MECH Tech. SIMULATIONS

Here you can download the Geometry, Mesh, FLUENT Case or Workbench files for the YouTube video "CFD Simulation | Transient Heat Flow | Advection and Convection | Ansys FLUENT | Ansys Workbench". You can make the necessary modifications (Boundary Conditions, Mesh Parameters, Turbulence Models, Transient Time Settings 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/1jbiA7pYo3_sydWWKiQ8-OjUqPBtLdt7A/view?usp=share_link

Mesh File

https://drive.google.com/file/d/17R1Qr4X2lxj7Q4S-RrJ2tFMBGf9XFgON/view?usp=share_link

Case File

https://drive.google.com/file/d/1sc3PAaWsfBdCLLn_U4IfS5gGwU6jNwwy/view?usp=sharing

Workbench File

https://drive.google.com/file/d/1QFpIAXUS3OYizhv9Mh2MaaGlINs5-D28/view?usp=share_link

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!