Turbulence Intensity

Problem/Description:

What is turbulence intensity and how is it calculated in Fluent?

Solution:

As an inlet boundary condition, you might want to use the turbulence intensity. It is often used as an index indicating the strength of the turbulence in the turbulence field.

Turbulence intensity is expressed by the percentage ratio of the flow rate variation with respect to the average flow rate (root mean square).

In general:
10% or more: the disturbance is very large
less than 1%: turbulence is very low

The turbulence intensity is actually measured empirically, and there is also a way to give as a boundary condition, but it can be difficult is not to estimate. For example, for a duct:

$I = 0.16 * Re ^ (- 1/8)$

Where:
I = turbulence intensity [%]
Re = Reynolds number (representative length hydraulic diameter)

You may guess why such formula is used. In the figure below shows the turbulence intensity in the case of a circular tube, which is calculated by the formula.

Note: This is the only prediction formula. Distance and physical properties from the entrance of the circular tube and disturbance condition will differ depending on the situation.

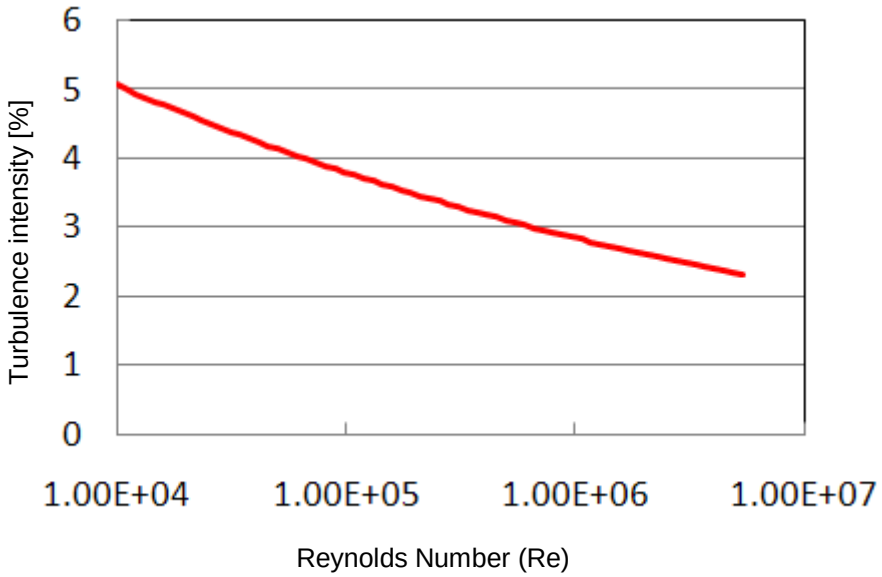

Figure 1: Turbulence intensity of the duct

Furthermore, turbulence intensity specified in the inlet boundary rather than being used as it is, numerical calculation is converted to a boundary value of k (turbulent kinetic energy) transport equation. The following relationship between k and turbulence intensity (I) is used in Fluent:

$$k = \frac{3}{2}(u_{\text{avg}} I)^2$$

Where:

uavg is the flow rate
I is the ratio rather than a percentage (0.1 instead of 10%)