

ANSYS Fluid Flow over a Bluff Body - Validation

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

Verification & Validation

In order to validate the results from the tutorial, they must be compared to a source of credibility. The results will be compared to those found in "Volvo Bluff-Body Stabilized Flame." by Dong, Yunqing, Christian Lübon, and Thomas Frank of ANSYS, which include experimental results. Below is the velocity profile on the centerline downstream of the triangular flame holder. It shows that, immediately after the body, the velocity is about half of the inlet velocity, momentarily zero and then slowly increases until it has just about reached the inlet velocity.

In order to find data within the AIM simulation that will support this, a velocity contour will be created with a plane that bisects the flow volume horizontally. Using the **Add plane** button, create a plane that horizontally bisects the flow volume. In the **Plane** panel, position it precisely by setting the **Origin and orientation definition** to **Based on a reference frame** and the Reference frame plane under **Origin and Orientation** to **ZX**. Then, create a contour with the new plane as the **Location** and the **Variable** as **Velocity X**. The contour will look like the image below.

After analysing the picture above, it can be confirmed that the velocity directly downstream of the flameholder it is about half of the original inlet velocity, momentarily zero, and then slowly increases back to the inlet velocity.

References

Dong, Yunqing, Christian Lübon, and Thomas Frank. "Volvo Bluff-Body Stabilized Flame - Part 1: Non-reacting Flow Investigations with AIM Fluids 16.2." ANSYS EKM Server. ANSYS, 2 July 2015. Web. 17 July 2017.

[Go to all ANSYS AIM Learning Modules](#)