

# Download Ansys Workbench Contact Analysis

Bonded Contact between Shell Faces in ANSYS® Mechanical (Workbench) v14.5 . Posted in Tips & Tricks - Finite Element Analysis (FEA) articles. ANSYS Mechanical (Workbench) has many settings for contact between surface body (shell) faces. Contact for Projects & online training Mobile/WhatsApp: +91-9481635839 | INDIA Email: [engineeringtutorsdesk@gmail.com](mailto:engineeringtutorsdesk@gmail.com) Skype: [engineeringtutorsdesk](https://www.skype.com/en/contacts/individual/engineeringtutorsdesk) ANSYS Workbench Tutorial Video on Structural ...Step by step procedure of how to do static structural analysis of frictional contact (clamp) ANSYS 13 workbench. We can do wear analysis due to frictional contact between its surface. Hello all simulation geeks... Just for fun, I made a comparison between ANSYS and Creo Simulate 3, to see what the new frictional contact can do in Creo 3.