

Stress Analysis of Double Lap Joint

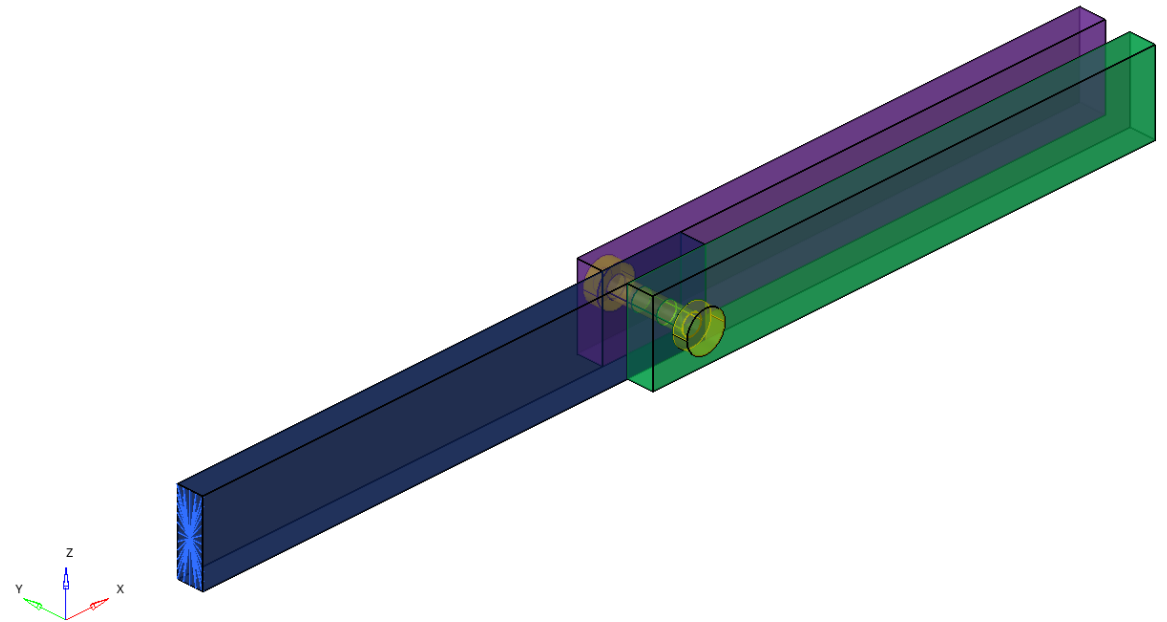
For questions, please use contact form



ALGO
Engineering
Simplifying FEA

Model Description

- Stress Analysis of Double Lap Joint is based on the experimental study documented in the following paper.
 - [Bhonge, Prasannakumar & Foster, Brian & Lankarani, Hamid. \(2011\). Finite Element Modeling and Analysis of Structural Joints Using Nuts and Bolts. ASME 2011 International Mechanical Engineering Congress and Exposition, IMECE 2011. 3. 10.1115/IMECE2011-62905.](#)

