ANSYS Flow over an Ahmed Body - Validation

Author(s): Sebastian Vecchi, ANSYS Inc.

Verification & Validation

An excellent way of validating simulations is by comparing them to research papers which are relevant. Since the Ahmed body is so widely studied and used as validation, it is not difficult to find supporting evidence. For this tutorial, the information gathered from the simulation will be compared to "Embedded Large Eddy Simulation of Flow around the Ahmed Body" by Domenico Caridi done in ANSYS FLUENT. Below is a contour of the pressure on the rear surface of the Ahmed body. This can be compared to our model by creating a similar contour in our simulation.

Add a **Contour** from the **Add** drop down menu and select the top, side and rear faces as the **Location**. After changing the **Variable** to **Pressure**, the following contour plot will be created.

While qualitatively similar, the numerical results are significantly different than those found in the validation. By refining the mesh, the results can be calculated to a more accurate degree. Go back to the **Mesh** task in the workflow and increase the **Mesh resolution** all the way up. Go back to the **Results** and press **Evaluate Results**. Travel back to the Pressure contour.

These results compare more favorably to the FLUENT results, though they are still significantly different. However, the mesh in the FLUENT study was much more refined than the one used here, especially in the critical wake region immediately behind the Ahmed body. Additional refinement of the AIM mesh would further improve the results and is recommended practice for external flow around vehicles.