

## Example 5. Modelling of Heat Transfer in a T-Junction of Pipes

### 5.1. Introduction and Objectives

The concept of heat is relevant to understand in many fields of engineering, including civil, mechanical and chemical engineering. The heat transfer process (heating and cooling) is an essential part of many industrial processes and plays a key role in such aspects as for example material selection, machinery efficiency and chemical reaction kinetics. In general, there are three distinct mechanisms of heat transfer, i.e. conduction (transfer of energy via solid material by direct contact, without mixing of any material), convection (transfer of energy by the bulk mixing, may be natural or forced) and radiation (e.g. light, infrared or ultraviolet which emanate from a hot body). In this example we will consider a turbulent heat transfer in a T-junction. In particular, we will study the interaction between two liquid streams of water with different temperatures (convection mechanisms) in a T-junction of circular cross-section. The main objectives of this exercise are as follows:

- create 3D geometry of the flow domain from 2D sketches located in different planes,
- illustrate the setup of the simulation with heat transfer between two streams – defining the temperature of the liquid streams,
- visualization of results – contours of fluid velocity and temperature, mean value of temperature at the outflow.

### 5.2. Problem Description

Details of the analysed flow problem are shown in Figure E5-1.

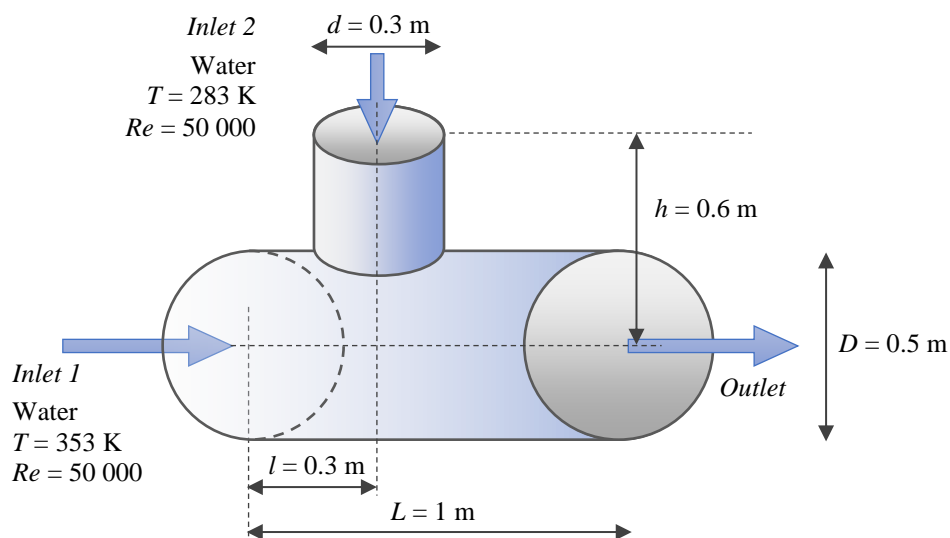
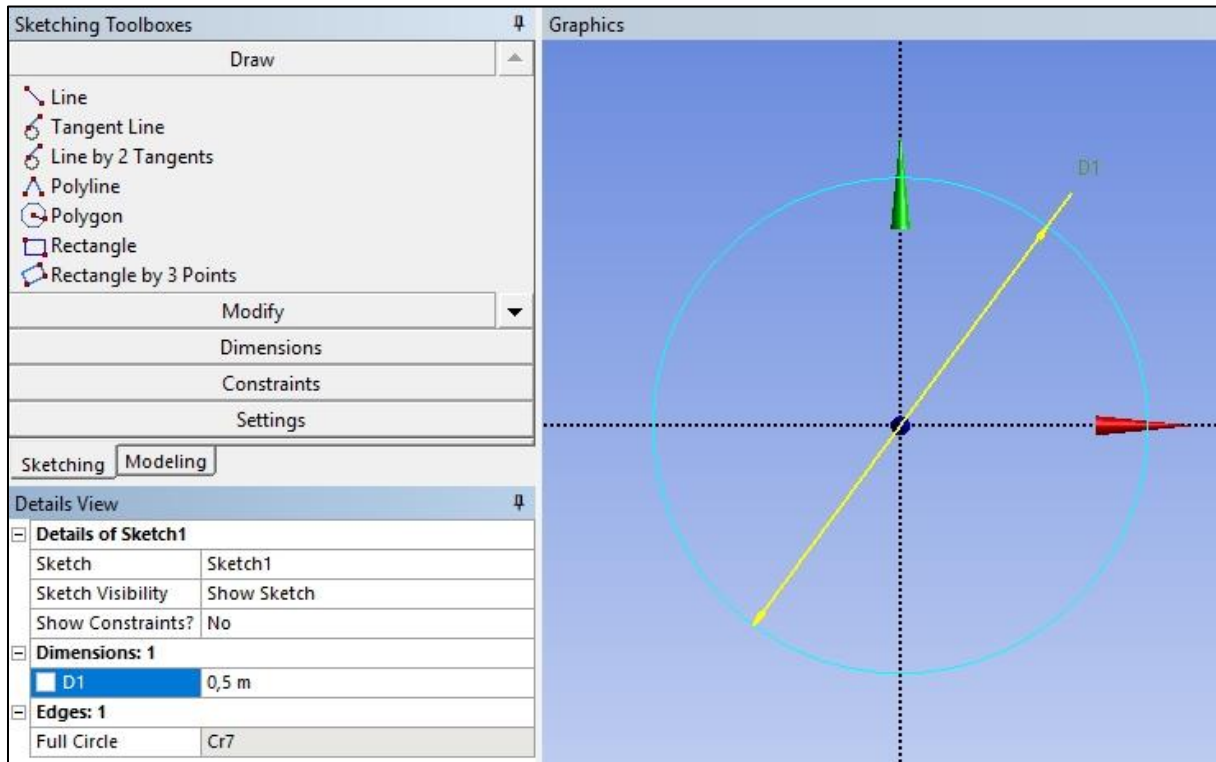


