

Tensile test of an aluminum 2024-T3 coupon

For questions, please use contact form

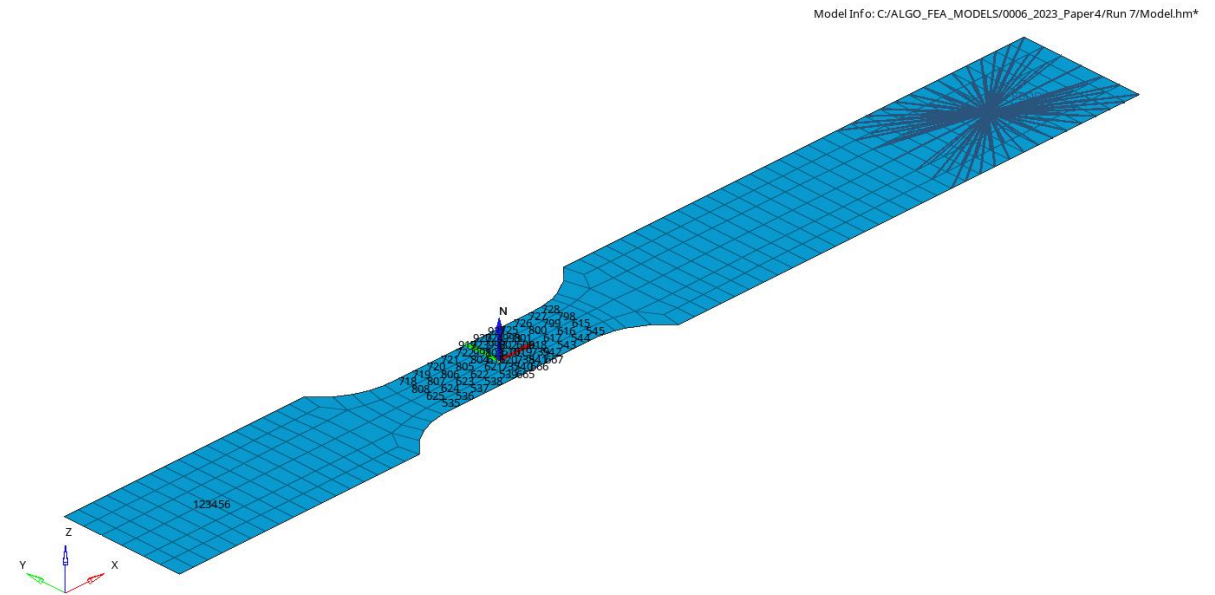
Sanka Dasanayaka



ALGO
Engineering
Simplifying FEA

Model Description

- Simulation of tensile test of an aluminum plate is based on the experimental study documented in the page 40 of following thesis.
 - “FINITE ELEMENT MODELING OF SINGLE SHEAR FASTENER JOINT SPECIMENS: A COMPARISON OF SIMPLIFIED JOINT MODELING TECHNIQUES,” 2014. Accessed: Mar. 02, 2023. [Online]. Available: https://soar.wichita.edu/bitstream/handle/10057/11358/ZINZUWADIA_Chandresh_FL14%20Sequester.pdf?sequence=1
- Tensile test of a monolithic 2024-T3 aluminum sheets

