Defence Science Journal, Vol. 57, No. 2, March 2007, pp. 165-171

Finite Element Analysis of Squirrel Cage Ball Bearings for Gas Turbine Engines

Benny George[1] and N. Siva Prasad[2]

[1]*Gas Turbine Research Establishment, Bangalore–560 093*
[2]*Indian Institute of Technology Madras, Chennai–600 036*

ABSTRACT

Squirrel cage ball bearings are used in recent aero engines to overcome the vibration and stability problems associated with rotor systems supported by conventional bearings. The critical design feature of squirrel cage bearing is to provide flexible support to rotor system. The outer ring of the bearing is configured such that it acts as a flexible bearing support, and hence, weight is minimised. The bearing is mounted directly on a rigid intermediate casing of a gas turbine engine. Finite element analysis helps to obtain a suitable geometry at the design stage. A 3-D model of the bearing is created using IDEAS software. Finite element analysis using contact element has been carried out for finding the deformation and stresses in the webs at various locations using ANSYS software. The deformations are compared with those of experimental values for an axial load. The deformation in the webs is used to check the pressure balance in the compressor and the load acting on the bearing. Finally, a procedure for doing the finite element analysis of ball bearings for combined axial and radial load is laid down.

Keywords: Ball bearing, contact analysis, gas turbine engine, finite element analysis, squirrel cage ball bearings, deformation

1. INTRODUCTION

A squirrel cage ball bearing is used to support a shaft on high pressure side of the compressor in an axial flow gas turbine engine. The configuration of the outer ring of the bearing accommodates one row of ball bearing on one end and the other end has a flange with holes for mounting the bearing to intermediate casing of the gas turbine engine. Flexibility is introduced in the bearing by suitable geometry of connecting webs between the two ends. The residual unbalance mass left in the rotors may lead to vibration and lower fatigue life. The geometry of the bearing and its support play an important role in determining the dynamic characteristics such as critical speed, vibration response, and stability of the rotor. Squirrel cage ball bearing helps in reducing the vibration levels and improves the stability, to some extent, in rotor support systems.

The strain induced in the webs during engine run gives necessary information regarding pressure balance in the compressor and the load acting on the bearing. With deep groove ball bearing, measurement of strain becomes very difficult as the whole bearing is mounted within the housing. The webs on the squirrel cage ball bearing enable easy measurement of strain.

Before assembly of the bearing on the engine, calibration of strain gauges fixed on webs is carried

Revised 02 January 2006

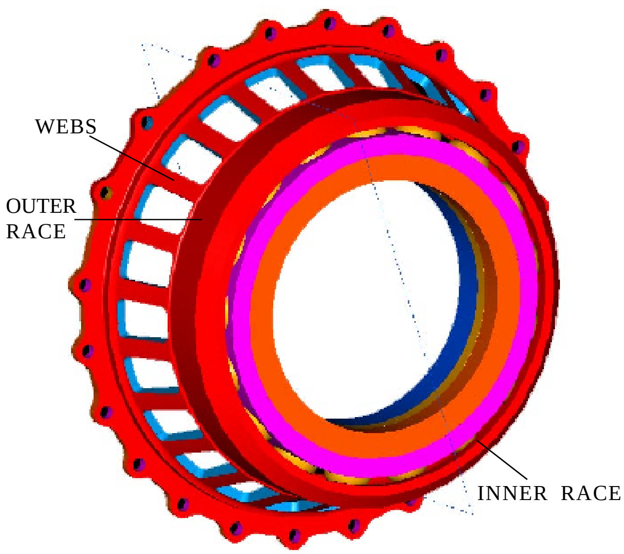

Figure 1. Squirrel cage ball bearing.

out in a rig. Axial load corresponding to maximum thrust developed by the engine is applied in the rig and corresponding strain values are measured. The direction of deformation of the webs is the subject of interest in this study. By virtue of the configuration and using strength of material approach, one gets the result that the webs are expected to bend inwards under the application of an axial load, but on the contrary in actual experimental work, it was found to deform outwards. To analyse the reason for the above phenomenon in the webs, which affect the stresses developed in the bearing under combined radial and axial loads, this study was undertaken.

The solid model of the bearing, created using IDEAS is shown in Fig.1. The geometry of the bearing and load transfer between the elements of bearing is complex in nature. In addition, friction is involved between the elements; hence close form solutions are not available for solving the problem. A finite element analysis was carried out to estimate the deformation of webs and stresses developed, and compared with the experimental values under the application of an axial load of 22 kN. The ANSYS has an advanced contact wizard, which can address the present problem. Wizard is a window within ANSYS, which helps the user to give various input required to solve the contact problems.

