COMPUTATIONAL STRUCTURAL DYNAMICS SIMULATION WITH FEM, INDIVIDUAL

EMS602U - Simulation Tools in Engineering Analysis and Design

Amir Pousti

| Queen Mary University of London

Abstract
The structural behaviour of a 2D spanner model under distributed load is examined using ABAQUS finite element analysis (FEA). High-stress areas and potential failure locations were Identified and this improves spanner performance and longevity. A comprehensive model was created using the material AISI 1045 Carbon-Steel selected from EduPack for its tensile strength and longevity. High-stress locations near the spanner head-handle area was examined. Stress distribution and solution convergence were assessed for six meshes with different element types, polynomial orders, and seed sizes. Mesh refinement greatly raised maximum von Mises stress values, showing us crucial stress concentrations. Quadratic elements captured stress gradients and displacement behaviour better than linear elements. This technique optimises spanner designs by emphasising mesh resolution, geometric refinement, and material choice to reduce stress concentrations and extend the tools life.

Introduction
Spanners are essential equipment for everyday engineering and maintenance tasks. Bolts, nuts, and other fasteners can be grasped, tightened, or loosened with them (Finder Tool, 2024). Even though they are straightforward, spanners are subjected to complex stress distributions while in use, which can cause failure if they are not built properly. Both tool longevity and worker safety depend on a spanner's capacity to safely support operating loads.

The choice of materials, geometry, and manufacturing procedures are all carefully considered while developing spanners. Engineers today mostly use finite element analysis (FEA) to predict how a spanner will react under different loading scenarios because of advancements in computer technology. Stress concentrations, deformation patterns, and possible failure sites can all be better understood by FEA.

In this study, we study the structural behaviour of a 2D spanner model under a distributed load using FEA in ABAQUS. By creating a model with the right geometric and material selections, we hope to reproduce realistic operational conditions. Finding high-stress regions that can cause structural problems or failure is the primary objective of the analysis. We also investigate how different mesh configurations, such as mesh refinement, element types (quadrilateral vs. triangular vs. quadrilateral dominated), and polynomial ordering (linear vs. quadratic basis functions), affect the convergence of the solution. This approach informs design changes for improved performance and safety and offers insights on the spanner's possible failure areas.

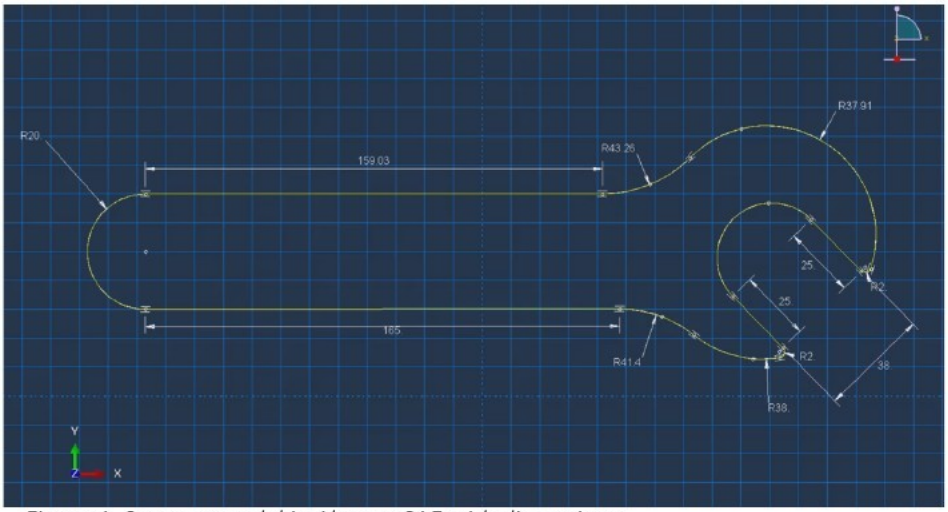

Figure 1. Spanner model in Abaqus CAE with dimentions

As seen in figure 1, the spanner fits nuts and bolts. With diameter of 38. Its design has a straight handle measuring 159.03 mm, ending in a rounded section with a radius of 20 mm to enhance comfort and distribute stress. The head section features a complex geometry with outer curves of 43.26 mm and 37.91 mm, and an open slot 25 mm wide, designed to fit securely over nuts or bolts. Small rounded corners with 2mm radius reduce stress concentration within the slot, improving durability. Additional curves with radii of 38 mm and 41.4 mm reinforce the head and enhance ergonomics with a spanner thickness of 20mm.

