MECHTech. SIMULATIONS

Here you can download the Geometry file for the YouTube video "Create fluid domain from solid geometry in Ansys Workbench | Ansys Fluent | Suppressing a geometry". 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/1LlPfem1eu9dhFy37hEYZgYB0gWSp7jcc/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!