

---

# Polyflow In Workbench Tutorial

???? ANSYS ??? ????? ? ?? ??. ANSYS Fluent  
Software CFD Simulation

???? ANSYS ??? ????? ? ?? ??

May 13th, 2018 - ?? ansys ?? ?????? ??? ????? ????? ?? ?? ?? ????? ?? ??? ????? ???

??? ??? ?? ? ? ??"ANSYS Fluent Software CFD Simulation

May 13th, 2018 - ANSYS Fluent CFD software includes well  
validated capabilities to deliver fast accurate results for the  
widest range of simulations'

Copyright Code : [3IL89rlmvPNzgUR](#)