

# MECHTech.

## SIMULATIONS

Here you can download the Geometry and Complete workbench file for the YouTube video "[How to use Ansys Workbench? | Static structural analysis | Step by Step Tutorial](#)". You can make the necessary modifications (Mesh Parameters, Boundary Conditions like Forces, Fixed points and Material Properties) 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 – Without Footrest**

<https://drive.google.com/file/d/1ZAUX6QG4uQW9IaG4QgVMn3yr9MubyKCF/view?usp=sharing>

### **Geometry File – With Footrest**

[https://drive.google.com/file/d/1dqVptcNr9Tn7mixnhEJC\\_MvQSxp\\_BhfD/view?usp=sharing](https://drive.google.com/file/d/1dqVptcNr9Tn7mixnhEJC_MvQSxp_BhfD/view?usp=sharing)

### **Complete Workbench File**

<https://drive.google.com/file/d/1yzQxcUlyD6MFAjL18JU5GgW-ZG9nTgVS/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!