MECH Tech. SIMULATIONS

Here you can download the Geometry, Mesh, Case file and Complete Workbench files for the YouTube video "2D CFD Simulation | Luxury Car Aerodynamics | Transient Flow | Ansys FLUENT". You can make the necessary modifications (Geometry, Mesh Parameters, Boundary Conditions, Turbulence Models, Fluid or Material Properties, Flow Reynolds Number 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/1PoVdYz3TYRtcj9nrQhVdhn7_oQHjZufr/view?usp=sharing

Mesh File

https://drive.google.com/file/d/1RiC1oTSOoIKTeBZJdaobJWWFjiP48osP/view?usp=sharing

Case File

https://drive.google.com/file/d/10MJfccf4e4odh4aA1fZfcdqyFcYY0jSp/view?usp=sharing

Workbench File:

https://drive.google.com/file/d/1RR7N6wAJ9PVUuPZ-5d7pKOm06fOVekM9/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!