft failure i. e. operating sources, mechanical sources and repairing sources has been presented in detail in [1] and it observed that the crank shaft failure usually occurs due to small cracks developed as an effect of thermal fatigue loading substantial overheating during shaft grinding process. A power shaft design method which accounts for variable amplitude loading histories and their effects on limited life design requirements considering the effects of combined bending &torsion loading and number of service factors is explained in [2]. The factors like surface roughness, stress concentration, residual stresses, corrosion resistance etc can be used to modify the fatigue strength. In [3] an analysis technique is explain to assess the fatigue life of a shell structure under variable loadings using finite element analysis technique for simulation works and it is seen that constant amplitude predict a life larger as long as one predicted by variable amplitude tests. A comparison of material test data with various approaches to estimate effect of mean stress on stress life and strain life behavior is explained in [4] and seen that the walkers mean stress equation gives superior results. In [5] fatigue analysis of a connecting rod is performed to evaluate critical point calculated stresses and displacements under maximum compression and tension loadings. A crack modeling approach for the prediction of fatigue failure of camshaft is introduced in [6] which allow the calculation of an equivalent stress intensity factor enabling standard fracture mechanics methodology. In [7] the analysis of failure reasons of a camshaft of automobile engine is carried out by using scanning electron microscopy and chemical analysis of the fractured camshaft material to assess the reason of fracture, whereas in [8] failure analysis of a hollow power transmission shaft is presented by performing stereo-binocular, scanning electron microscope examination.

Thus, in this paper, existing camshaft design study is carried out based on the experimental reading. Experimental results and analytical calculations are validated with stress analysis. Stress analysis is performed with virtual engineering code "ABAQUS" and "Sub-Modeling" technique is used for the local failure region study. Design changes are suggested and comparison between design revisions is demonstrated.

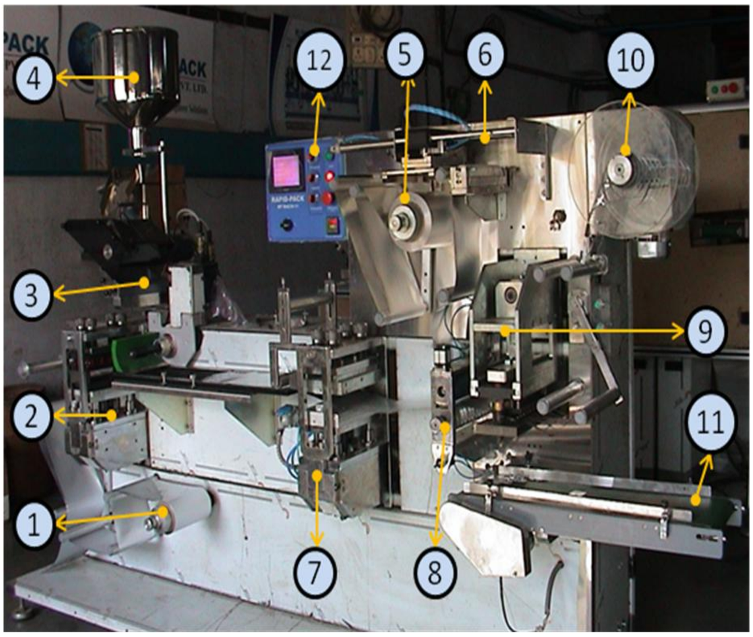
II. PROBLEM DEFINITION AND OBJECTIVES

In this work camshaft under consideration operates forming die of a blister forming station at 45 cycles per minute. When forming die is in closed condition, operating load of the assembly is maximum i.e.19828N, and in open condition it takes only dead load i.e. 266N in opposite direction. Thus, camshaft is subjected to alternate tension and compression due to bending loads. The shaft fails after few cycles wherein PLC screen shows the number of cycles completed till failure.

