6. Finite Element Analysis

The ever-increasing advances in computer technology have enabled engineers to simulate physical conditions eliminating the costs of traditional prototype based research and development. At Swansea Metropolitan University, I-DEAS CAD software is used to model components[1]. Figure 1 shows a typical High Torque fastener design used in the analysis. It is then possible to perform a finite element analysis of the component. The finite element method is a numerical method used to solve complex engineering problems. The method was first developed almost fifty years ago and has become so well established that today it is considered to be one of the best methods for solving a wide variety of practical problems efficiently. I-DEAS has been used to analyse the performance of the High Torque system under mechanical loads. This allowed the engineer to run numerous simulations of the problem greatly reducing project time-scales and resulting in a more thorough investigation.

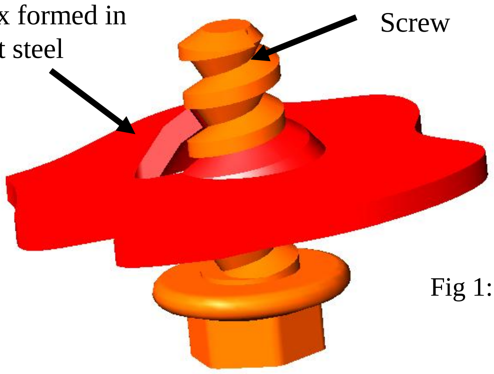


Fig 1: Typical CAD model used in the analysis

A geometric non-linear calculation has previously been undertaken by imposing various loads and restraints on the helix geometry alone. The non-linear has the advantage over the simpler linear approach in that as the traction becomes progressively larger in magnitude, the displacement of the fastener plate is more accurately modelled. A linearly increasing traction was imposed on the surface of contact between the screw and the fastener plate. This permitted a range of imposed loads that helped to determine the point where the maximum calculated stress on the plate exceeded the permitted value. This report looks at a geometric linear calculation with contact, analysing both the helix form and the thread of the fastener. Incorporation of plasticity into a non-linear model with contact will form the basis of future work.

Meshing and Boundary Conditions

Considerable effort has gone into obtaining an accurate finite element mesh (Figure 2a). Parabolic tetrahedral elements were used throughout the calculations. Experience has taught us that calculated surface stresses tend to overestimate the problem areas. The industry standard is to use three elements through the thickness of any component in order to correctly model the stresses through that thickness. This was considered important, since comparisons between calculations for different fastener designs required a consistent approach to meshing and hence the accuracy from calculation to calculation. This proved to be very time consuming, but has resulted in a meshing methodology that we now have confidence in. The head of the bolt was not included in the analysis to allow more elements to be used in the areas of interest around the base of the helix and the area of contact between the helix form and the bolt (Figure 2b).

---

[1] **Donne, K, Payne, C.** (2004) *Computer Modelling of Novel Metal Fasteners.* IMechE 5th International Conference on Quality, Reliability & Maintenance. Oxford University

6. Finite Element Analysis

The head of the bolt will be used in future work for evaluating torque when friction under the head will be important. The accuracy of the mesh was increased until the total error norm for each calculation was less than 5%.

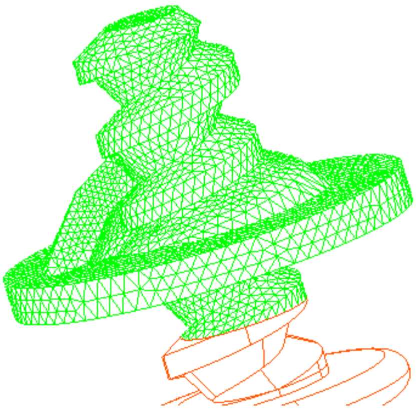


Fig 2a: Typical mesh being used in the analysis

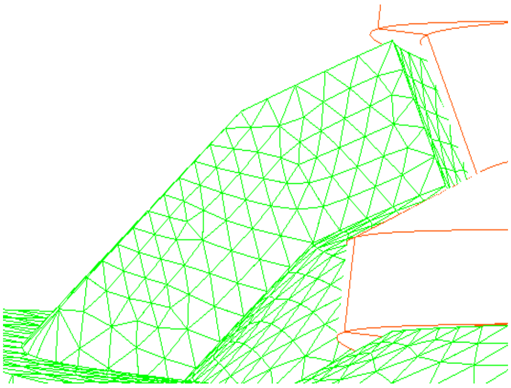


Fig 2b: Concentration of elements around the areas of interest

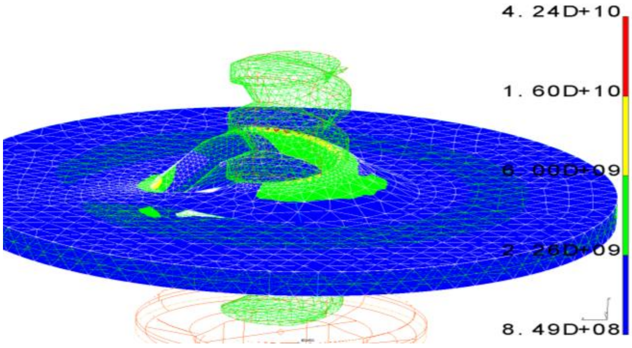
The lateral surface of the fastener plate was restrained to represent the presence of the larger plate geometry (Figure 3). This diameter is consistent with that of the experimental test and that of any clearance hole that would exist on assembled products. Initial analysis involved a translation calculation. All surfaces of the bolt were restrained with the y & z axis remaining constant at zero to simulate the bolt pulling through the helix form.

