
ANSYS Fluid Dynamics Tutorial Inputs.zip ##BEST##



1. Start FLUENT. In the FLUENT project workspace, in the FLUENT Process Settings window, click Run\FLUENT in the Processes menu, enter a name, and select a time step. Select Outflow from the Start FLUENT menu to . 2. Start the Fluid Solver dialog by selecting Fluid Solver\Open Fluid Solver in the FLUENT Process Settings window. For the Fluid Flow Type, select Steady State. 3. Select the Steady State project in the Processes window. In the Fluid Properties menu, select Watertight Geometry. Select the Steady State project cell in the Fluid Schematic window, select the Design cell in the Fluid Solver dialog, then select Simulation in the Fluid Solver dialog, as shown in the image below. 4. Click the Load icon next to the Input file to . 5. Click Add to load the tutorial files (listed in the "Adobe Acrobat Reader Required" section below) . 6. In the Fluid Solver dialog, select the input file, I13_061_O11_002_S30_004_V01_0010.ice, in the File Path pane, as shown in the image below. 7. Click OK . 8. Select Save Schematic in the Fluid Solver dialog, enter a name in the Name box, and select OK. 9. Select Fluid Solver\Save . 10. Click Run\FLUENT. 11. Select View Simulation Results\Done in the FLUENT Process Settings window, then click Close. 12. Select Selection from the Simulation Results\Options menu, then click Close. The Fluid Solver dialog should look like the following image. 13. Select View Simulation Results from the Simulation Results menu. 14. The simulation results are displayed in the right column of the FLUENT workspace. 15. Select Analysis from the Simulation Results menu. The analysis results are displayed in the left column of the FLUENT workspace. A: I have figured out the issue as following. It was a 10.4.2 bug. Actually I had to download 10.4.2 from the ANSYS web site, then replace all the ANSYS files in the Ansys Fluid Dynamics Tutorial Inputs.zip. ANSYS 9.3 SP0 570a42141b

2 / 3

