MECHTech. SIMULATIONS

Here you can download the Geometry and Complete Workbench files for the YouTube video "Ansys Fluent Tutorial | CFD Analysis of Airflow in a Room with Air Curtains | TUI Commands". 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. For further assistance, please feel free to contact us at +91 9488469801 or support@mechtechsimulations.com

Mesh of Horizontal File

https://drive.google.com/file/d/1PgqttrtXsUIOxdkTUPsT-RAJOs86lHNq/view?usp=sharing

Mesh of Horizontal File

https://drive.google.com/file/d/1l_6hXnURa0i-Wbw74i25PRtuuOwzHP15/view?usp=sharing

Case File:

https://drive.google.com/file/d/1fnem_lbEV8aucw4kLlWiVAMxi7APtfYX/view?usp=sharing

Settings:

https://drive.google.com/file/d/13p4_rLJEsH6g9xWO4TnJv3h5xbfUxqq8/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!