

MECHTech.

SIMULATIONS

Here you can download the Geometry file and Complete workbench file for the YouTube video "[Shell and Tube Heat Exchanger | Ansys workbench tutorial for beginners | Ansys FLUENT Tutorial](#)". You can make the necessary modifications (Mesh Parameters, Boundary Conditions like Temperatures, Fluid flow rates, velocity, Reynolds Number, Fluid Properties etc.,) 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/1BEapW_Z-Vo3zCuskYB2uf-B136sPKU7t/view?usp=sharing

Complete Workbench File

https://drive.google.com/file/d/1apyXlja_JHvCAZ6ijM61V2x0Ga9V0qfY/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!