The objective of this work was to modify the shaft to withstand such an operational alternating loads which intern provides a high service warranty to the customer. While validation with finite element analysis (FEA) it is found that the stress is higher at the step, where failure was found in field. The stress contour on fillet in the failure region was not smooth; the reason was coarsen mesh at the geometric discontinuity. Increasing number of elements on the fillet would increase the size of the problem, which could rise the solution time. To overcome this problem sub-modeling technique in ABAQUS is used. Sub-modeling is the technique of studying a local part of a model with a refined mesh, based on interpolation of the solution from an initial, global model onto appropriate parts of the boundary of the sub-model. The method is most useful when it is necessary to obtain an accurate, detailed solution in the local region and the detailed modeling of that local region has negligible effect on the overall solution. The response at the boundary of the local region is defined by the solution for the global model and it, together with any loads applied to the local region, determines the solution in the sub-model. The technique relies on the global model defining this sub-model boundary response with sufficient accuracy.

III. EXPERIMENTAL SETUP AND READINGS

The Fig. 1 shows experimental set-up of a blister packaging machine.



1- Base Foil Assembly
2- Forming Station
3- Feeding Unit
4- Hopper
5- Sealing Foil
6- Batch code Printer
7- Sealing Station
8- Indexing Unit
9- Punching Station
10- Trim Winder
11- Outlet conveyor
12- PLC screen

Fig. 1: Experimental Set-up

When machine starts aluminum or PVC (Poly-vinyl chloride) foil is forwarded to forming assembly through set of guide rollers. The web of formed blister pockets at forming assembly moves ahead to feeding station. Same time tablets in hopper flows to feeding unit, as blister pocket web moves forward tablets get filled in it. Simultaneously batch code Printer prints batch code on sealing foil, and then guided to sealing station through guide rollers. Guide track guides web of filled pockets to sealing station for sealing the foil. Indexing unit drives sealed pockets to punching station, where blisters get sheared off from the web. At the same time, waste trim gets winded at trim winder unit. After cutting station, blister fall on conveyor and gets in to the next process i.e. cartooning machine. PLC (Programmable Logic Control) is used to provide inputs to the machine, to monitor all function of machine and to check the outputs like total number of blister packs.

The Fig 2 shows PLC screen reading photograph from the baseline production machine. The reading shows total number of packs, the final life till failure was calculated from number of total packs. From the broken shaft (see Fig 3-b) it is observed that the shaft is failed at the step, a high geometrical discontinuity and small fillet radius was a stress raiser. The shaft failed at the cam mount, where a maximum bending moment was produced by the transmitted force.