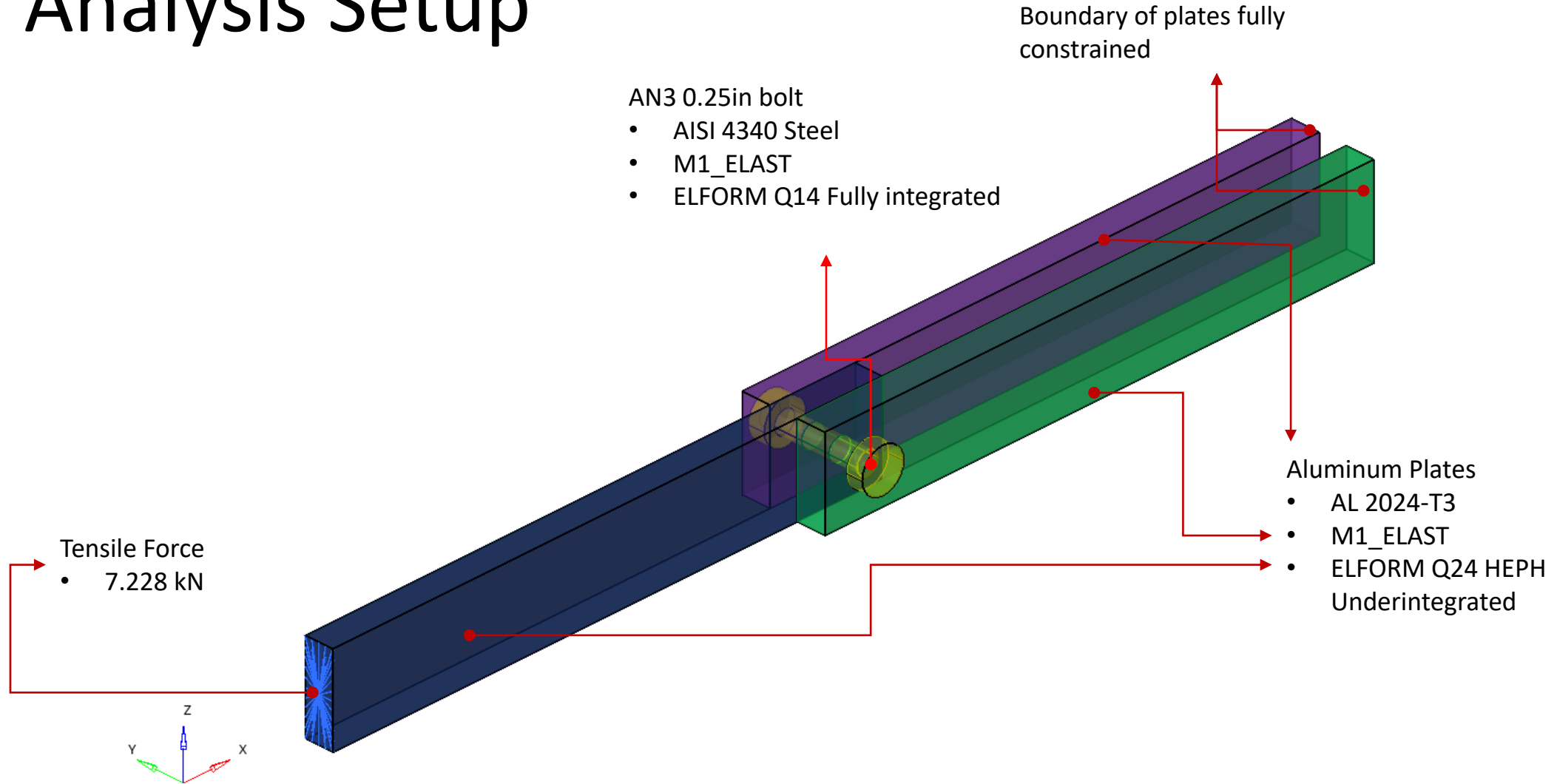


Model Parameters

Entity	Type
Solver	Altair Radioss
Version	2022.1
Processors	2
Threads	2
CPU	Intel(R) Core(TM) i7-9750H CPU @ 2.60GHz
Total run time	2717.55 s

