Analysis of a Single Lap Screwed Joint Under Tension

A. Vineeth[1]
K. Abhishek[1]
MVNR. Saketh[1]
V. Narayanamurthy[2]

Abstract

Lap joints fastened by countersunk screws are commonly adopted in many engineering applications wherein the screw heads are flushed with the external surfaces of the joined structural members. These joints are subjected to various loadings such as tension, compression, bending, etc. This paper presents the finite element modelling and analysis of a typical screwed lap joint joining two plates with two countersunk screws and subjected to tension. The distribution of stresses along the screw, above and between the clamped members in the vicinity of the screws and the effect of pre tightening on the screws are explored. The joint stiffness, which plays a crucial role in the structural connections, are studied with and without the application of pre tension on the screws. The results of modal analyses gave an insight into the performance of the screws under the applied pre-tension. The results are compared with the theoretical calculations. The results highlight the variation of stresses in joint which are generally not captured by well-established theoretical analyses.

Keywords: *Countersunk screws, lap joint, pre-tension, clamping stress, joint stiffness.*

[1] *Mechanical Engineering, MVSR Engineering College, Hyderabad, India- 501510.*
[2] *Research Centre Imarat, Hyderabad, India – 500069.*

Introduction

Lap joints are routine in many engineering applications. The joints can be temporary (i.e. openable without damage) using fasteners or permanent using rivets or welds (i.e. non-openable). The temporary joints can be either bolted or screwed with either a single lap or double lap. Single lap joint of two plates or two cylindrical airframe sections with countersunk (CSK) screws are common in aerospace applications, wherein, the screws are flushed or conformal with the external surface of the clamped members [1, 2]. The joint does not offer any external projections. A single lap joint of two aluminium plates joined typically by two M5 CSK screws of 10.9 class as shown in Figure 1 and subjected to tension is investigated in this paper. The distribution of stresses in screws and, at above and between two clamped plates under screws are always not well understood by theoretical calculations which provides averaged estimates of stresses. These variations can be captured either by a detailed finite element analysis (FEA) or by an experimental technique like photo elasticity. The FEA is powerful in predictions if the lap joint is idealized and modelled appropriately simulating the real joint behaviour.

Researchers have conducted extensive studies by numerical modeling of bolted joints by FEA and different finite element (FE) modeling approaches have been explored [3-6]. These screwed lap joints are very common in airframe structures and in-between two airframe sections whose diameters are small. They are helpful in easy maintenance activities which requires opening of the airframe sections. Screwed joints are designed with threads and without nut, which allow this fastener to be removed without damage to the system. The main characteristics in the bolted joint are the pretension and the contact at the mating part. Previous studies on the structure with a bolted joint are mostly dedicated to extraction of stiffness for the joint region and estimation of contact stress through a detailed model for a bolted joint using the FEM [7–10]. Based on the earlier works, it can be noted that in order to accurately predict the physical behaviors of the structure with a bolted joint, a detailed three-dimensional bolt model is desirable, which fully includes the friction due to the contact on mating parts and pretension effect to tie. However, for a large complex structures or the joints, the detailed modeling of the bolted joint is difficult because of restriction of the problem size and computational cost to analyze the entire structure [6]. A detailed modelling of single lap screwed lap joint, its FEA and results are discussed in this paper.

Numerical Modelling

The numerical modelling is done by considering a fragment in the overall cylindrical airframe section. The developed 3D finite element model of a single lap joint connecting two members viz. Plate-1 and Plate-2, as seen in Figure 1 is modelled in SOLIDWORKS and analysed using linear static formulation in ANSYS and ABAQUS. The plates are made of aluminium alloy AA 2024 and screws are made of steel with a property class of 10.9. The elastic modulus E = 70 GPa and Poisson's ratio ν = 0.33 for aluminium and E = 200 GPa and Poisson's ratio ν = 0.28 for steel are adopted in FEA. Frictional contacts with co-efficient of friction μ = 0.4 are established between 1) two plates on overlap region, and 2) between CSK screw head and CSK hole in Plate-1 as showed in Figure 1. In the FEM-based models of joint, the in-built connections between the elements are omitted. The threaded connection between the CSK screw and Plate-2 is established with a bonded contact. The plates and screws are modelled in ANSYS Workbench with C3D8R (three dimensional 8 node hexahedral) solid elements which is a reduced integration element. Numerical modelling is the most appropriate way to get the desired results as there is no possibility of simple formulas for the calculation of induced stresses and stiffness of the joined flange element.