2. EXPERIMENTAL ANALYSIS

An experimental rig as shown in Fig. 2 was used for calibration of strain gauge. Metallic strain gauges of foil-type (KARMA Alloy) with a gauge factor of 2 were fixed on the webs. The circuit used was either full bridge or quarter bridge. The maximum strain that could be measured with the strain gauge was 2500 micro strain. The strain gauge was fixed on the webs using epoxy-based M-Bond 610 and cured in the oven at 125 °C for 2 h.

The bearing was mounted on a rigid structure as shown in Fig. 2. A hydraulic actuator was used to apply axial load on the bearing. A mandrel was used as the connecting member between the bearing

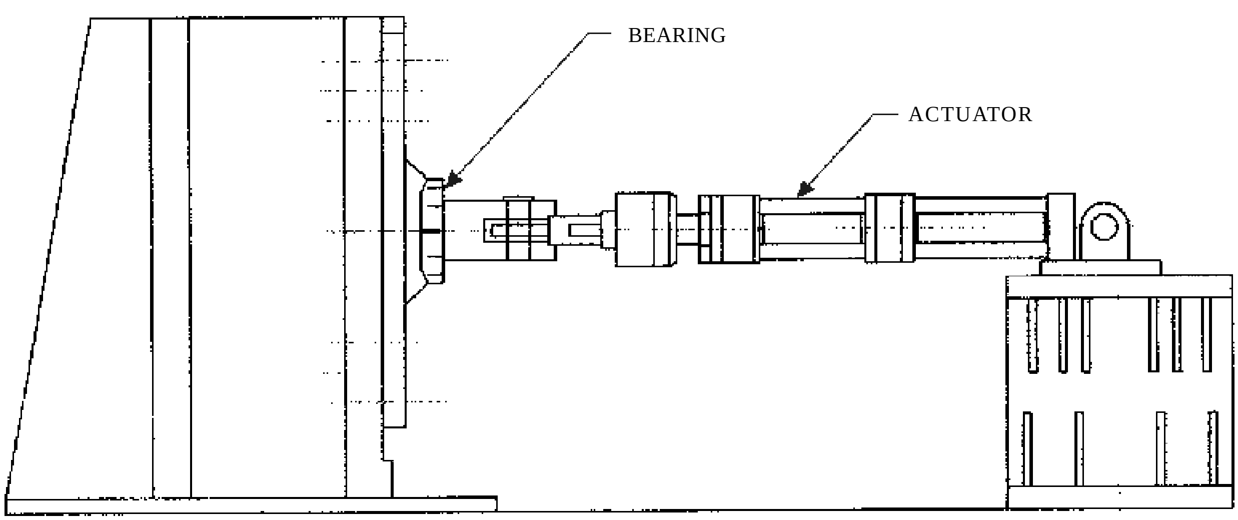

Figure 2. Experimental rig setup.

and the actuator. Force up to 100 kN can be applied through the actuator. The bearing, mandrel and actuator piston were aligned within an accuracy of 50 μ.

During calibration of the strain gauge, it was observed that webs were deforming outwards wrt horizontal axis, rather than expected inwards bending. As the webs were inclined towards the centre of the bearing by geometry, it was expected to bend inwards under the application of an axial load. This lead to some ambiguity in the calibration procedure. Hence, a finite element analysis was done to confirm this aspect. A finite element analysis using contact elements was carried out to cross check the deformation characteristics of the webs. As the load was transferred to the outer ring, and thereby to the webs, by contact between the balls and the inner race, usage of contact elements was nearer to the actual conditions.

3. FINITE ELEMENT ANALYSIS

A finite element model with inner and outer races, and balls used for contact analysis is shown in Fig. 3. The modelling of contact conditions in the finite element technique is involved. A contact and a target surface define a contact pair as shown in Fig. 4(a). The parameters, which highly influence the result of contact analysis, are initial contact condition, normal contact stiffness, contact-opening stiffness, separation of contact surface during analysis, location of point of contact, and coefficient of friction.