To select the material, Edupack was used where the Young modulus was compared to the Poisson's ratio. Non-Ferrous and precious metals were filtered and only ferrous were added to the selection due to their superior ensile strength and durability. The final material selected was AISI 1045 Carbon-Steel where the mechanical properties needed are. Presented in table 1.

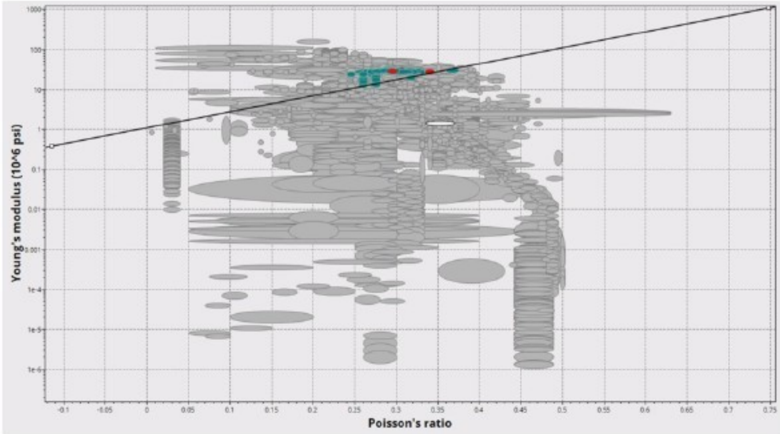

Figure 2. material selection graph from Edu pack (Ansys. 2024)

Table 1. Material Properties (Matweb, n.d.)

AISI 1045 Carbon-Steel	
Property	Metric
Young Modulus	206 GPa
Poisson's Ratio	0.29

In figure 3, we can see a distributed load of 350N was applied to a length of 65.20 mm, average length of a hand, 41% of the total 159.03mm length. The area needed to be solved so the length was multiplied to the thickness. The calculation below shows the exact pressure applied to the spanner.

$$Pressurre = \frac{force}{area} = \frac{350}{65.2 \times 20} = \frac{350}{1304.046} = 0.2684\ MPa$$

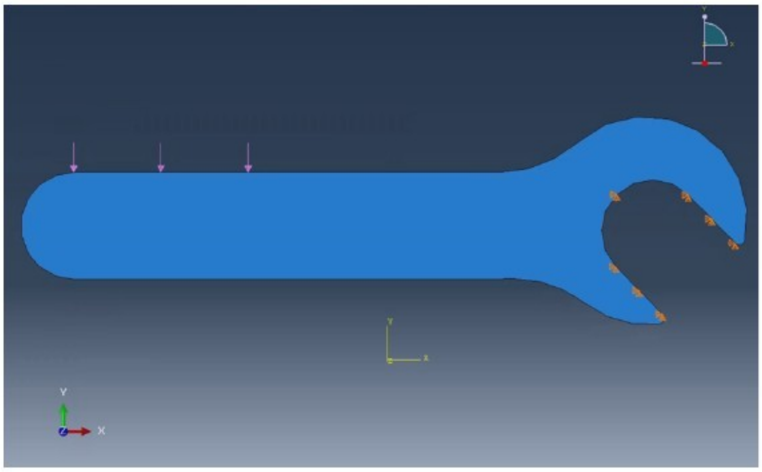

Figure 3. Load and boundary conditions on spanner

A displacement/rotation boundary condition was applied to the jaw area of the spanner locking it in the U1 and U2 (X and Y) direction restricting in all degrees of freedom. by fixing these edges, you are simulating a real-world scenario where the spanner is tightly gripping a bolt, nut, or another object. The boundary conditions can be seen in figure 3. (orange arrows)