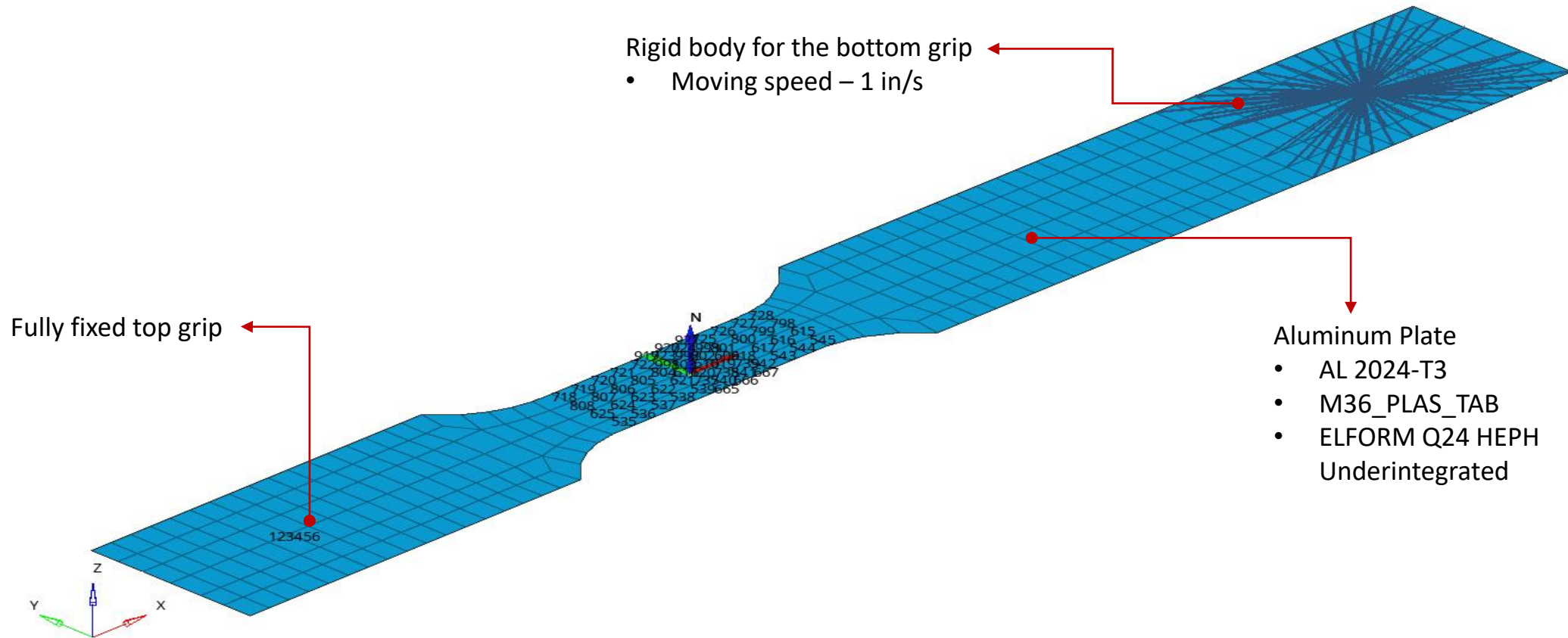


Model Parameters

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	421.05 sec

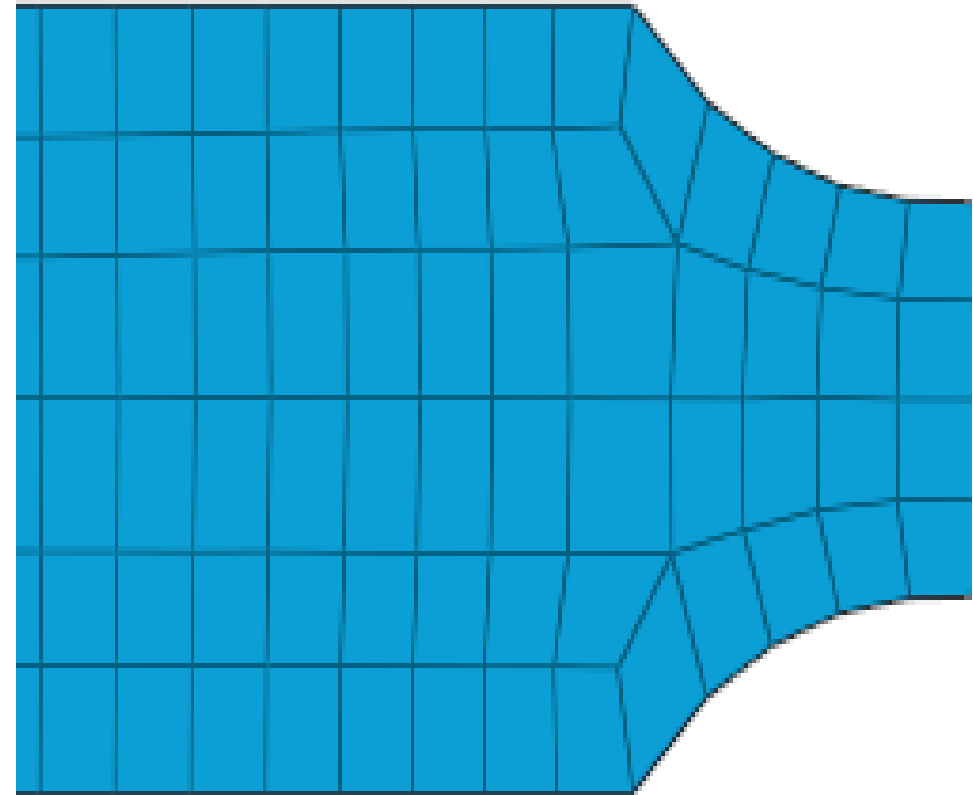
FEA Entities	Type
Analysis Type	Dynamic Explicit
Unit System	kg, mm, ms
Element Type	P1_SHELL
Element Formulation	Q24
Material Type	M36_PLAS_TAB

Analysis Setup



Mesh Quality

- 1.1 mm – min element length
- 0.74 – Jacobian
- 2 – max aspect ratio
- 366 shell elements