Figure – 1: A single lap screwed joint

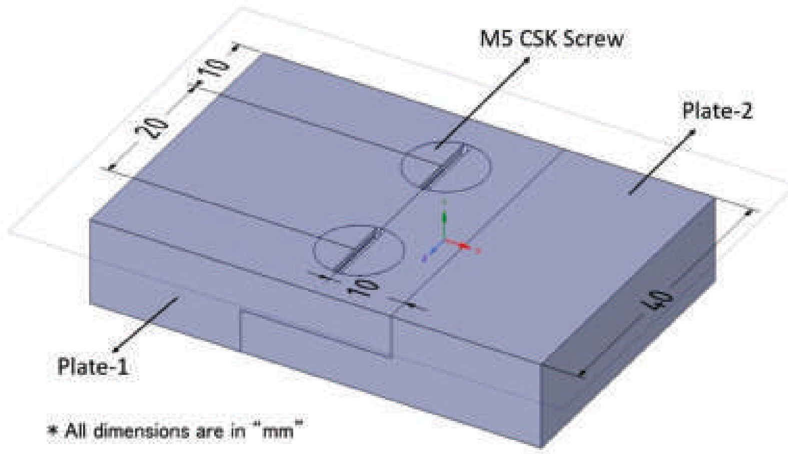
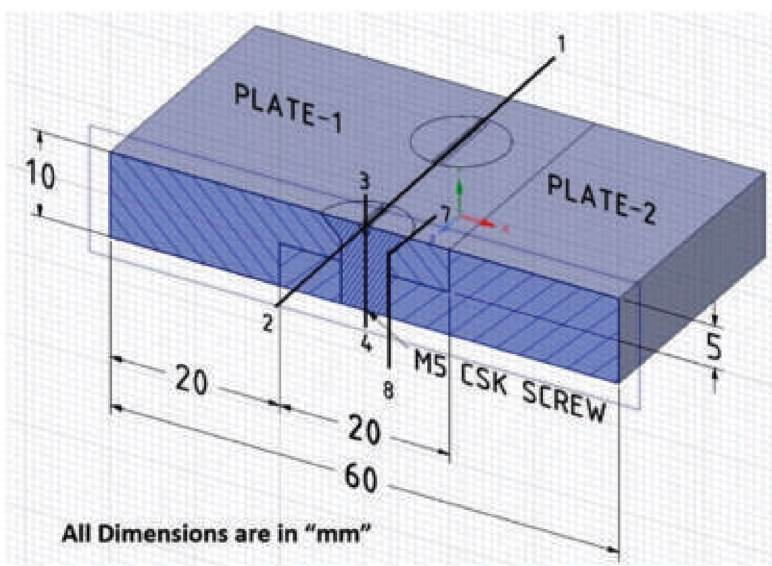