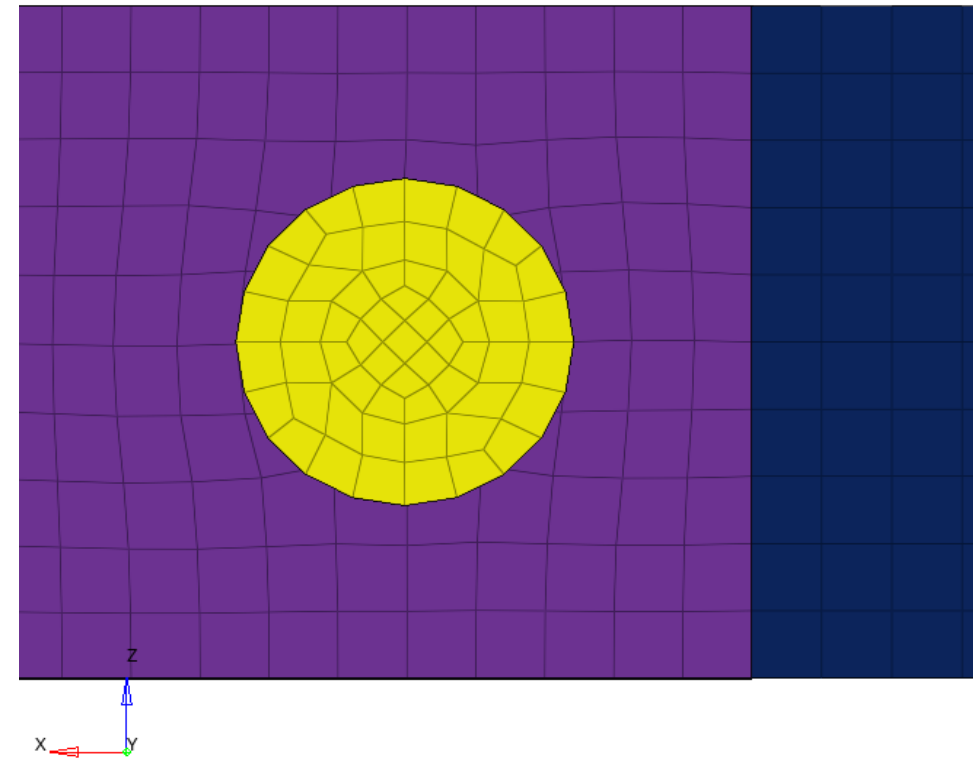
FEA Entities	Type
Analysis Type	Dynamic Explicit
Unit System	kg, mm, ms
Element Type	P14_SOLID
Element Formulation	Q24
Material Type	M1_ELAST

Analysis Setup



Mesh Quality

- 0.649 mm – min element length
- 0.623 – Jacobian
- 2.39 – max aspect ratio
- 8310 solid elements