Fig 3: Boundary Condition Restraints around the plate geometry

Finite Element Analysis Results

Fastener Diameter 4mm for 1.0mm material has been used here as an example. Figure 4a shows the Von-Mises stress plot for a 5mm axial displacement of the screw, and figure 4b the stress concentration at the base of the leg. It is important to observe that, although the maximum stress value significantly exceeds the Yield Stress for the material, it is the depth of penetration that determines failure. This is entirely consistent with numerous simulations performed for other industrial clients, where this observation is used in predictive modelling.

6. Finite Element Analysis



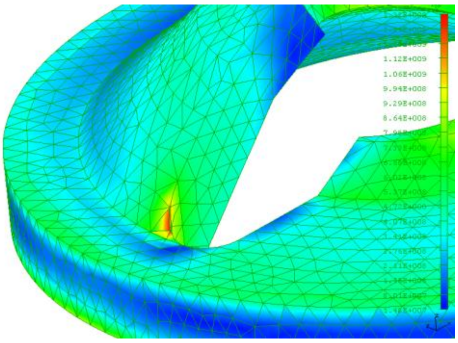
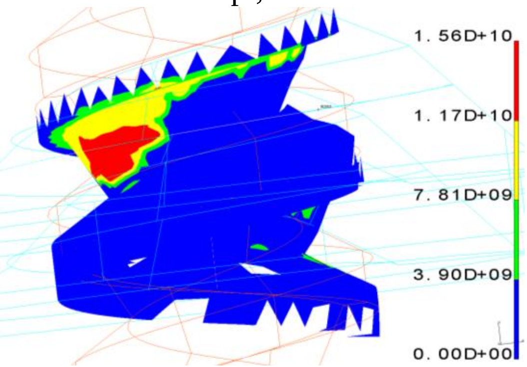


Fig 4a: Von Mises stress for a 5mm screw displacement

Fig 4b: Violation of yield stress at base of helix

The computer model is proving useful in clarifying the failure mode of the fastener. Figure 5a shows the contact stress on the screw surface, while Figure 5b shows the contact stress on the helix surface. The maximum value of the contact stress on the screw is less than that on the helix, but is of a similar order of magnitude. For mild steel, the predicted failure mode is due to failure of the helix form and this is confirmed by experiment. Further tests indicate that when a work-hardened helix is used, the failure mode is then due to the screw thread failing. Future computations will concentrate on modelling the non-linear behaviour of the helix-screw interaction with plasticity and creep included, in order to develop a predictive model that can be applied to novel applications of the fastener concept, in non-standard sizes.



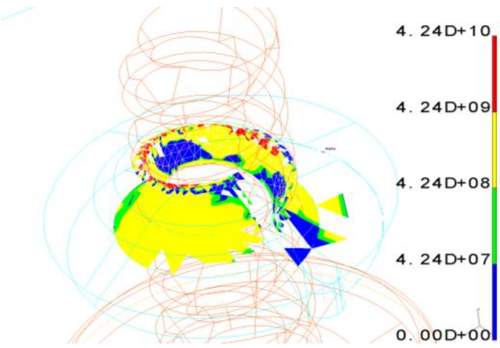


Fig 5a: Contact stress on screw

Fig 5b: Contact stress on helix

Conclusion

The computer model shows that simply exceeding the plastic limit does not immediately lead to failure. The violation of the yield stress must occur through a significant depth of the wall thickness. As more experiments are performed, the model can be calibrated so that this penetration through the wall can be quantified. Currently it appears that some 75% of the wall thickness at the base of the slot must be violated before failure occurs. There is an analogy with a simple beam cantilever: the free end of a cantilever experiences the greatest deflection – just like the helix in the fastener. However, the fixed end of a cantilever experiences the highest stress – just as seen in the base of the slot. A more accurate criterion for predicting failure can be made after we calibrate the model with further experiments. We will then refine the 75% estimate above with a more accurate figure.
The computer model is intended to predict failure for yet-to-be-manufactured geometries and other materials. However, at present, the customer should refer to the experimental values in determining fastener performance. Please note that fastener performance may vary in applications due to the influence of product geometry and environmental factors.