

# MECH Tech.

## SIMULATIONS

Here you can download the Geometry and Mesh files for the YouTube video "[Ansys Fluent Tutorial for beginners | Multiphase Flow | Three Phases | Ansys Workbench](#)". You can make the necessary modifications (Boundary Conditions, Mesh Parameters, Turbulence Models 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](mailto:support@mechtechsolutions.com)

### Geometry File

<https://drive.google.com/file/d/1qGWf-2SeZ85PGgkOcvkesTvcl1JQ5Hk5/view?usp=sharing>

### Mesh File:

<https://drive.google.com/file/d/1rpseivCez3zd-bvU24fgbdnSQp-ULdvn/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!