# MECH Tech. SIMULATIONS

Here you can download the Geometry, Mesh and Workbench files for the YouTube video "Ansys Fluent tutorial for beginners". You can make the necessary modifications (Boundary Conditions, Mesh Parameters, Turbulence Models etc.,) 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

**Geometry File**

https://drive.google.com/file/d/17UGro59SiwUKFe64cyz-K7zbsWSq3dhJ/view?usp=sharing

**Mesh File**

https://drive.google.com/file/d/1g1X1ODJ-yO6vA1KYC9HXLo--zw6p_n6m/view?usp=sharing

**Workbench File**

https://drive.google.com/drive/folders/1_a1bAEDhYkH5ChVK5akTGMRvxTfcIxkG?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!