

Transition Duct - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

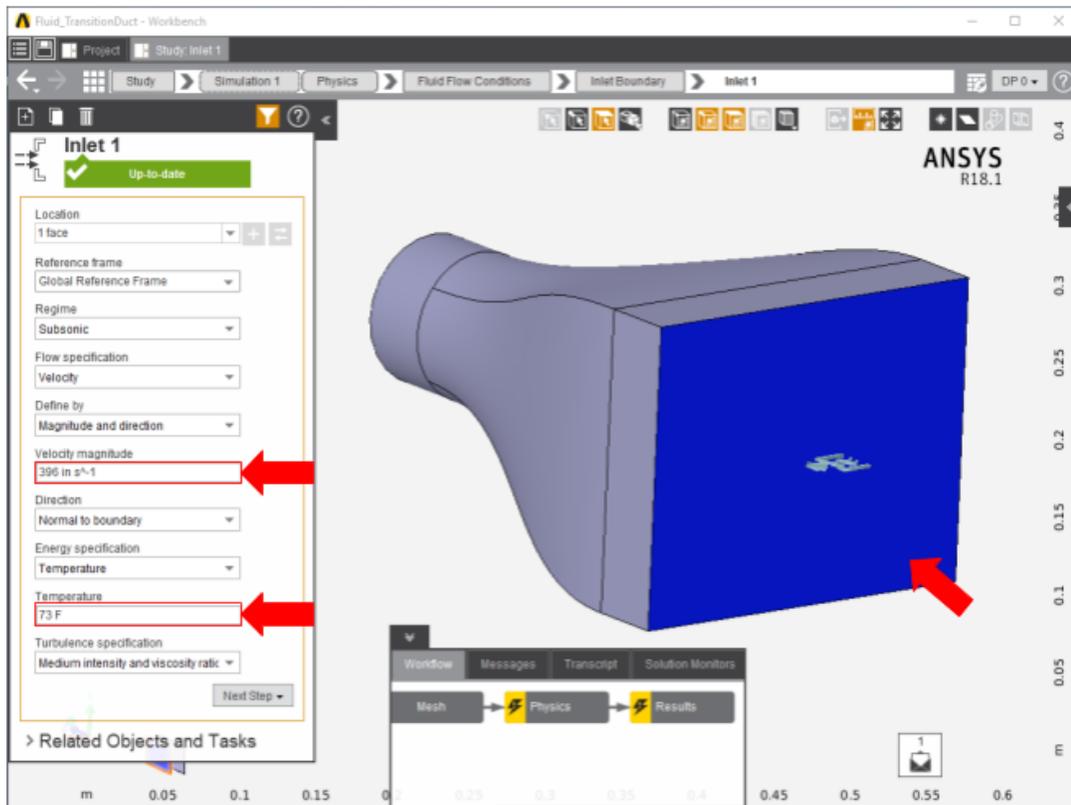
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results

Physics Set-Up

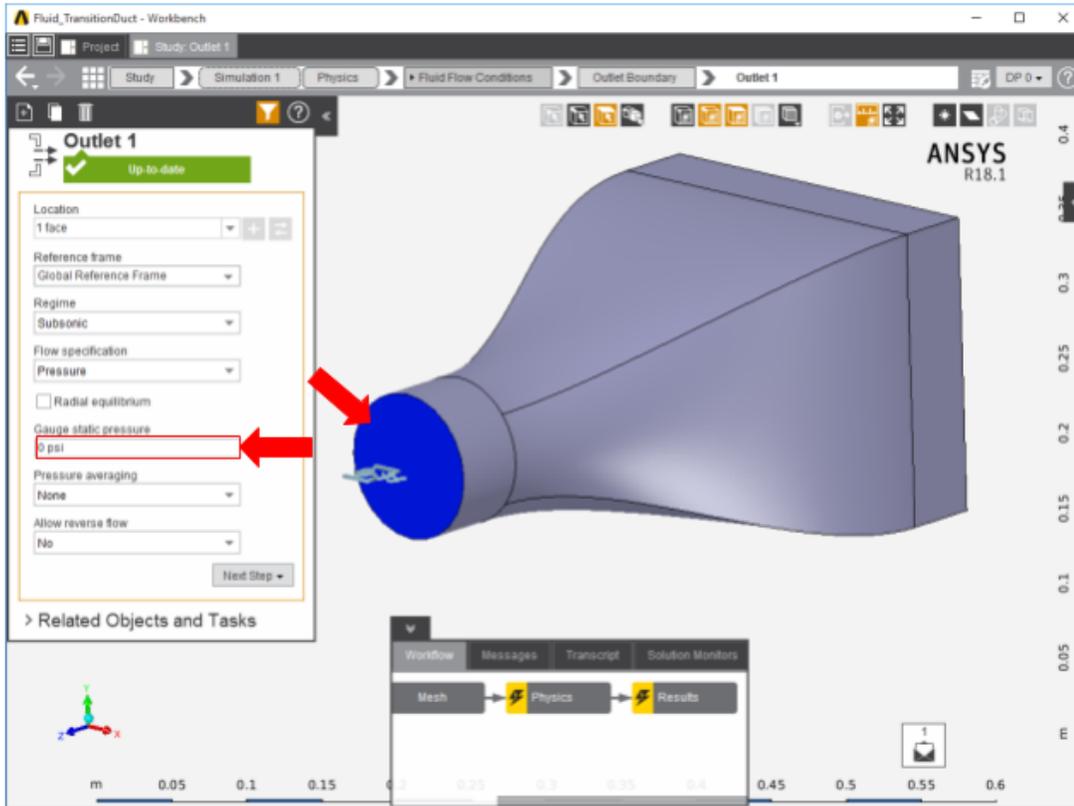
Select the **Physics** task in the **Workflow**.

Boundary Conditions / Forces

First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select **Inlet**. Then, using the **face selection tool**, define an inlet at the wide end of the duct. Input the **Velocity magnitude** as 33 [ft s⁻¹] and a **Temperature** of 73 degrees [F]. **When entering the Velocity magnitude, be sure to enter the units correctly.** After entering the value in "ft s⁻¹", it will be automatically converted to the default units of "in s⁻¹" and displayed accordingly.



Once the inlet is defined, the outlet is next. In the same **Add** menu, define an **Outlet** at the small end of the duct. Assign a **Gauge static pressure** of 0 psi.



Next, a **Wall** condition must be added to all surfaces that are not already defined. **Wall** can be found in the same **Add** menu as the previous conditions. By default, the wall condition is thermally insulated. AIM will automatically create the **Wall** once the option is selected, automatically selecting every face that doesn't already have a constraint on it.

[Go to Step 5: Results](#)

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