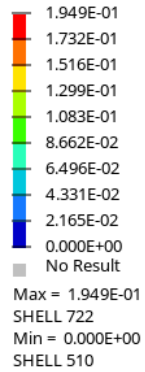


Assumptions

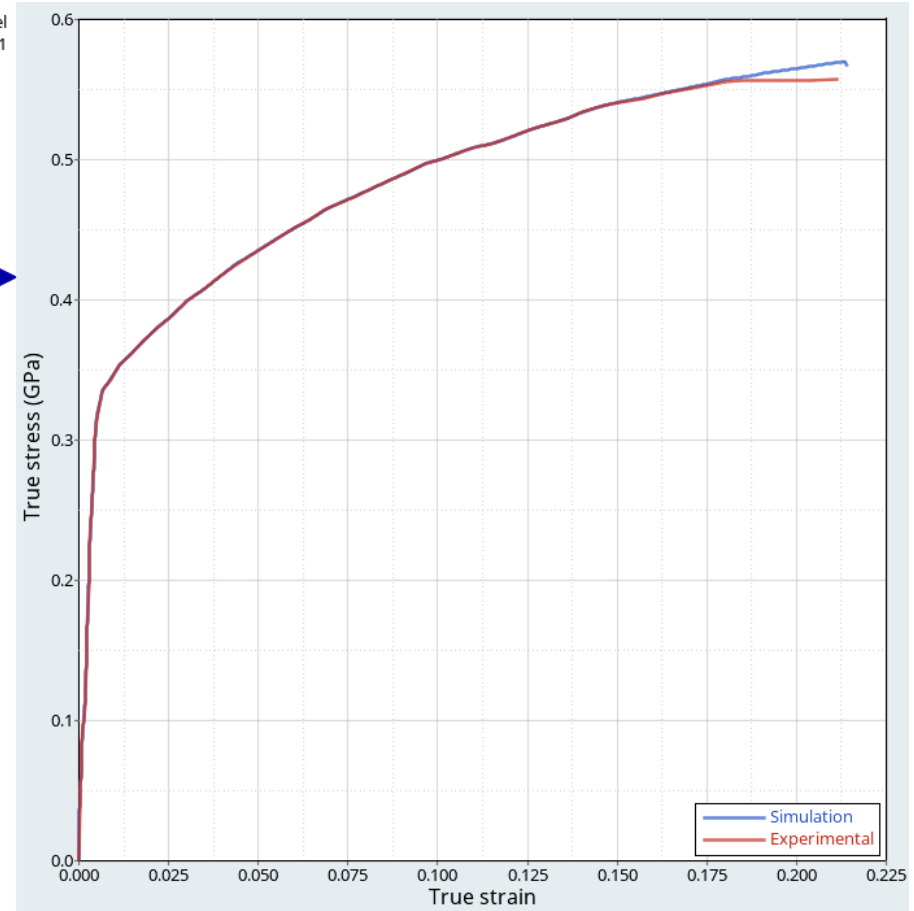
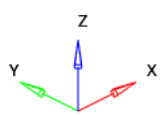
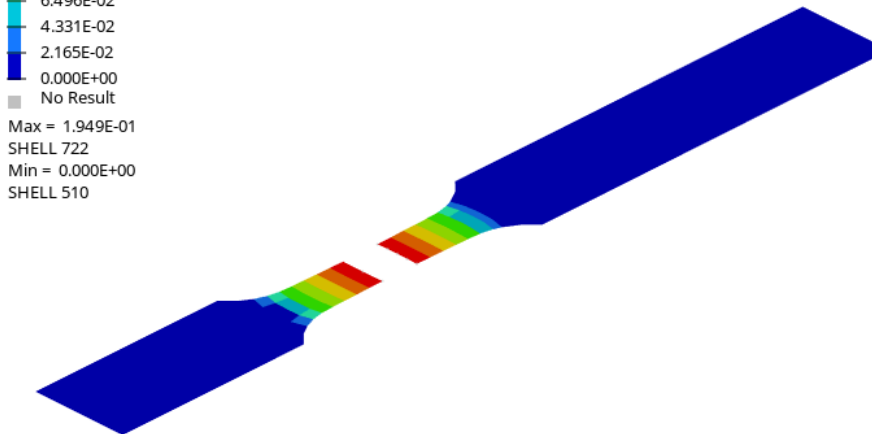
- It is assumed that there is no slip between grips and the plate.
- The rate of displacement of the bottom grip used in the test is 0.08 in/min which is too slow for a simulation. Hence 1 in/s is used as the rate.

Analysis Results

Contour Plot
Plastic Strain(Scalar value, Mid)



1: Model
Loadcase 1 : Time = 3.0000e+02 : Frame 301



Conclusions

- Simulation of tensile test of Aluminum 2024–T3 was conducted.
- The simulation shows good correlation for the stress strain curve when compared to the test data presented in the paper.
- 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
- The model can also be used to simulate tensile test of other materials.