

MECHTech.

SIMULATIONS

Here you can download the Geometry and Complete Workbench files for the YouTube video "[Static structural analysis in ansys workbench | Spaceclaim Tutorial | Ansys 2022 Student](#)". You can make the necessary modifications (Geometry, Boundary Conditions, Mesh Parameters etc.,) and analyze them for your academic or learning purposes. For further assistance, please feel free to contact us. For further assistance, please feel free to contact us at +91 9488469801 or support@mechtechsimulations.com

Solidworks Geometry File

<https://drive.google.com/file/d/1cAHroYmightcS--68EivDfUHMDagGXh3/view?usp=sharing>

Spaceclaim Geometry File

https://drive.google.com/file/d/1OACaYQzASN4NNG1_ABUw9oHF_rokROy5/view?usp=sharing

Workbench File:

https://drive.google.com/drive/folders/10vTx6cBUW_EdUNS4y2ojMVFIJ3DLqD7?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!