Figure – 2: Different paths 1-2, 3-4, and 7-8 to analyze the stresses



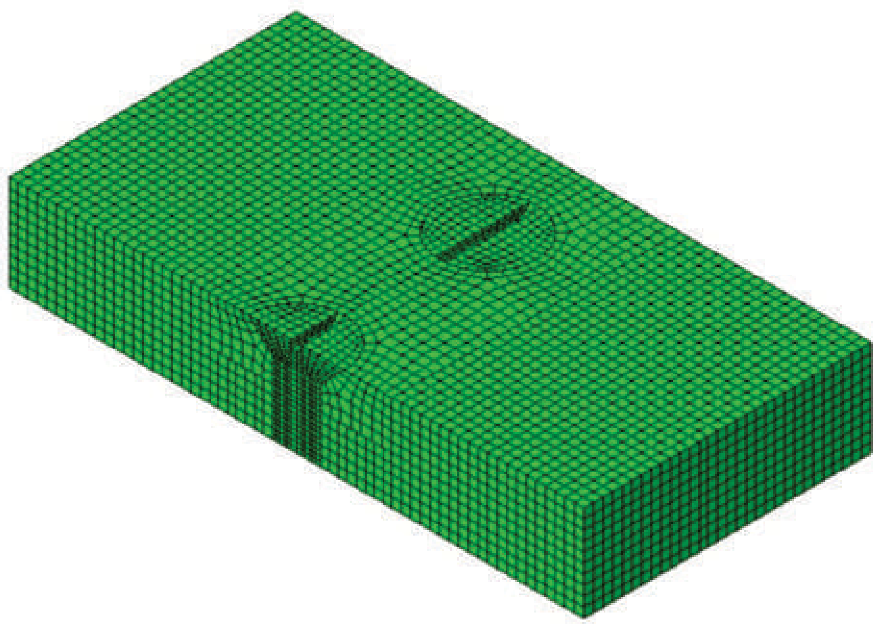
Solid modelling

Solid modelling is distinguished from related areas of geometric modelling using computer graphics. A wireframe representation of an object is done using edges and vertices. Whereas the surfaces are logically made using faces (surfaces), edges and vertices. In this sequence of developments, the solid modelling uses topological information in addition to the geometrical information to represent the object unambiguously and completely for correct definition of solid. The modelled object from the CAD software (Solid Works) are imported into the finite element analysis software in their respective formants STEP or IGES.

Meshing

Finite element method reduces the degree of freedom from infinite to finite with the help of discretization of the model using nodes and elements. The generated FE model is mapped mesh with the element size of 1 mm using hexahedral mesh type. The element size is finalized after a detailed mesh convergence study on induced stress along the path 3-4 i.e. screw length. This is achieved by partitioning the object using datum axis, datum planes and also using faces. Mapped mesh generally provides accuracy in result as it is generally used in various engineering applications.

Figure-3: Discretised model of lap joint specimen for FEA



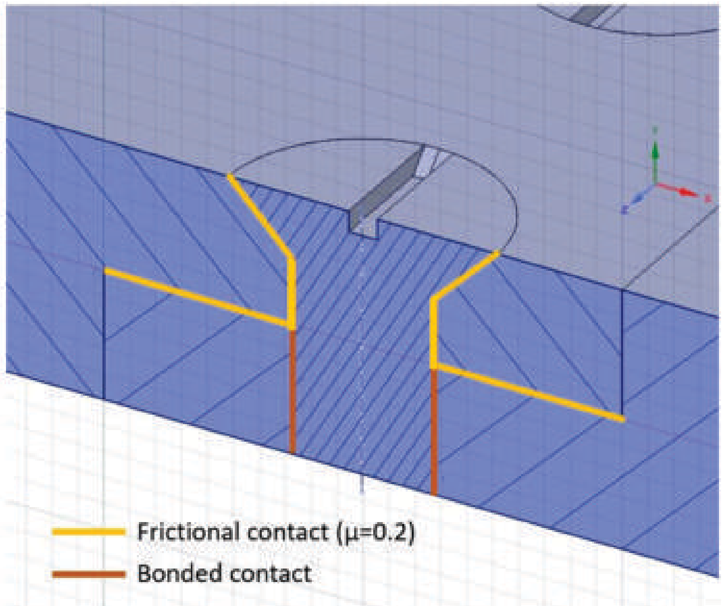
Contact Definition

A full definition of the contact zones is implemented by modelling the contact between

a. The main plates
b. Screws (head) and upper plate hole
c. Plate holes and screw shanks

The contacts are solved using penalty method with hard contact for frictional contacts between above mentioned (a) and (b). The coefficient of friction (μ) used is 0.4. The surface to surface discretization is employed for contacts between the plate holes and the screws shank to avoid element interpenetration at the thread edges of the screw. This type of bonded contact in (c) are established by tying the adjusted surfaces.

Figure-4: Contact definitions



Load Step Definition

Boundary, initial, and loading conditions play a decisive role in the simulation. Prescribing these conditions is usually done easily using commercial pre-processors. Users can specify these conditions either to the geometrical identities (points, lines or curves, surfaces, and solids) or to the mesh identities (nodes, elements, element edges, element surfaces). Again, to accurately simulate these conditions for actual engineering systems requires experience, knowledge, and proper engineering judgments. The analysis is divided into two stages one with pretension and the other without pretension.

