

MECH Tech.

SIMULATIONS

Here you can download the Geometry, Case file and Complete workbench file for the YouTube video "[Ansys Fluent tutorial for beginners | Solar Air Heater | Ansys Student 2023 | Workbench](#)". You can make the necessary modifications (Mesh Parameters, Boundary Conditions like Inlet Temperatures, Air Velocity, Mass Flow rate 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@mechtechsolutions.com

Geometry File

<https://drive.google.com/file/d/1oV14vWgyeyBGS4tjamEUiCygNltdQ2xm/view?usp=sharing>

Case File

<https://drive.google.com/file/d/1NuQCzSzfJXnult2hIWeuOlcJULPBjxO/view?usp=sharing>

Complete Workbench File

<https://drive.google.com/file/d/1j5o1MOifoUgH9-dZ57JzSJHCEhCtkpai/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!