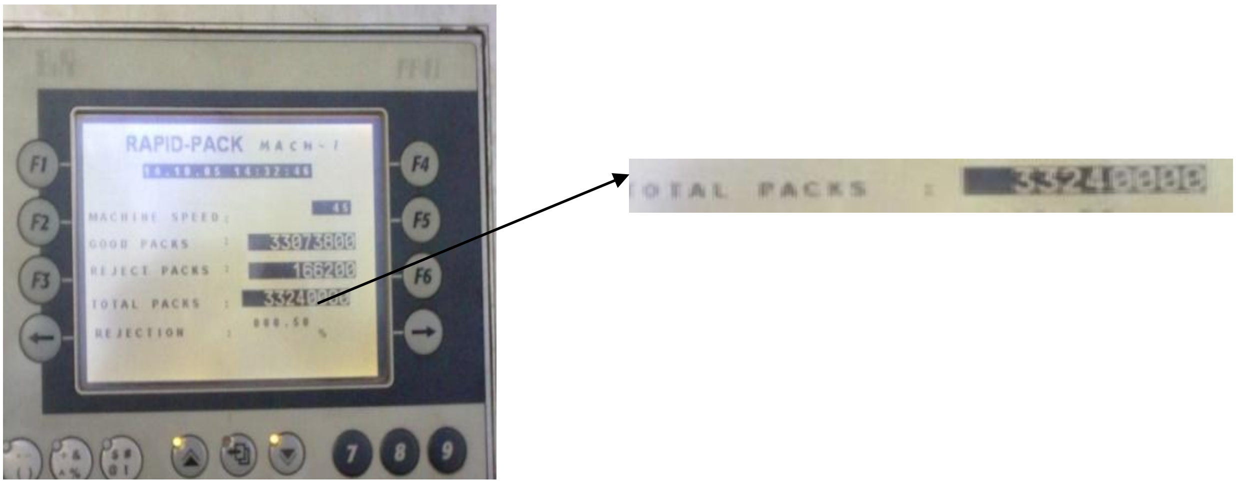
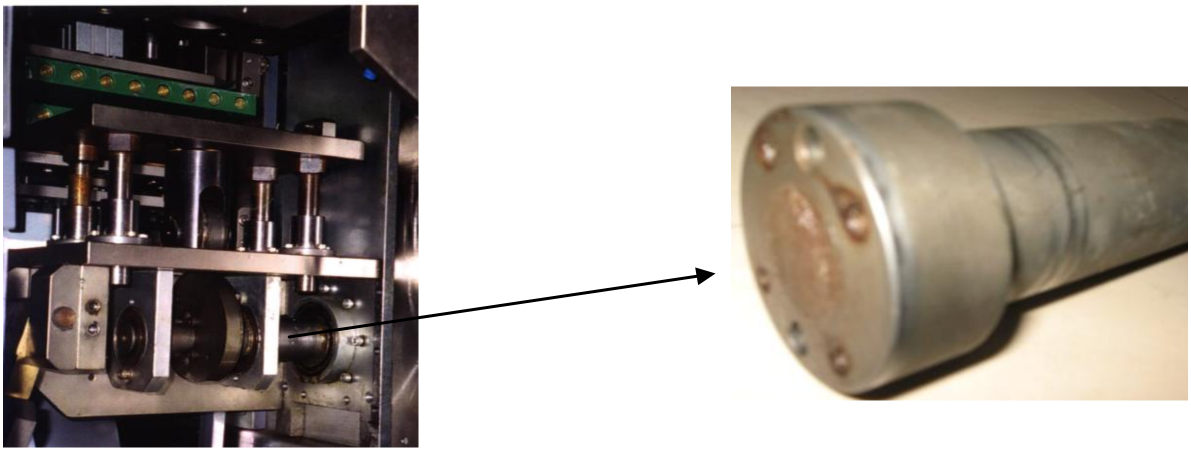


Fig. 2: PLC screen reading at the time of failure



a) Forming Station Assembly b) Failed Shaft

Fig. 3: Forming assembly and a failed shaft

The Fig 3(a) shows photograph of the forming assembly. The source of load was Top and Bottom mounting, cooling plates, both side tools reactions and force exerted by coil and disc springs. The fracture surface was vertical to the axial direction of the shaft. Detailed observations showed that cracks were originated at the step towards the small diameter of shaft. Similarly the finite element result showed higher stress at same location. This suggests that the smaller side step of the shaft is region of stress concentration and is a preferential site to initiate failure after 33240000 blister packs. As machine punches out 4 packs per cycle, actual cycles at failure equals to 8310000Cycles. The machine is running at 45rpm, considering 12 working hours per day, total number of days till failure comes to be ~257Days.

IV. ANALYTICAL APPROACH

In this study, failure analysis of a cam shaft of a blister packaging machine is presented using finite element approach through sub-modeling technique in “Abaqus” and verified with theoretical and experimental investigation. Fig 4(a) shows a typical camshaft region of a typical forming assembly under study. All components of forming assembly are mounted on retainer plate. The horizontal support plate is fixed on back plate and supported from bottom by a stiffener plate. The cam is mounted on shaft which intern is supported with bearings at main housing, i-support and o-support. The shaft is driven by a drive motor through bevel gear arrangement.

Using the basic theory, the radial load from the bevel gear assembly is calculated as P2=9099N. While shaft rotates, cam follower follows the cam profile. When cam is at bottom position forming tool gets opened, and it carries only dead weight of bottom die components and their mountings i.e. P1min = -266N. Going forward, when shaft completes one rotation and reaches to its top position, the forming tool closes and gets compressed further to form a shape of blister pocket. At top position, cam carries load of bottom die, top tool and load exerted due to spring compression, therefore total load comes to be P1max =19828. The Fig 4(b) shows the typical shaft, its support system and loads as discussed above.