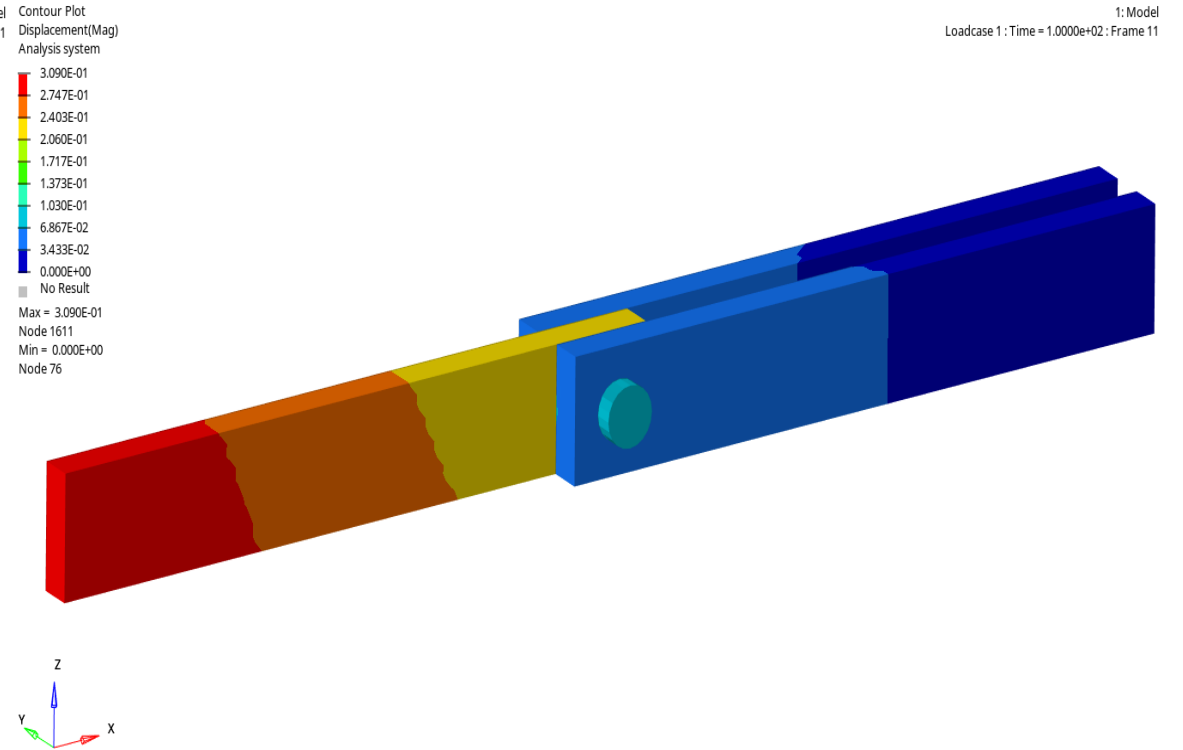
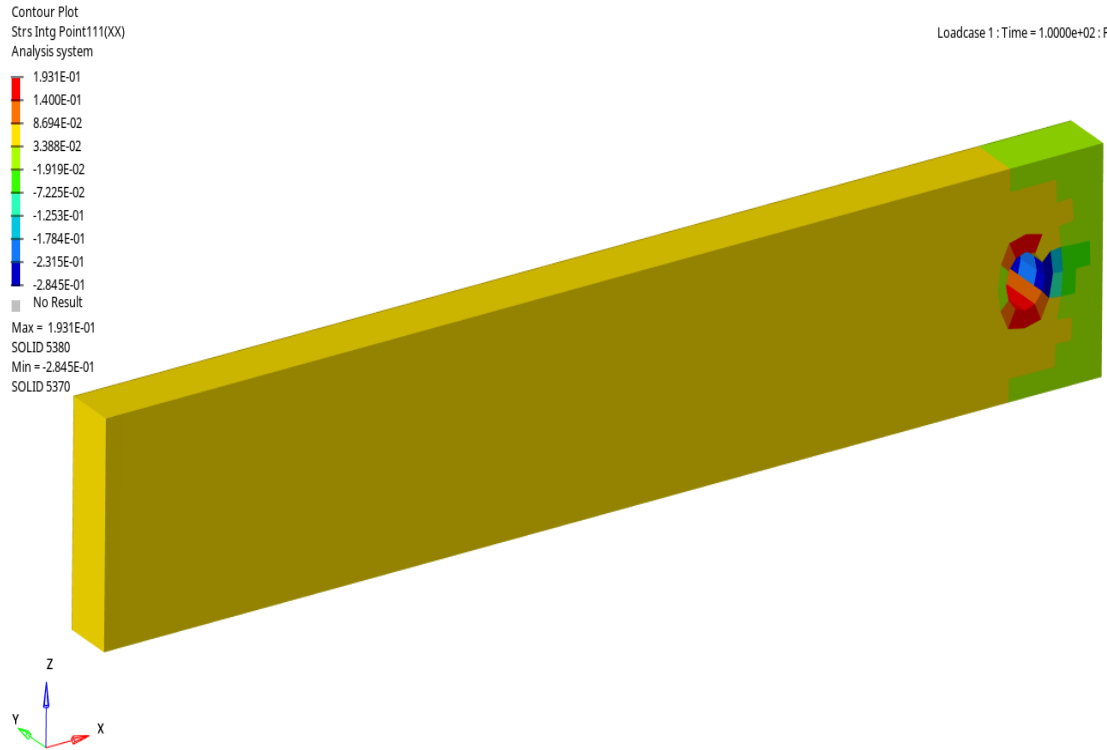
Assumptions

- Dimensions of the bolt head was not mentioned in the paper and is thus assumed.

Analysis Results

Stress (XX)

Displacement



Max value - 0.1931 GPa (28006.79 psi)

Max value - 0.309 mm (0.0122 in)

Analysis Results

	Analytical solution in the paper	FEA solution in the paper	FEA solution of ALGO
Stress (XX) - psi	28310	26760	28006.79
Displacement - in	0.0127	0.0131	0.0122

NOTE: The stress value here is mesh dependent due to stress concentrations at the hole boundary. The selected mesh provides results like the paper, however, refining the mesh will likely result in higher stresses. A more in-depth study would be required to find out the right mesh size for capturing the exact stress value. This model provides a good starting point to do that.

Conclusions

- The simulation shows good correlation for maximum stress values when compared to the conventional method presented in the paper.
- The model runs very fast so it can be useful to study the effects of different material models and property parameters.
- The model mesh can be further refined to study detailed stress and strain distributions in the plate.
 - i.e. Mesh size and clearances based on your application