Each of the above parameters was carefully selected to reduce errors in the analysis. The coefficient of friction was taken as 0.005 as the elements maintain rolling contact. In analysing contact problems, it is essential to specify stiffness between the contact surfaces. The amount of penetration between the two surfaces during the analysis depends upon this stiffness. The normal contact stiffness is taken 10 times the value of Young's modulus as raceways are hardened and ground[2]. Initial contact condition is a very important parameter upon which results of the analysis depend. Even though the geometry was created in just-touch condition, due to meshing and other geometrical surface irregularities, either an initial gap or penetration was formed at the

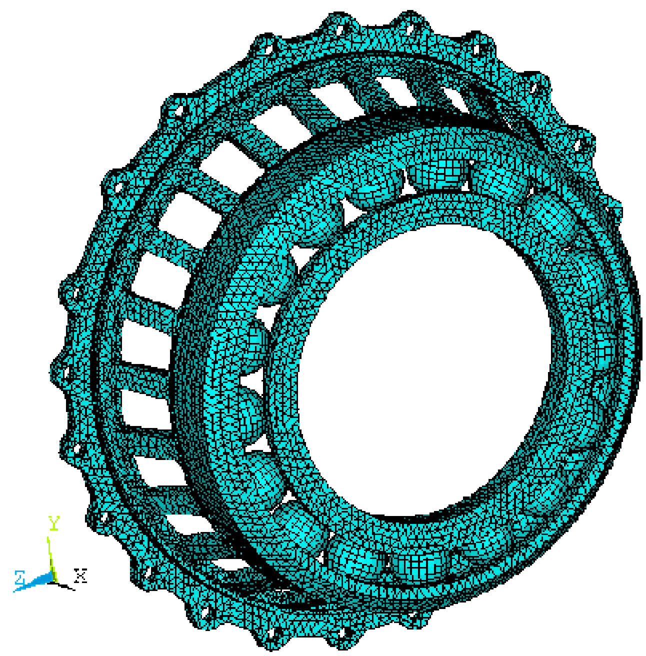

Figure 3. Finite element model of the bearing.

contact surface. Care was taken to reduce this initial gap or penetration by changing the constants like contact surface offset (CNOF), initial contact closure (ICONT), pinball region, and p_{max}/p_{min} of target surface. It was found that the adjustment of any two parameters at a time yields faster results. For example these parameters could be a combination of contact surface offset and p_{max}/p_{min}. Parameter contact surface offset shifts the entire contact surface with the distance value of contact surface offset to bring it in contact with the target surface. When initial contact closure was specified, it moved all initially open contact points, which were inside of adjustment band initial contact closure onto the target surface.

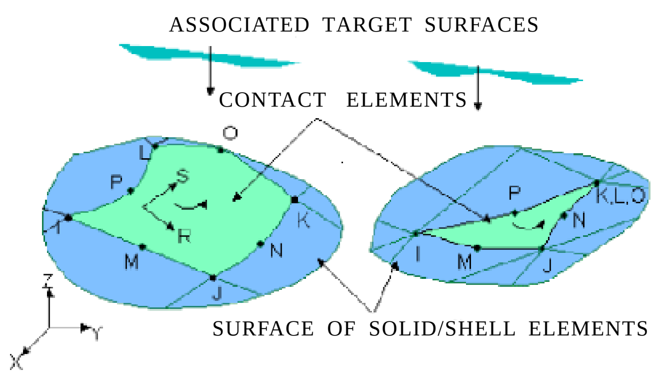

Figure 4 (a). Surface-to-surface contact element (8 nodes).

The constants p_{max} and p_{min} are used to specify an initial allowable penetration range. These constants ensure that proper contact is maintained between two surfaces at the start of analysis. When either p_{max} or p_{min} was specified, ANSYS brought the target surface into a state of initial contact at the beginning of the analysis. If the initial penetration was larger than p_{max}, the target surface was moved to reduce the penetration. If initial penetration was smaller than p_{min}, the target surface was adjusted to ensure initial contact.

The elements solid 45, conta 174 and targe 170 available in ANSYS software were used in the analysis. All the the three elements have 3-DOFs, namely displacements in x, y, z directions. Solid 45 is a tetrahedron element. Both, conta 174 and target 170, have 8 nodes each and are better suited for the contact of curved surfaces compared to other contact elements. Conta174 was used to represent contact and sliding between 3-D target surface (targe170) and a deformable surface. It had the same geometrical characteristics as the connected face of the solid element. The 8-noded conta 174 would degenerate to a 6-noded element out of which 3 nodes will have connectivity with the underlying face nodes of solid 45 tetrahedron element.

