ANSYS Compressible Flow over a Wing-Body Junction -Startup

Author(s): Sebastian Vecchi, ANSYS Inc.

- **Problem Specification**
- 1. Start-Up
- 2. Geometry
- 3. Mesh
- 4. Physics Setup
- 5. Solution/Results

Start-Up

A few words on the formatting on the following instructions:

- 1. Notes that require you to perform an action are colored in blue
- 2. General information is colored in black, but does not require any action
- 3. Words that are **bolded** are labels for items found in ANSYS AIM

We are ready begin simulating in ANSYS AIM. Open ANSYS AIM by going to Start > AII Apps > ANSYS 18.1 > ANSYS AIM 18.1. Once you are at the

?

4. Most important notes are colored in red starting page of AIM select the Fluid Flow template as shown below. A Unsaved Project - Workbench _ × • . Learn Structural Fluid Flow Getting Started - Part 1 Getting Started - Part 2 ? Overview of AIM ? ANSYS AIM Video Library Recent Thermal Electric Condu Fluid_SDuct svecchi\Doci uments\NEW Al.. C-ALLee SteadyCylinder_V2 :\Users\svecchi\Desktop Fluid_Cylinder C:\Users\svecchi\Documents\NEW AL. Magnetics Polymer Extrusion Fluid_UnsteadyCylinder nts\NEW AI. C:\Users\svecchi\Docu Open Other File Fluid-Structure Interaction Fluid-Solid Heat Transfer

You will be prompted by the Fluid Flow template to either Define new geometry, Import geometry file, or Connect to active CAD session. Select Imp ort geometry file and press Next.

Go to Step 2: Geometry

Go to all ANSYS AIM Learning Modules