MECH Tech.
SIMULATIONS

Here you can download the Case file, TUI Commands and Complete workbench file for the YouTube video "Ansys Fluent Tutorial for beginners | Sloshing | VOF | Expression in Ansys Fluent | Without UDF". You can make the necessary modifications (Mesh Parameters, Boundary Conditions like Sloshing Acceleration or sloshing speed, Viscosity, Surface Tension and other Fluid Properties) and analyze them for your academic or learning purposes. For further assistance, please feel free to contact us at +91 9488469801 or support@mechtechsimulations.com

Case File

https://drive.google.com/file/d/1b7IhQ_mnzjHWRmUEWH9pGqmLTvj-OdzF/view?usp=sharing

TUI Commands

https://drive.google.com/file/d/1lCMNvzeK-gxgE1lcBgIJfiOPnXA3uAqb/view?usp=sharing

Complete Workbench File

https://drive.google.com/file/d/1r8l77JwTvZwTWsSTb0W_yVZ2tkj9tQB6/view?usp=sharing

I hope you have given a good review to my Google profile. If not, please do so; it will be really helpful to me. (https://g.page/r/CdbyGHRh7cdGEBM/review) Have a happy learning.

Thank you so much for your support!