Contact occurred when the element surface penetrated into the specified elements of a target surface. Both the elements were paired through a single real constant set. A surface-to -surface contact element is shown in Fig. 4(a). A contact pair between the ball and the outer race is shown in Fig. 4(b). The surface-to-surface elements method is well suited for the above application as it supports both lower-order and higher-order elements, provides better contact results, has no restriction on the shape of target surface (surface discontinuities can be due to discretisation or geometry of the objects). This element has rigid-to-flexible, flexible-to-flexible and surface-to-surface features. The target surface can be either rigid or flexible. If one surface is stiffer than the other, the softer surface should be the contact surface and the stiffer surface should be the target surface. Accordingly, raceway surface is treated as rigid surface (hard) and ball surface is taken as contact surface.

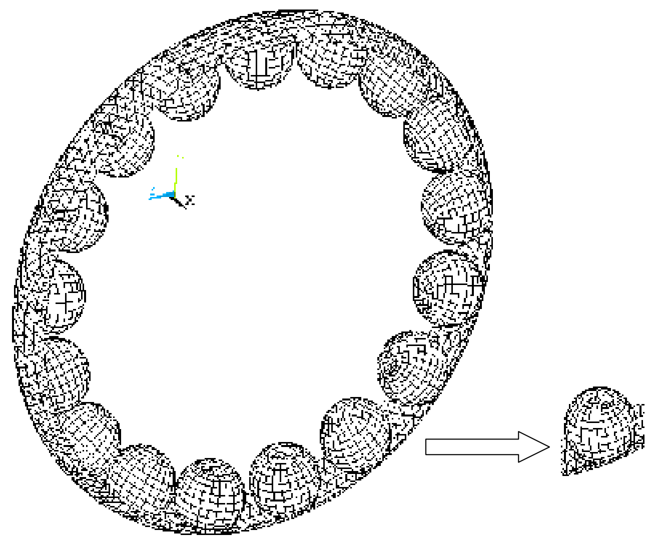

Figure 4(b). Contact pair between balls and outer race.

3.1 Method of Analysis

The contact analysis is highly nonlinear due to the following reasons:

- *Change in status of element*: Normally in contact analysis, elements at some load levels establish and at some other load levels loose contact with each other in a largely unpredictable and abrupt manner, especially if sliding is permitted. This leads to change in status of the element.
- *Frictional laws*: ANSYS uses Coulomb law of friction. All frictional laws are nonlinear in nature.

In ANSYS, geometric and material nonlinearities with change in status of elements are available. The ANSYS uses Newton-Raphson method to solve the nonlinear problems. In this method, the load is divided into a series of steps. Before each solution, this method evaluates the unbalanced force vector, which is the difference between the applied load and the restoring force. The restoring force is calculated based on the load corresponding to the element stresses. Using the unbalanced force vector, the program then performs a linear solution and checks for convergence. If convergence is not achieved, the unbalanced force vector is re-evaluated, the stiffness matrix is updated, and a new solution is found. This iteration is repeated till solution converges.

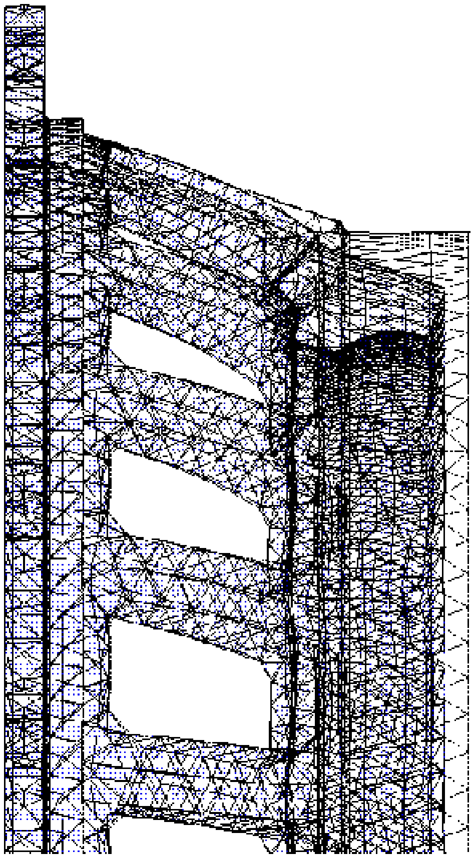

Figure 5. Deformation of webs in outer ring under an axial load of 22 kN without balls and inner race.