Fig 4(c) shows a simplified schematic representation of loading and support conditions of the shaft, where B1, B2, B3 and B4 are supports, P1Max, P1Min and P2 are loads. From the block representation it seems that the beam is statistically indeterminate, hence the reactions at supports are calculated by Three Moment Theorem. From the results of finite element analysis stress is maximum at a distance 58.74 mm from the support B1, and this is the region of failure in field (step-F, see Fig4-c).

Stress concentration factor Kt is calculated using Peterson’s stress concentration formulae .

Bending Moment at F

M= 6357.27 X 58.74 = 373426.04 N-mm

σ_{Nom}= 32 M/π(d)3 = 32 X 373426.04 / 3.141(30)3 = 140.90 N/mm2

$K_t = \sigma_{Max} / \sigma_{Nom}$ {....Kt= 3.573 (calculated)}

σ_{Max} = 503.5 N/mm2(Tensile)

Similarly when tool is open and tool load will be -266N and the stress at the same point is

σ_{Min} = -11.20N/mm2 (Compressive)

$$\sigma_{ar} = \sigma_a \left(\frac{2}{1-R} \right)^{1-\gamma}$$

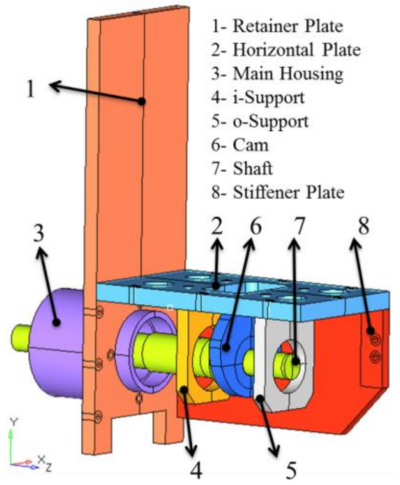
σ_{ar} = 325.48

Life calculations by Walker’s Approach Putting in stress life equation

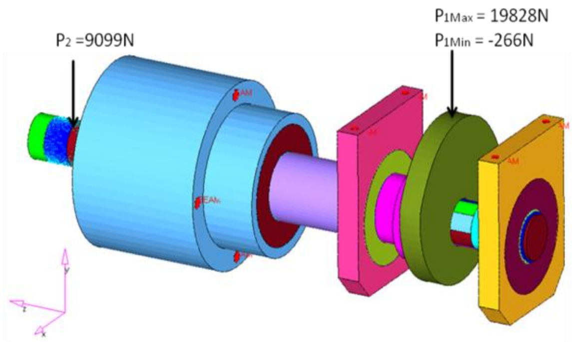
N_f = (1/2) * (σ_{ar} / σ'_f)1/b,

Thus life till failure is N_f = 7.50E+06Cycles.

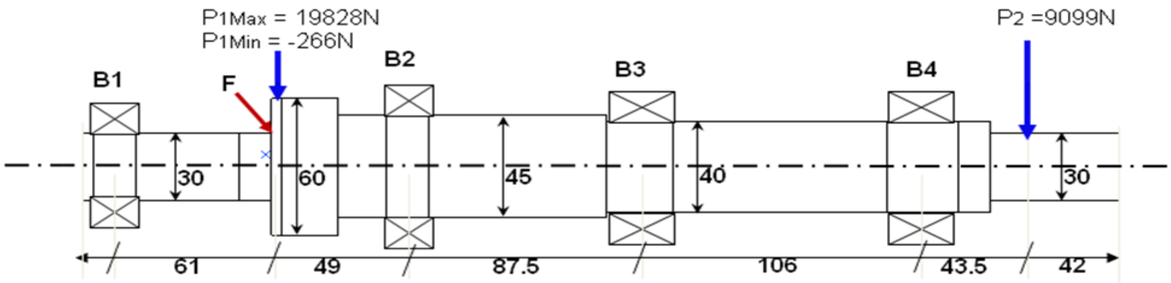
Let us consider 12 working hours per day, and then total number of hour for calculated cycles comes to ~232Days.