Figure E5-1. Problem description.

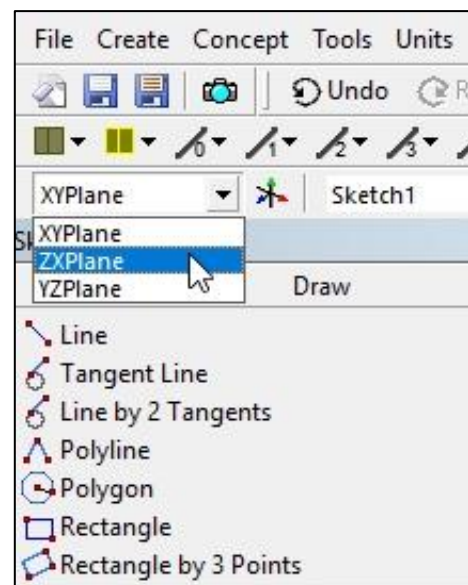
### 5.3. Geometry

Open *Ansys DesignModeler* in order to draw the geometry of the flow domain and go to the *Sketching Toolboxes* since we will draw a 2D sketch first. Make sure that *Auto Constraints* options are enabled and setup a proper view by clicking *Look At Face/Plane/Sketch* icon from the top ribbon. From the *Draw Toolbox* select *Circle* and draw a circle starting in the origin of the coordinate system. Then measure the diameter of this geometry using the *General* or *Diameter* dimension and set the value as 0.5 m (Fig. E5-2).



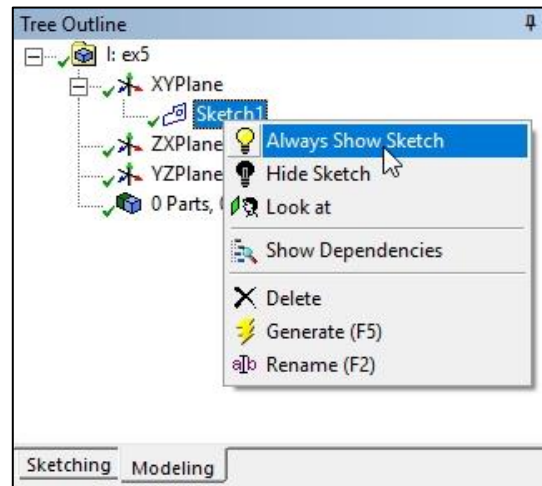
**Figure E5-2.** First circle drawn in the *XYPlane* with its dimensioning.

This circle is a base of the first pipe with bigger diameter and it was drawn in the default *XYPlane*. The steps of drawing the second circle with smaller diameter are analogous, however it is perpendicular to the first one, therefore it has to be drawn in other plane. In order to change the plane, move the cursor in the area on the top ribbon where the active plane is indicated and from the dropdown list select the *ZXPlane* (Fig. E5-3).