For surface-to-surface contact elements, ANSYS uses either the augmented Lagrangian method or the penalty method. The augmented Lagrangian method is an iterative series of penalty updates to find the exact Lagrangian multipliers, contact tractions. Compared to the penalty method, the augmented Lagrangian method usually leads to better conditioning of stiffness matrix and is less sensitive to the change in magnitude of the contact stiffness coefficient. If elements are highly distorted, augmented Lagrangian method may take more time for analysis, as additional iterations are required.

3.2 Analysis Procedure

The regular meshing of the model is done using solid45 elements. The surface of raceways and balls are meshed using conta174 and targe170 elements, respectively. The degree of freedom of flange is arrested in all six directions. Total numbers of elements is 90,000 with an element size of 3 mm and number of nodes is 28,000. An axial load of 22 kN is applied on the front face of inner ring similar to the load applied in the test rig.

4. RESULTS AND DISCUSSION

An axial load of 22 kN was applied on the outer ring alone in the first part of finite element analysis. The webs were found deforming inwards, ie, towards the axis as shown in Fig. 5, when the problem was analysed without taking contact of race and balls into account.

The deformation of web by the contact analysis is plotted in Fig. 6. Even though the load is applied axially in the initial condition, there is a change in the direction of load during analysis due to the change in element orientation. Surface pressures always acts normal to the element surface. With the deformation of the element, it will have an axial and a radial component. As point of contact changes, the direction of the radial component changes and this leads to an outwards bending of the webs.

The outer layer of web is subjected to compressive stress and inner layer is subjected to tensile stress. It is found that stress distribution is not uniform on all the webs. This might be due to the location of the balls not being symmetrical wrt all the webs

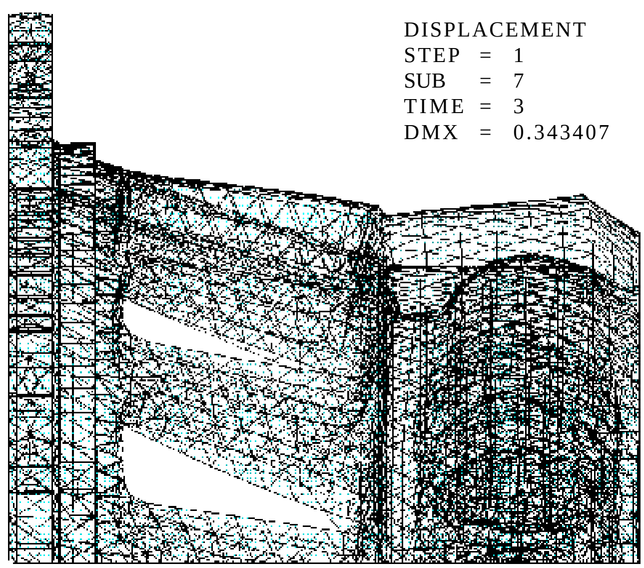

Figure 6. Outward deformation of webs in contact analysis.

in the finite element model, and in real model and surface irregularities in the real model, and thereby, variation in initial contact conditions. The reason for non-symmetry is that in the real model when the load is applied, the stress developed on each web depends upon the location of balls wrt webs, which cannot be controlled. The von-Mises stress for a distance of 3.6 mm on the middle of the web(46.0 mm from the front-end of the bearing) at 9 o' clock position is in the range 12 MPa to 18 MPa as shown in Fig. 7 and the average experimental value measured by the strain gauge at the same location is 14.5 MPa.

The stresses in the webs are not uniform in the case of experimental analysis also. It might be mainly due to the variation in symmetric location of webs and variation in the tolerances and dimensions of the balls.

A comparison of load versus von-Mises stress at the middle of the web between the experimental and the FEM analysis is shown in Fig. 8.

5. CONCLUSION

The above analysis shows that the problems of load transfer in case of ball bearings should be treated as a contact problem. As friction is involved and mechanism of load transfer is complex in nature, it is essential to do a contact analysis by considering the load acting on inner race, balls, and outer ring.