Six different mesh types were utilized to evaluate their performance under various conditions. These included quadrilateral elements with linear polynomial orders, quadrilateral elements with quadratic polynomial orders, triangular elements with linear polynomial orders, triangular elements with quadratic polynomial orders, quadrilateral-dominated elements with linear polynomial orders, and quadrilateral-dominated elements with quadratic polynomial orders. A range of seed sizes was employed—specifically 10, 9, 8, 7, 6, 5, 4, 3, 2, 1, 0.75, and 0.5 to assess the impact of mesh density on the results. The mesh type used was global seeds, where the scope is applied to the entire model. It provides a mesh density for all edges that do not have specific local seeds assigned. However, for local seeds, the scope is applied to specific edges, faces, or regions of the model. It overrides the global seed settings to refine or coarsen the mesh in areas where different mesh densities are required.

Results

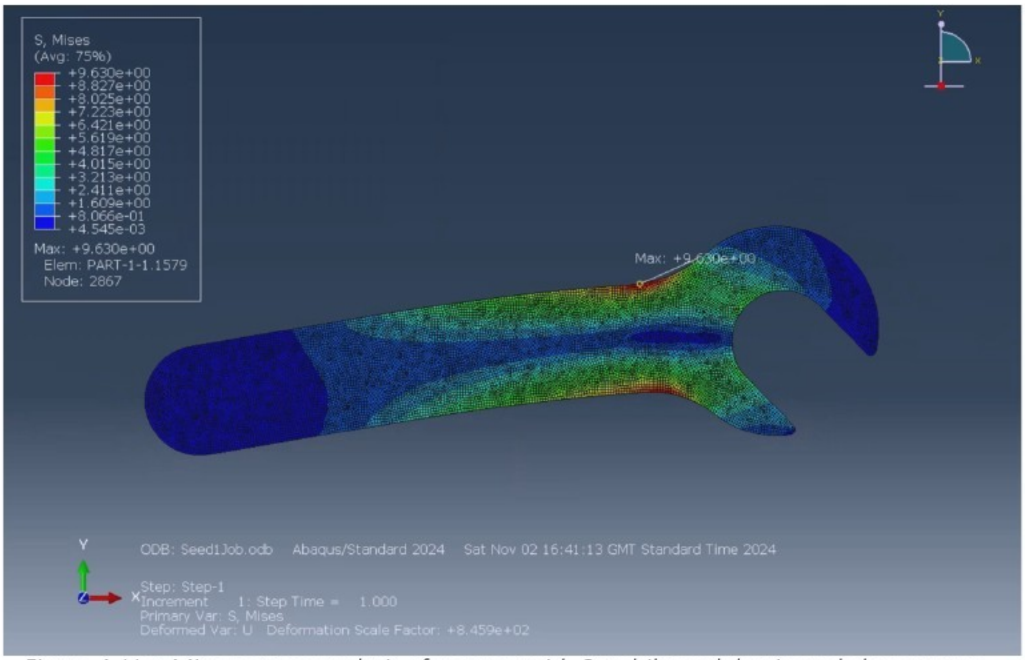

Figure 4. Von Misses stress analysis of spanner with Quadrilateral dominated element type, linear polynomial order and Seed size of 1

The orange and red areas in Figure 4 represent high-stress areas. High stress seems to be centred at the meeting point of the spanner head and handle and at the curved areas where the spanner holds a fastener. This is typical for a spanner because these areas endure the greatest amount of applied torque or force. Deformation or fatigue failure may result from high-stress locations, particularly if the spanner is loaded often. It is crucial to make sure that these stress concentrations never surpass the material's fatigue limit or yield strength. The change in geometry from the spanner's handle to its head results in a concentration of tension. The way forces are distributed throughout the spanner's body is significantly influenced by its size and shape. If the tension in the critical areas is greater than the yield point of the material, the spanner may gradually deform, reducing its efficiency. In high-stress locations, fatigue cracks may develop as a result of frequent spanner use, especially when loads are varied. The spanner may abruptly fail as a result of these cracks spreading over time.