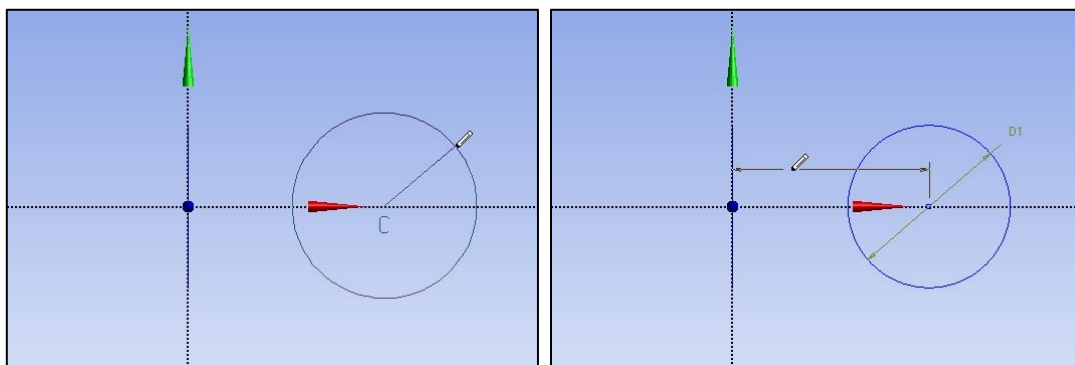
**Figure E5-3.** Changing the plane.

After changing the plane, the first circle will disappear from the *Graphics* window. It does not mean that it was removed, it is just not visible in this view. If you want to make the first circle visible go to the *Modeling* tab, select the *Sketch1* from the *Tree Outline*, click the RMB and select the “*Always Show Sketch*” option. It will make your sketch always visible, even if you will change the active plane. Remember that in this way it is possible to see all of the non-active geometries, but you cannot modify them. Any modifications are available for active planes and sketches only.



**Figure E5-4.** *Always Show Sketch* option.

If you did the operations described above the program automatically changed the active plane for *XY* since exactly there the first circle was drawn. Therefore, again change the plane for *ZX* and press the *Look At Face/Plane/Sketch* icon . Then from the *Draw Toolbox* select the *Circle* and draw it. Please note that although the first circle started in the origin of the coordinate system, this is not the case for the smaller circle. It does not start in the origin of the coordinate system therefore draw it in such a way that the centre of the circle will lie on the *X* axis but at the same time it will be moved away from the *Z* axis by a certain distance (Fig. E5-5a). Now measure diameter of the circle and then move it in a proper position. In order to move the circle, measure the distance from the *Z* axis to the circle centre. From the *Dimensions Toolbox* select the *Horizontal* tool and click with the LMB on the circle centre (it will be highlighted with yellow colour), then click on the *Z* axis, and then in any free place of the *Graphics* window that is moved by a certain distance from your geometry, i.e. in the place where you want to drop a dimension line (Fig. E5-5b). In the *Details View* window enter the proper values of measured distances, i.e. 0.3 m for the diameter and 0.3 m for the horizontal distance. Your 2D sketch is ready.



**Figure E5-5.** Drawing the second circle with its dimensioning.

You can check the view of the 2D geometry (Fig. E5-6) by clicking the isometric view (blue sphere in the right bottom corner).

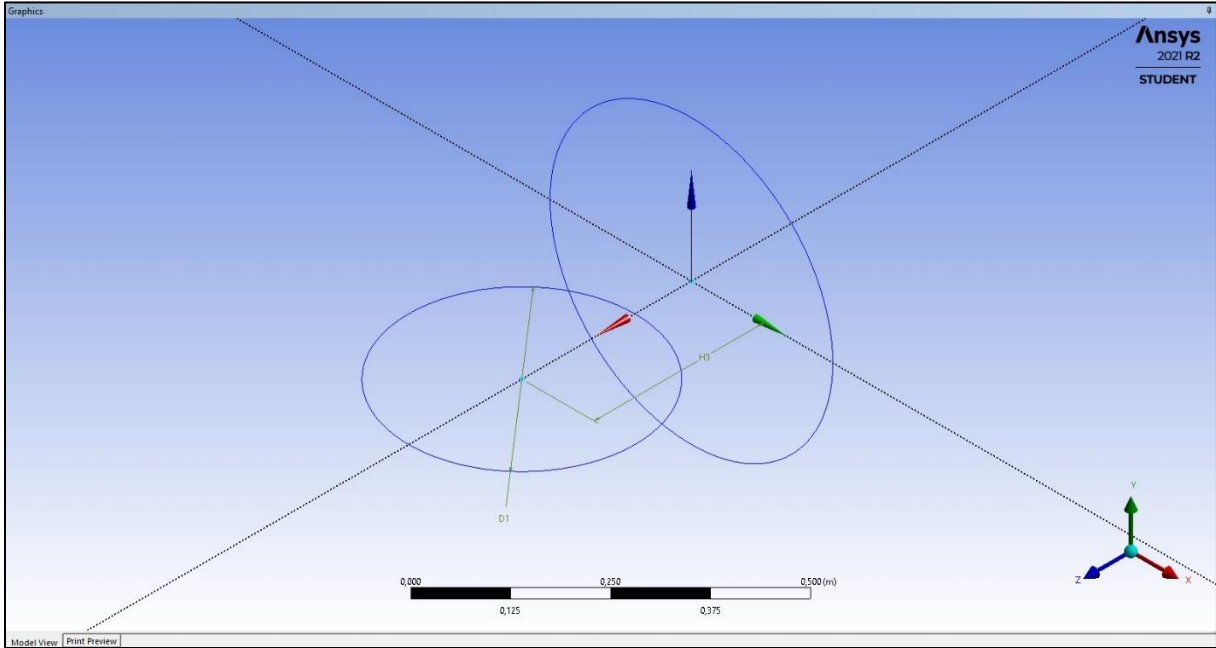


Figure E5-6. View of the 2D geometry.