The first stage is divided into two steps. The left edge of the Plate-1 is clamped in all degrees of freedom and the right edge of the Plate-2 is subjected to a tensile displacement δ = 0.04 mm. This corresponds to 50% of tensile yielding of the plate. This step simulates the experiment as displacement-controlled static test.

$$\delta = \frac{\sigma_y l}{2E} \quad (1)$$

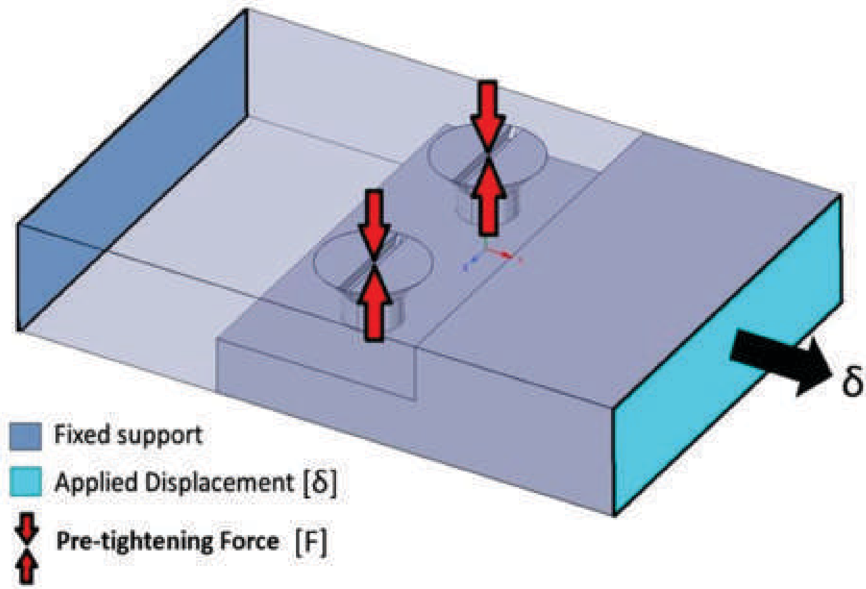
where σ_y denotes the yield strength of the plate material = 280 MPa and *l* denotes the length of the Plate-2 = 50 mm. The screws are pre-torqued to *T* = 10 Nm and its effects are simulated on the joint by applying a pretension which is the secondary stage F_i =10 kN as calculated in Eq. 2 and an equivalent compressive clamping force is applied on the countersunk hole of Plate-2. Initial pretension is given by

$$F_i = \frac{T}{kd} \quad (2)$$

where *d* denotes the nominal diameter of screw = 5 mm and *k* denotes a coefficient of 0.2 for this joint. Linear elastic FEA are carried out including and excluding the effect of pretension.

The second stage is addition of pretension to first stage. The clamping force is applied to the screws. In this the screw is progressively shrunk along the screw axis with the force of 10 kN as calculated above. This step simulates the pre-tightening of the joint as shown in Fig. 5.

Figure-5: Loads and boundary conditions



Prediction of Joint Stiffness

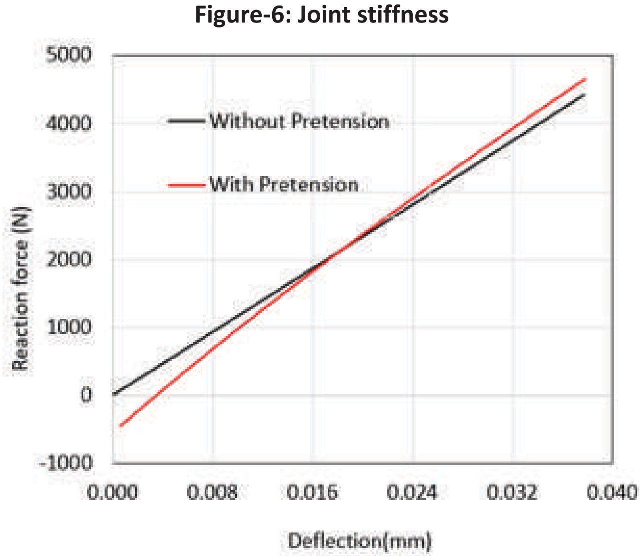
Joint stiffness is the apparent loss of range of motion of a joint. Simply it is the load required to produce a unit deflection in a joint.

$$K = \frac{P}{\delta} \qquad (3)$$

where K denotes stiffness (N/mm), P denotes load or the reaction force (N), and δ denotes the deflection (mm).

With the application of preload on to the joints, the slope of the load vs deformation increases moderately which indicates that the preload effect reduces the stiffness at the first point even before the start of the displacement is applied. With the application of the displacement uniformly with time, the preload helps in relieving the stresses in the joints and helps in preventing the failure of the joint eventually with an enhancement in stiffness, which is shown in the Fig. 6.