As seen on figure 5, for all mesh types and polynomial orders, the maximum von Mises stress rises as the mesh is refined. Because coarser meshes are unable to precisely record stress concentrations and average out stress peaks, they have a tendency to underestimate the maximum stress values. These peaks become more noticeable as the mesh gets finer, which raises the stress values that converge to the actual solution. For instance, the maximal von Mises stress in the quadrilateral element type with linear polynomial order rises from 6.954 at a seed size of 10 to 9.939 at a seed size of 0.5. This pattern may be seen for both linear and quadratic polynomial orders, however it is less evident for quadratic elements how quickly stress increases with mesh refinement. This suggests that even with a coarser mesh, quadratic components are better able to capture stress gradients than linear elements.

Mesh refinement significantly raises the stress values for linear elements at maximal von Mises stress. So the first analyses using coarse meshes may significantly underestimate the stress and possibly miss important stress concentrations. Quadratic elements also show an increase in maximum stress with mesh refinement, however this effect is less apparent. Quadratic elements capabilities allow them to better capture stress concentrations, resulting in more accurate stress forecasts even with fewer components

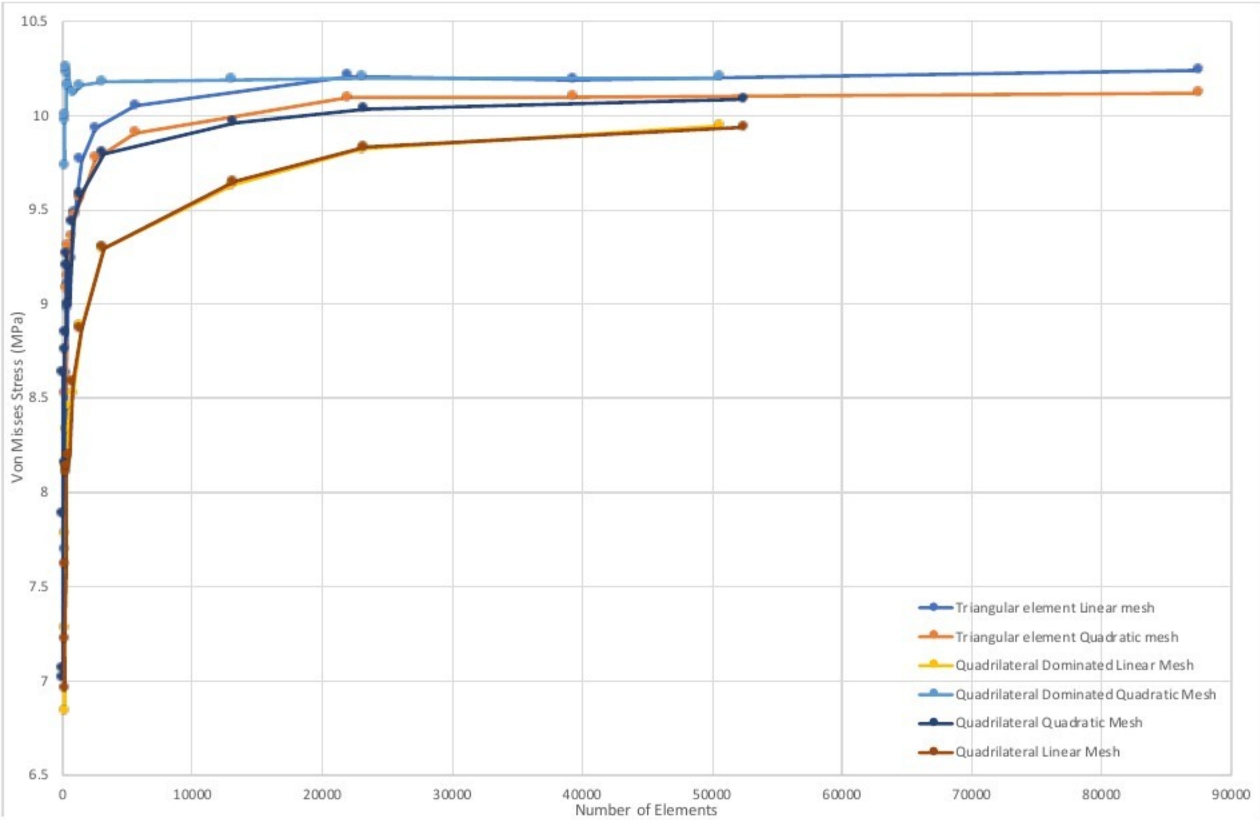

Figure 5. Graph of number of elements vs Maximum Von Misses Stress

A peak in stress readings may be produced at specific intermediate mesh densities. These anomalies may be introduced by elements such as aspect ratios, element distortion, and the particular element type. In the quadrilateral-dominated element type with a quadratic polynomial order this is seen. The model underestimates the maximum stress at bigger seed sizes (coarser meshes) because it lacks the detail necessary to correctly represent stress concentrations. The predicted stress rises when the mesh is fine-tuned (lower seed sizes) because the elements more accurately reflect the geometry and loading circumstances.

