

MECH Tech.

SIMULATIONS

Here you can download the Mesh File, Case File and TUI Commands for the YouTube video "[Ansys Fluent Tutorial for beginners | Transient 3D | VAWT | Savonius Turbine | Step-by-step procedure](#)". You can make the necessary modifications (Mesh Parameters, Boundary Conditions, Blade Speed -TSR [Tip Speed Ratio], 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@mechtechsimulations.com

Mesh File

<https://drive.google.com/file/d/1Q2pcpW6uZJvQTXAlyBNIRp6CeXCvD7ga/view?usp=sharing>

Case File:

<https://drive.google.com/file/d/15qBvOFKiu53hbOFBkOGLSx68w8orK9S/view?usp=sharing>

TUI Command file:

https://docs.google.com/document/d/1_RVdHEENQpiwVBe4gYlmonuxlhmY388Z1FAzIsPrFtg/edit?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!