MECH Tech.
SIMULATIONS

Here you can download the Case file for the YouTube video "Ansys Fluent Tutorial for Beginners | Simulation of Venturimeter | An Experiment in CFD". The mass flow rate can be varied and the flow rate can be verified. You can practice for your academic or learning purposes. For further assistance, please feel free to contact us at +91 9488469801 or support@mechtechsimulations.com

Case File

https://drive.google.com/file/d/1TthRBfuxdXPhFfIzknj30mzlYUYfDfbt/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!