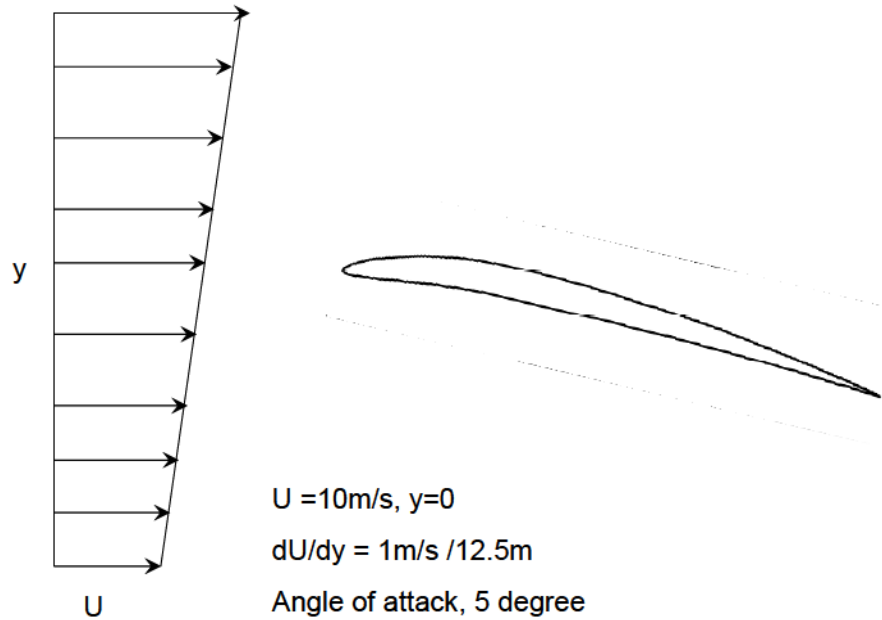


2. STAR-CCM+ user coding

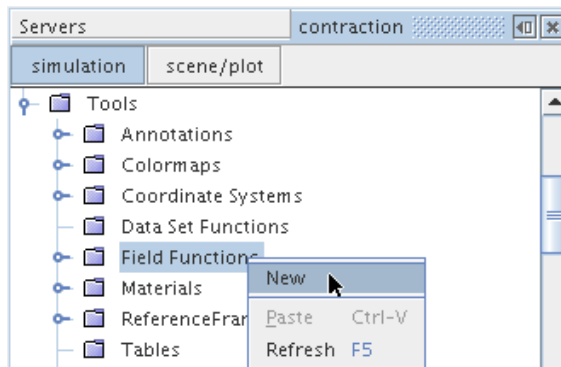
An airfoil in a free shear layer (a wake, or due to the stratification of the atmosphere).

Design the inlet conditions for this case. Then apply this inlet condition for a computation of an airfoil (you can use your coursework airfoil).



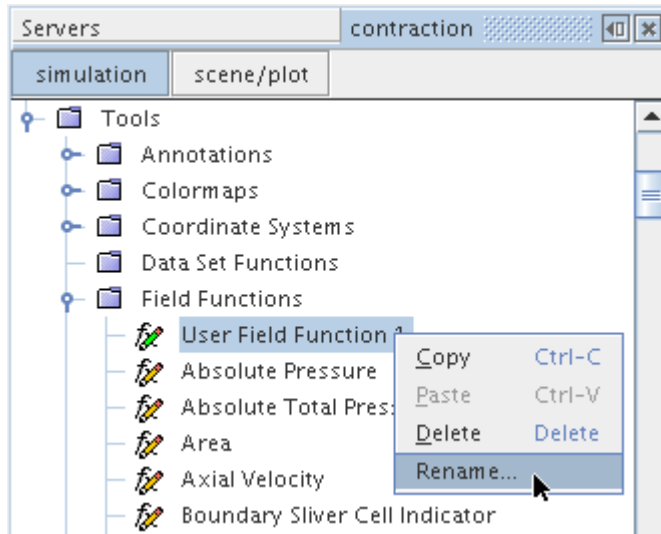
The field function for this profile is implemented as follows:


- Right-click the **Tools > Field Functions** node and select **New**.

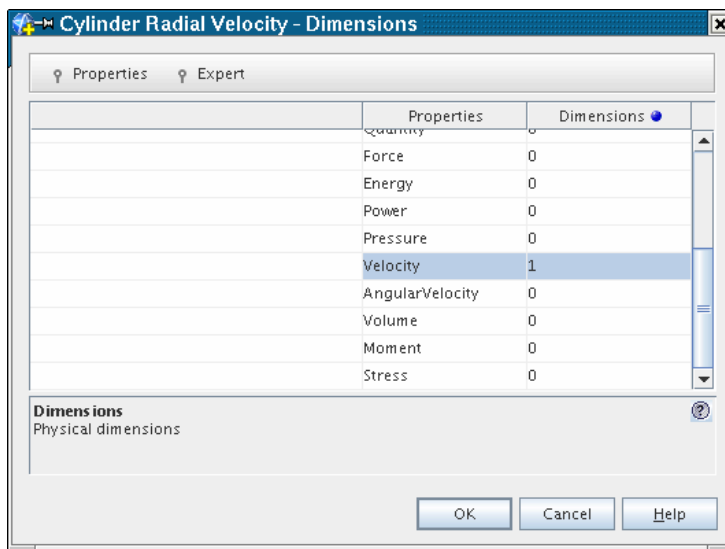



A new **User Field Function 1** node will appear in the **Field Functions** tree.

- Right-click the **User Field Function 1** node and select **Rename...**



- Change the name to inflow_bc0 in the Rename dialog and click **OK**.
- In the Properties window, set the **Type** to **Vector**.
- Continuing in the Properties window, click the  (**Custom Editor**) button for the **Dimensions** property. The Dimensions dialog will appear.