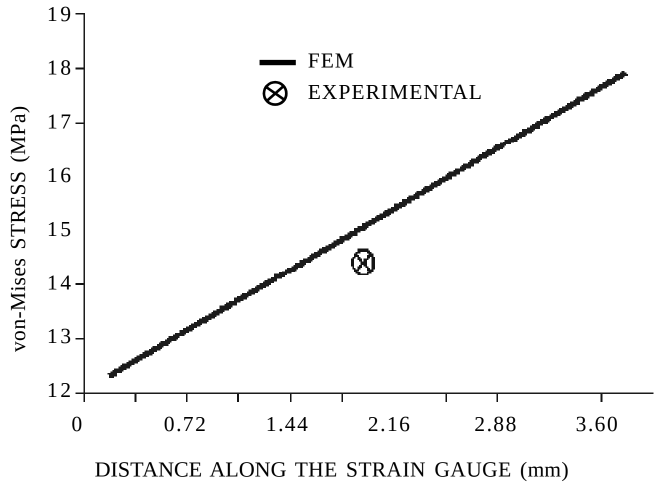

Figure 7. von-Mises stress distribution at 9 O'clock position measured at the middle of web for an axial load of 22 kN.

The consideration of load acting on the outer ring alone may lead to erroneous results. The finite element analysis results in the above case correlates well with the experimental values. The anomaly regarding the deformation of webs, which were found to be bending outwards during the experiment against the anticipated deformation of bending inwards is sorted out by FEM analysis. This is explained when contact analysis is employed. Once the stresses acting on the webs and their directions due to the axial load are known from contact analysis, equivalent forces and their directions on the bearing due to axial load can be calculated.

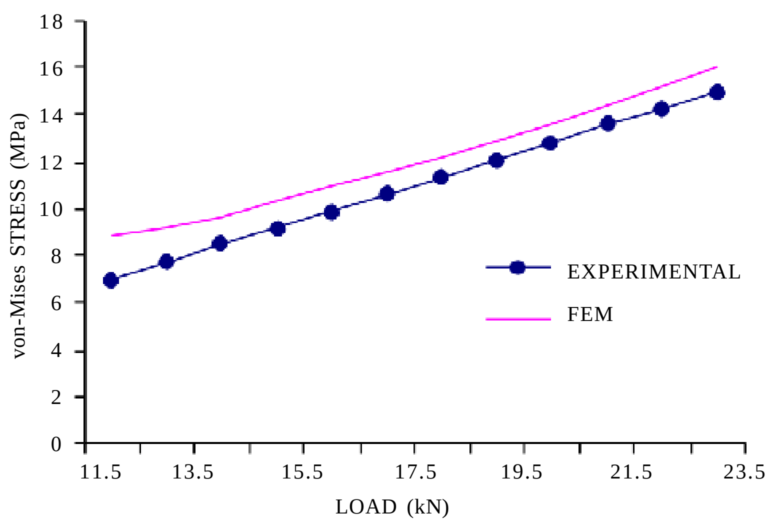

Figure 8. Variation of von-Mises stress at 9 O'clock position measured at the middle of web.

REFERENCES

1. Burguete, R.L. & Patterson, E.A. Comparison of numerical and experimental analysis for contact problems under normal and tangential load. *Proc. Inst. Mech. Engrs.*, 2001, **215** (Part G), 113–23.

2. Johnson, H. Principles of simulating contact between parts using ANSYS. Tech Papers. www.ansys.net/ansys.

3. Sheldon, Imaoka. Preventing rigid-body motion in contact problems. Tech Papers. www.ansys.net/ansys.

4. Arvid, Palmgren. Ball and roller bearing engineering. Ed. 3. S.H. Burbank & Co, Philadelphia, 1959.

5. ANSYS Release 5.6 ANSYS Structural analysis Guide, Ed. 5. ANSYS Inc, November 1999.

6. Bidwell, Joseph B. Rolling contact phenomena. Elsevier Publishing Co, New York, 1962.

7. Shigley, J.E.; Mischke, C.R. & Budynas, R. Mechanical Engineering Design, Ed. 6. McGraw-Hill International, New York, 2003.

8. George, Benny. Finite element analysis of squirrel cage ball bearing. Dept of Mechanical Engineering, Indian Institute of Technology Madras, Chennai, 2003. MTech Project Report.

Contributors

Mr Benny George graduated from the Institution of Engineers (India), Kolkatta, in 1993. He received his MTech from the Indian Institute of Technology Madras (IITM), Chennai, in 2003. He has been working in DRDO since 1989 and currently he is Scientist C at the Gas Turbine Research Establishment, Bangalore. His areas of interest include: Machine design, finite element analysis and instrumentation of gas turbine engines for aerospace applications.

Dr N. Siva Prasad graduated from the University of Mysore in 1975. He received his MTech and PhD, both from the IITM, Chennai, in 1977 and 1984, respectively. He has been working at the IITM since 1977, and currently, he is Professor in the Dept of Mechanical Engineering. His research areas include: Machine design, finite element analysis and computer-aided design.