Figure-6: Joint stiffness

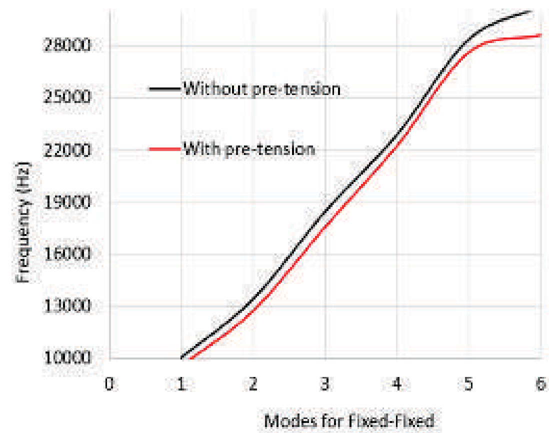


Modal analysis

Modal analysis is implemented in three boundary constraints: Free-Free; Fixed-Free; and Fixed-Fixed. The natural frequencies in the modal analysis increases with increase in joint stiffness, but the present case of the analysis predicted higher frequencies considering no pretension and slightly low frequency considering the pretension as shown in Fig.7. This is only because the effect of pretension in the joint caused relatively lower stiffness during the initial application of displacement as shown in the Fig. 6, which specifies that the joint stiffness is less, when the applied load is very less (or) zero in the case with pretension. Since, in the analysis of these frequencies, no load or displacement is applied, other than pretension, the frequencies are lower compared to the case without pretension justifying the results obtained in all the stages as shown in Fig. 7.

Figure-7: Results from modal analysis

(a) Fixed-free boundary

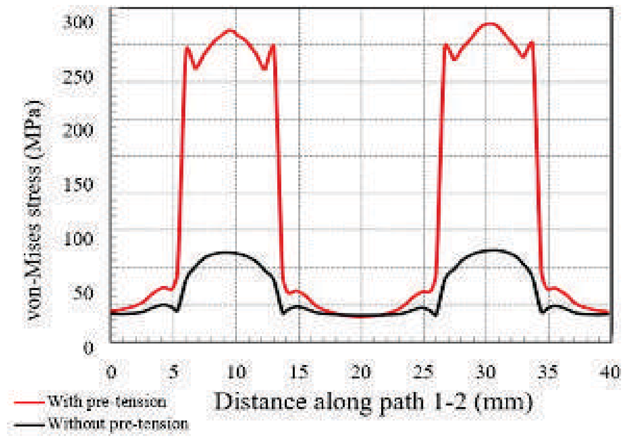


(b) Fixed-fixed boundary

Results and Discussion

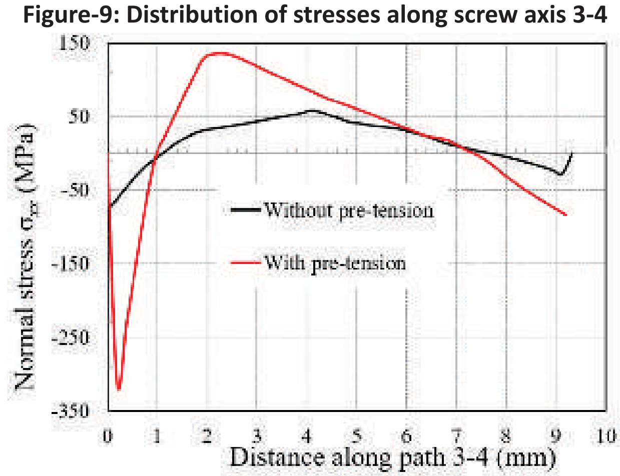
The stresses along different pre-identified paths on the screws and clamped members are extracted. The distribution of induced stresses along these paths on screws and clamped members are as shown in Fig. 8. The pretension induces higher local compressive stresses just below the CSK head and the CSK hole in the Plate-1 which is noticed in Fig. 8. This is not the case when pretension is not applied on to the screws. In the Fig. 9, the countersunk screw head undergoes complete compression due to pretension and changes to tension in the shank and the threaded portions. As the pretension imparts counteracting force upon the screw the stresses are high for the pretension case continuously in both tensile and compressive zones.