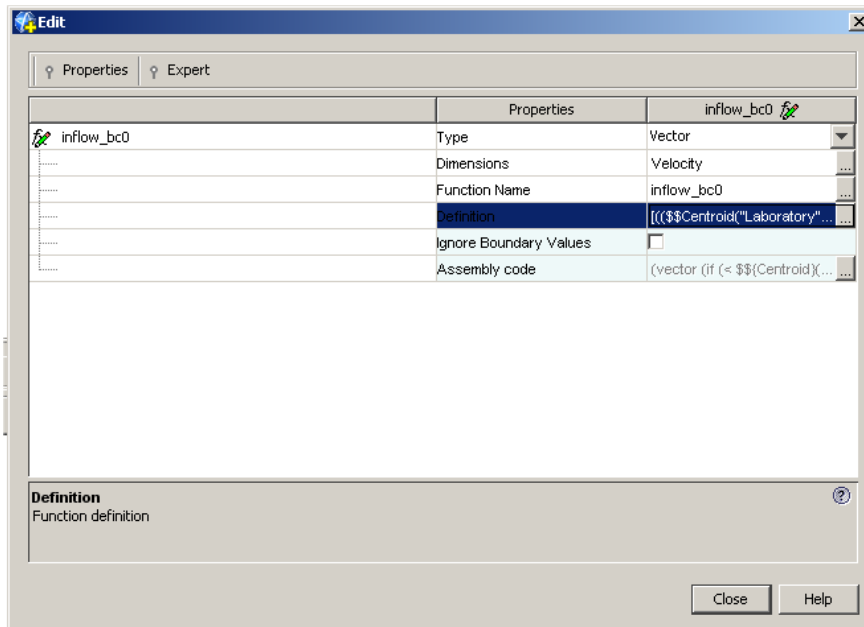


- Set **Velocity** to 1 as shown and click **OK**.
- In the Properties window, set **Function Name** to inflow_bc0.
- Click the  (**Custom Editor**) button for the **Definition** property. The Definition editor dialog will appear.
- Enter the following definition:

```
[(($$Centroid("Laboratory")[1] < 0) ? (10+1/12.5*$$Centroid("Laboratory")[1])*
cos(5.0/180.*3.1415927) : (10+1/12.5*$$Centroid("Laboratory")[1])*
cos(5.0/180.*3.1415927)), ((10+1/12.5*$$Centroid("Laboratory")[1])*
sin(5.0/180.*3.1415927)), 0]
```

Note that the function `$$Centroid("Laboratory ")` returns a cartesian vector of the face centroid coordinates in local coordinate system Laboratory; the subscript operator `[]` is appended to access a single component of the centroid vector (with an index of `[0]`, `[1]`, or `[2]`).

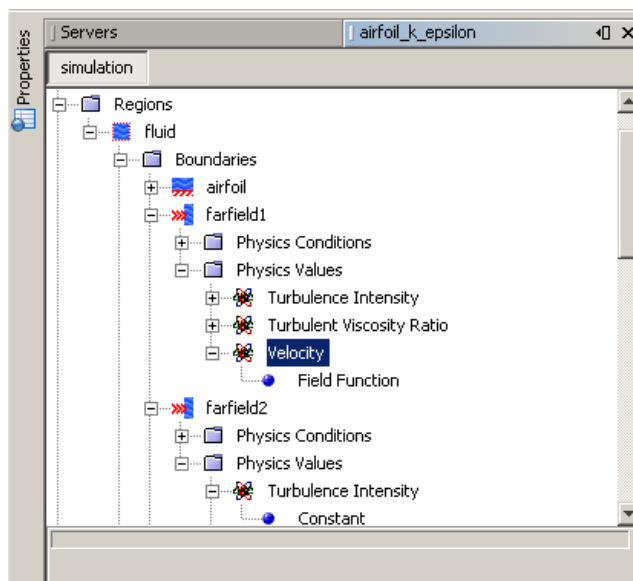
The Properties window will be as shown below when you are done.



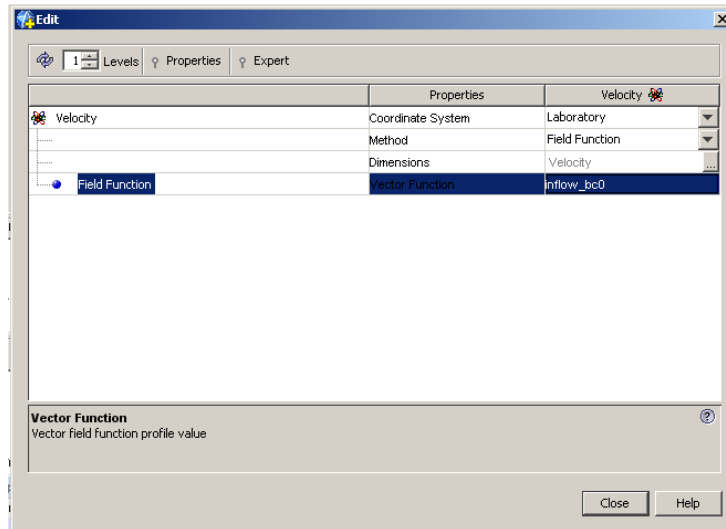
Setting Boundary Conditions and Values

In this section we will set physical values where necessary,

- Expand the **Regions > Fluid > Boundaries > farfield1 > Physics Conditions** node.



Double click the Velocity node, and setup as below,



You should also setup appropriate turbulence intensity and turbulence viscosity ratio for this inlet.

Setup other inlet, e.g. farfield2.