a) Forming assembly detail view



b) Cam shaft showing loading points and supports



(c): Block diagram Of Shaft (All dimensions are in mm)

Fig 4: Geometrical representation forming assembly, shaft loads with support system and simplified block diagram

V. FINITE ELEMENT ANALYSIS

Fig 5 shows FEA model of cam shaft assembly with loads and boundary conditions. The CAD model of shaft assembly is generated using I-deas, and preprocessing is done in hyper mesh. The CAD model in “iges” format was imported in hyper mesh. The shaft bearings, housing, support bracket and cam are meshed with second order tetrahedral elements and element type is chosen to be C3D10M. Global element size is taken as ~3mm.The fillet radius at step-F (Fig4-c) is captured with three elements. Bolts are represented with rigid elements. Contact is defined between bearings, shaft, support brackets and housing.

The master nodes of rigid bolt locations are fixed in all directions. A load of 19828N is distributed through the node set on the cam top, and radial load of bevel gear i.e. 9099N is applied on the master node of rear gear rigid. Fig 6 (a) shows stresses induced in shaft due to the loads applied. The stress is high at the base of step-F. From the stress plot it is clear that further mesh refinement is needed to normalize the drastic stress variation within an element. The sub-modeling of the local stress concentration region is done. A sub-model is generated in ABAQUS with 16 elements on the fillet. Fig 6 (b) shows the results for sub-modeling analysis.

With sub-model analysis maximum value of Max-Principal stress on the shaft is 498Mpa (tensile), and minimum is -11MPa (Compressive). The life calculated by walker approach using the stress values from FE analysis, is given as Nf = 8.48E+06Cycles. And total numbers of days till failure are calculated to be 262Days.

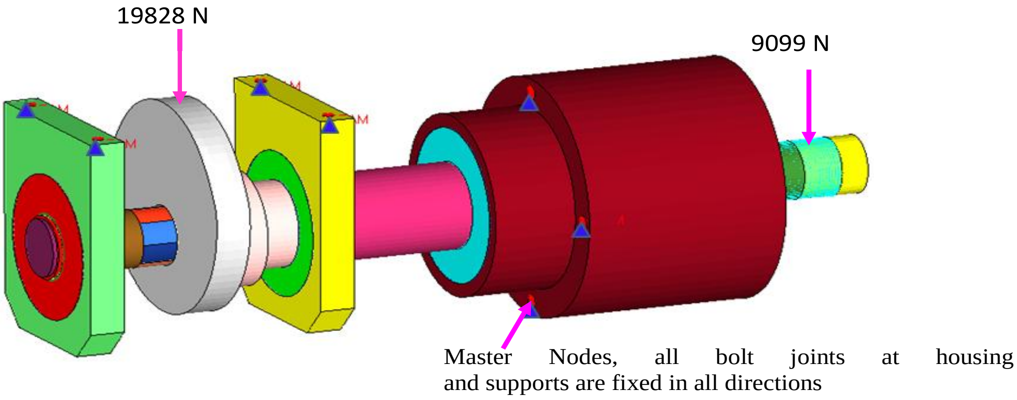
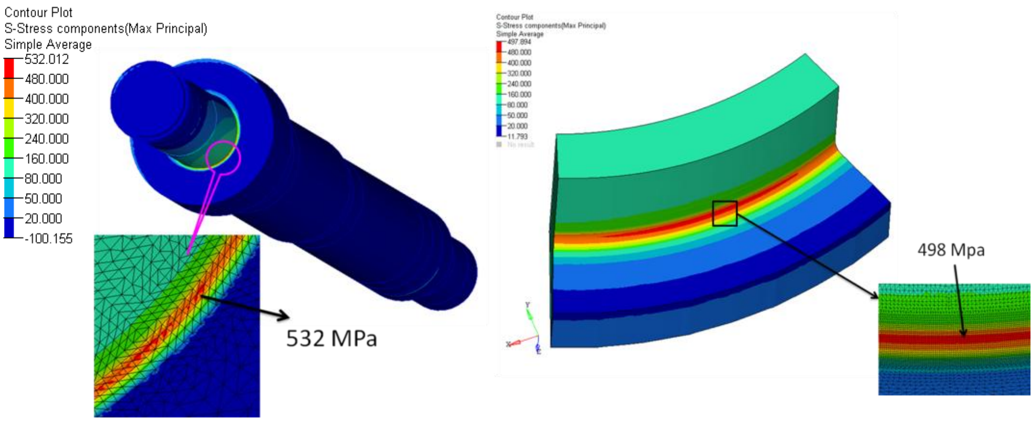


