# MECH Tech. SIMULATIONS

Here you can download the Geometry, Mesh, Case file and Complete Workbench files for the YouTube video "Step by Step Tutorial for Heat Transfer Analysis in Ansys FLUENT | Setting Convergence Criteria". You can make the necessary modifications (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/15WinIOzaeXiIJoGQe__olB9D9R5Gejyo/view?usp=sharing

**Mesh File**

https://drive.google.com/file/d/1lzAQiyLTCxOzJEUhHLiXyI26dfsB1MN-/view?usp=sharing

**Case File**

https://drive.google.com/file/d/1PaOOzKmOAqBUi03MCHVdd9Sb6OU-ZpzJ/view?usp=sharing

**Workbench File:**

https://drive.google.com/file/d/1IIsgTqKN_BCiX4awI96OnAq1YmOYSY1u/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!