For all mesh types, the maximum displacement values converge towards a certain value when the seed size reduces. This is seen in figure 6. The initial maximum displacement is higher for quadrilateral components with linear polynomial order at bigger seed sizes as it is a. coarser mesh. The maximum displacement falls and converges to 0.03209 with 52,452 elements at seed size 0.5 as the mesh is fine-tuned by reducing the seed size. The maximum displacement of triangular elements with linear polynomial order, on the other hand, begins lower and increases initially as the mesh is refined before convergent. With 87,577 elements at seed size 0.5, they converge to 0.03208. Mesh refining is implied to improve displacement calculations by lowering numerical errors. Mesh types vary in their initial displacement values and rates of convergence, which highlights how crucial mesh selection is in FEA.

Displacement results' accuracy and convergence are affected by the finite elements' shape. A more dependable and rapid convergence to the maximum displacement value is frequently provided by quadrilateral components as opposed to triangular ones. Their displacement values are close to the converged value, and they require fewer components. In contrast, triangular elements need a larger number of elements to reach the same level of displacement precision as quadrilaterals. This is evident from the variability in initial displacement values obtained from triangular meshes, suggesting that coarser triangular meshes yield less stable outcomes. Quadrilateral-dominated meshes—which utilize quadrilaterals where possible and triangles in complex geometric areas—offer performance that falls between purely triangular and purely quadrilateral meshes. This hybrid approach effectively combines the adaptability of triangles with the efficiency of quadrilaterals for meshing intricate geometries.