Fig 5: FE model of system, loads and boundary conditions applied



(a) Result in global model (b) Sub-model with 16 elements on fillet

Fig 6: Max-Principal Stress Plot for shaft-global model and sub-model

VI. COMPARISON OF RESULTS

Table 1 shows comparison of life cycle determined using i) Theoretical approach , ii), Experimental investigation approach and ii) Finite element analysis approach, for all results have been shown for baseline model (Fig 4). The machine was run for 12 working hours per day at 45 rpm till failure of the cam shaft. It is seen that Experimental Approach provides the actual life of 257 days where as Finite Element Analysis Approach shows life of 262 days, based on this comparison decision was made to perform the further iteration studies with finite element approach.

Table 1: Comparison of results of baseline design with various approaches

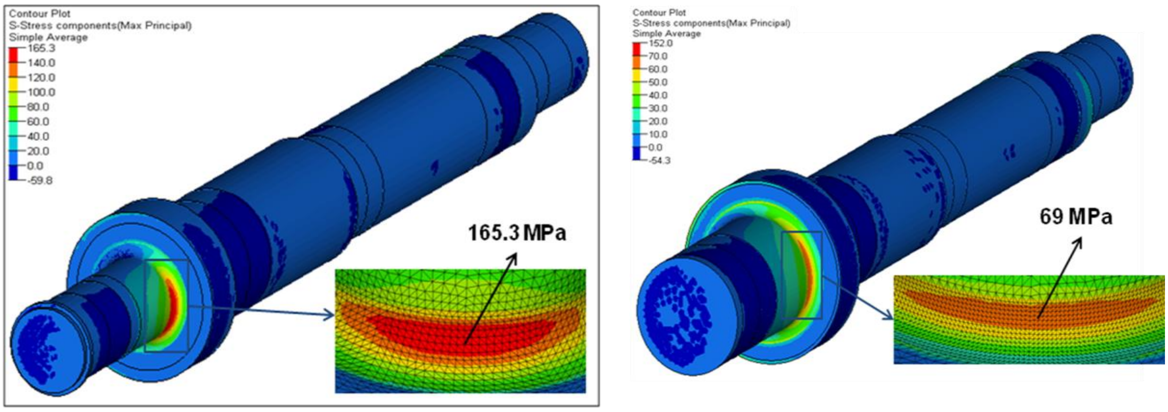
Sr. No.	Experimental Approach	Theoretical Approach	Finite Element Analysis Approach	Out Comes
1	8.31E+06 Cycles	7.5 E+06 Cycles	8.48E+06 Cycles	Number of Cycles till failure
2	257 Days	232 Days	262 Days	Life in number of days (12 working hrs per day)

VII. DESIGN REVISION

The objective of the design review is to increase the life of the shaft. Based on the studies there are some alternatives to minimize the stresses on the shaft:

1) Relaxing the fillet size in the stress concentration region,
2) Changing the material which can give better life estimation, and
3) Changing the size of the shaft

As the stresses in the base design are higher in the region of stress concentration, where the shaft step has sudden geometric changes with no fillet relaxation(at the step-F, Fig4-c), decision was made to increase the fillet radius from nearly sharp edge to 4.5 mm (Design revision-1), providing chamfer on the cam for easy accommodation in the assembly. On the top of design revision-1, the analysis for second design revision was performed by increasing minor diameter at step-F to 45 mm and major diameter to 70mm, which was 30mm and 60mm simultaneously in baseline design (please see Fig4-c).