Figure-8: Distribution of stresses in clamped members along path 1-2



It experiences tensile stresses at the middle of the screw and then compression as it reaches the ends i.e., towards the head and the tail of the screw as shown. On the application of pretension, the stresses developed are higher along all the paths, indicating that with the application of pretension, the joint exhibits high reaction to deform under an applied displacement of 0.04 mm.

Figure-9: Distribution of stresses along screw axis 3-4



Thus, the strength of the model increases by minimizing the effects of displacement over the body, when pre-tension is applied. Aluminium, the plate material which might undergo different types of failure modes such as tensile failure, compressive failure, shear failure due to the stresses which are caused by the internally generated resistance due to thermal expansions, external conditions such as intense aerodynamic forces, causing pressure variation on the sections of the flight which directly affect the joint. All these forces and the expected failures experienced by the flight during its operation, are controlled by the pre-tightening force applied on the screwed joints, by reducing the stresses induced on the materials. As this paper focuses towards the study of the effect of the applied pre-tightened force on the joints and their behaviour, the failure analysis of the is ignored.

Conclusions

This paper presented a single lap CSK screwed joint of two aluminium plates subjected to tension. The details of FE modelling, contacts and boundary conditions are presented and analyzed through a displacement-control to an extent of 50% of tensile yielding. The effect of pre-tightening on screws and distribution of stresses along screws and along different paths on the clamped members are obtained. The results provide an insight into the behaviour and distribution of stresses in the joint. The induced stresses are high when pretension is imparted to the screws compared to the case without pretension. This indicates that the stress i.e. the force required to pull the model to a displacement of 0.04 mm is more in the case with pretension which means, the stiffness of the joint with the application of pretension is increased at the end of applied displacement, preventing the failure of the model. On the application of pretension, the stiffness increases with the increasing displacement applied on the model.

References

- Bruhn E.F, Analysis and design of flight vehicle structures, Jacobs Publications, USA., (1973).
- Niu M.C.Y., Airframe stress analysis and sizing, Hong Kong Conmilit Press Ltd, Hong Kong., (1997).
- Bursi O.S. & Jaspart J.P., Basic issues in the finite element simulation of extended endplate connections, Computers & Structure, 69, 361-382, (1998).
- Maggi Y.I., Goncalves, R.M., Leon, R.T., & Ribeiro, L.F.L., Parametric analysis of steel bolted end plate connections using finite element modeling, Journal of Constructional Steel Research, 61, 689-708, (2005).
- Sherbourne, A.N., & Bahaari, M.R., 3D simulation of bolted connections to unstiffened coloumns-I. T- Stub connections, Journal of Constructional Steel Research, 40(3), 169-187, (1996).
- Kim, J., Yoon, J.C., & Kang, B.S., Finite element analysis and modeling of structure with bolted joints, Journal of Applied Mathematical modeling, 31(5), 865-911, (2006).
- Gould, H.H., & Mikic, B.B., Areas of contact and pressure distribution in bolted joints, Trans. ASME J. Mech. Des. 94, 864–870, (1972).
- Wileman, J., Choudhury, M., & Green, I., Computation of member stiffness in bolted connections, Trans. ASME J. Mech. Des. 113, 432–437, (1991).
- Nabil, M., Determination of joint stiffness in bolted connections, Trans. ASME J. Mech. Des. 98, 858–861, (1976).
- Schiffner, K., & Helling, C.D., Simulation of pre-stressed screw joints in complex structures, Comput. Struct. 64, 995–1003, (1997).

Alluri Vineeth *is B.E. in Mechanical Engineering from MVSR Engineering College, having graduated in October 2020. This paper is an abridged version from his B.E. academic project. He can be reached at vineethalluri92@gmail.com.*

Abhishek Kundu *is B.E. in Mechanical Engineering from MVSR Engineering College, having graduated in October 2020. This paper is an abridged version from his B.E. academic group project. He can be reached at abhishek.kundu0610@gmail.com*

MVNR Saketh *is B.E. in Mechanical Engineering from MVSR Engineering College, having graduated in October 2020. This paper is an abridged version from his B.E. academic group project. He can be reached at sakethmylavarapu@gmail.com*

Vijayabaskar Narayanamurthy *is Scientist at Research Centre Imarat, Hyderabad. His research interests include modelling of hybrid structural members, impact mechanics, flight interface structures and mechanisms. He can be reached at v.narayanamurthy@rcilab.in.*