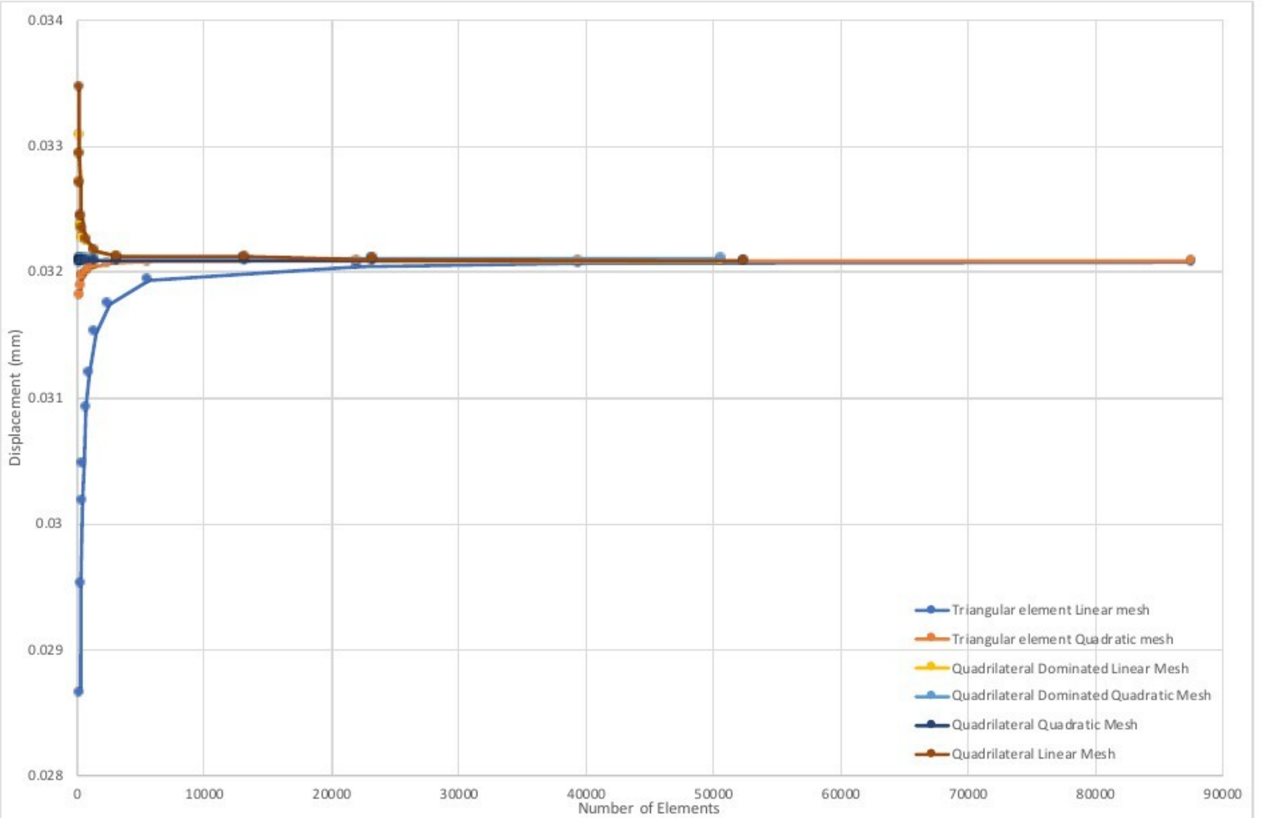

Figure 6. Graph of number of elements vs Maximum Displacement

The convergence tendency and accuracy of displacement findings are significantly impacted by the polynomial order of the elements. Compared to linear elements, quadratic elements, which have a higher polynomial order, require a significantly smaller number of elements to converge to the maximum displacement value. To achieve similar displacement accuracy, linear elements demand a considerably finer mesh, smaller seed sizes, and a greater number of elements. For instance, a quadrilateral linear mesh may require 52,452 elements to converge to a displacement value of 0.03209. Quadratic elements are better at modeling curvature within elements, allowing them to capture displacement behavior more accurately even with coarser meshes. Consequently, they use less computational power to attain the desired degree of accuracy.

Conclusion

A 2D spanner model was subjected to a 350 N distributed load, and its structural behaviour was assessed using ABAQUS finite element analysis. The fastener contact regions and the spanner head-handle intersection, which are vulnerable to deformation and fatigue failure, had high stress concentrations. Developing an understanding of these important areas improves spanners' durability, safety, and design. Mesh resolution had a major effect on the results. Reducing seed size improved mesh refinement and maximum von Mises stress values. Finer meshes more precisely recorded stress concentrations, increasing estimates of stress and displacement, whereas coarser meshes overstated stress by averaging stress peaks. The analysis was also affected by the polynomial order. Even at coarser meshes, quadratic components required fewer elements than linear elements to precisely record displacement behaviour and stress patterns. It was shown that linear elements needed more elements and finer meshes to attain the same accuracy as higher-order polynomial elements in finite element studies.

In summary, identifying high-stress areas, maximising mesh resolution, and utilising higher-order polynomial elements are necessary for precise finite element analysis of spanner designs. Stress concentrations in the spanner can be decreased by increasing fillet radii at critical spots, optimising shape for stress distribution, reinforcing high-stress sections, and possibly using materials with higher yield strength or fatigue resistance. adding grooves or notches to relieve stress in key areas, shot peening, and enhancing handle ergonomics. By controlling spanner stress, these traits can reduce the probability of failure at critical locations.

References

Ansys. 2024. Granta EduPack (level 3). [Software]

Finder Tool. (2024). Understanding the Function and Use of a Spanner Wrench - Finder Tool. [online] Available at: https://www.finderhardware.com/understanding-the-function-and-use-of-a-spanner-wrench/

Matweb (n.d.). AISI 1045 Steel, cold drawn, high temperature stress relieved, 50-75 mm (2-3 in) round. [online] www.matweb.com. Available at: https://www.matweb.com/search/datasheet_print.aspx?matguid=193434cf42e343fab880e1dabdb143ba.