(a) Stress Plot for Design Revision-1 (b) Stress Plot for Design Revision-2

Fig. 7: Stress Plot for Design Revisions

The stress plot in Fig 4 (a) shows the maximum stress value σmax of 165.3 MPa for design revision-1 in the stress concentration region; whereas Fig 7 (b) shows σmax of 69 MPa for design revision-2. Thus design revision-2 shows 86% reduction of σmax over baseline design, where baseline was showing 498MPa (see Fig6-b). Design revision-2 shows better improvement, and hence is the acceptable design modification for the cam shaft.

Table 2: Comparison of results for design revisions

	Minor Diameter	Major Diameter	Fillet Radius	Maximum Stress	Number of Life Cycles	Number of Days
Baseline Design	30	60	0.5	498	8.48 E6	262
Design Revision-1	30	60	4.5	165	1.71 E12	5.29 E7
Design Revision-2	45	70	4.5	69	3.08 E16	9.52 E11

VIII. CONCLUSION

A failure analysis of a blister packaging machine cam shaft has been presented using finite element analysis and fatigue failure considerations. The major findings are as follows...

- All three studies indicated the stress concentration at cam step region is reason of failure
- The use of sub-modeling technique with "ABAQUS" solver can save lot of computation time and gives accurate results in the region of interest.
- To design a cam shaft with high service durability, mechanical characteristics such as fatigue strength,

ultimate tensile strength, and fracture toughness are important properties that should be considered in material selection at design stage.

- The suggested design of shaft is with high service durability (better warranty to customer) and there are hardly few components needed to be replaced to adopt the design.
- The shaft is manufactured and assembled on production machines, and running successfully without any failure.

REFERENCES

[1] F. S. Silva, An investigation in to mechanism of a crank shaft failure, *Key Engineering Materials, Vol 245-246, 2003*.

[2] Stuart H. Loewenthal, Factors That Affect the Fatigue Strength of Power Transmission Shafting and Their Impact on Design, *International power transmission and gearing conference by ASME 8-12 October 1984*

[3] S.Abdullah, S.M. Beden, A.K. Ariffin and M.M. Rahaman, Fatigue Life Assessment for metallic structure: A case study of shell structure under Variable Amplitude Loading, *Journal of Applied Sciences 2008.*

[4] Norman E. Dowling, Mean Stress Effects in Stress-Life and Strain-Life Fatigue, *Society of Automotive Engineers, F 2004/51.*

[5] M. Omid, S.S. Mohtasebi, S.A. Mireei and E Mahmoodi, Fatigue Analysis Of connecting Rod of U950 Tractor in the Finite Element code Ansys, *Journal of Applied Sciences 2008.*

[6] G. Wang, D. Taylor, B. Bouquin, J. Devlukia, A. Ciepalowicz, Prediction of fatigue failure in a camshaft using the crack modelling method, *Engineering Failure Analysis, Volume 7, Issue 3, 1, Pages 189-197, June 2000.*

[7] H. Bayrakceken,I Ucun, S Tasgetiren, Fracture analysis of a camshaft made from nodular cast iron, *Engineering Failure Analysis Volume 13, Issue 8, Pages 1240–1245, December 2006.*

[8] S.K. Bhaumik, R. Rangaraju, M.A. Parameswara, M.A. Venkataswamy,T.A. Bhaskaran, R.V. Krishnan, Fatigue failure of a hollow power transmission shaft, *Engineering Failure Analysis, Volume 9, Pages 457-467, 2002.*

NOMENCLATURE

P_{1max} : Maximum forming Load, when tool is in closed condition
P_{1min} : Minimum load, when tool is in open condition
P_2 : Radial load through bevel gear
M : Bending moment at point F
σ_{nom} : Nominal Stress
σ_{max} : Maximum Stress
K_t : Stress concentration Factor = 3.573
σ_{min} : Minimum Stress
σ_m : Mean Stress
σ_a : Alternating Stress
σ_{ar} : Stress variable
γ : Walker’s Fitting Constant = 0.65
σ_f' : Fatigue Strength Coefficient =1440 MPa
b : Fatigue Strength Exponent= -0.9
R : Is the ratio $\sigma_{min} / \sigma_{max}$
N_f : Cycles needed for fatigue failure