When 2D sketch is ready launch the *Modeling* tab in order to create 3D geometry. To do this use *Extrude* option described in *Example 3*. Here you have to use *Extrude* twice, separately for each pipe in T-junction. According to the *Problem Description* the bigger pipe has to be extruded for 1 m, while the smaller one for 0.6 m. The view of final 3D geometry is shown in Fig. E5-7. You can close *Ansys DesignModeler*.

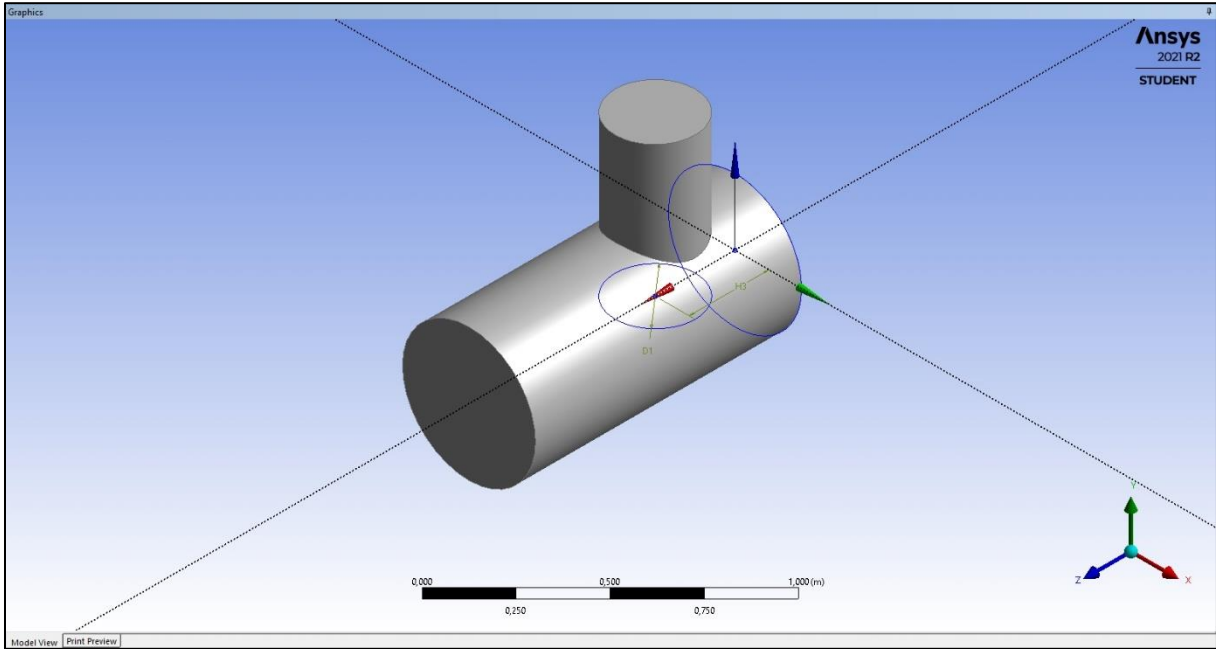


Figure E5-7. 3D geometry representing the flow domain.

## 5.4. Mesh

Open the *Ansys Meshing*, rename the geometry as *T-junction*, and set the *Material* as *Fluid*. Next, choose the *Mesh* from the *Tree Outline*. Ensure that in the *Details of "Mesh"* the *Physics Preference* is set as *CFD* and *Solver Preference* as *Fluent*, and confirm by clicking *Generate Mesh*. In order to modify the grid from the *Top Menu* select the *Mesh* tab and the *Sizing* option. In the *Details of "Sizing"* – *Sizing* dialog box define the geometry. In this example we will use *Body sizing*. From the *Graphics Toolbar* select the *Body* option that allows you to choose a particular body directly from the geometry drawing. Choose the whole *T-junction* geometry and confirm by clicking *Apply* in the *Details of "Sizing"* – *Sizing* window. Then, in the same dialog box set the *Definition Type* as *Element Size* and enter the *Element Size* value as 0.02 m. Regenerate the mesh and examine its *Quality* and *Statistics*. In the last step create four *Named Selections* – *Inlet\_1*, *Inlet\_2*, *Outlet* and *Wall*. The subsequent steps of mesh generation are presented in Fig. E5-8.

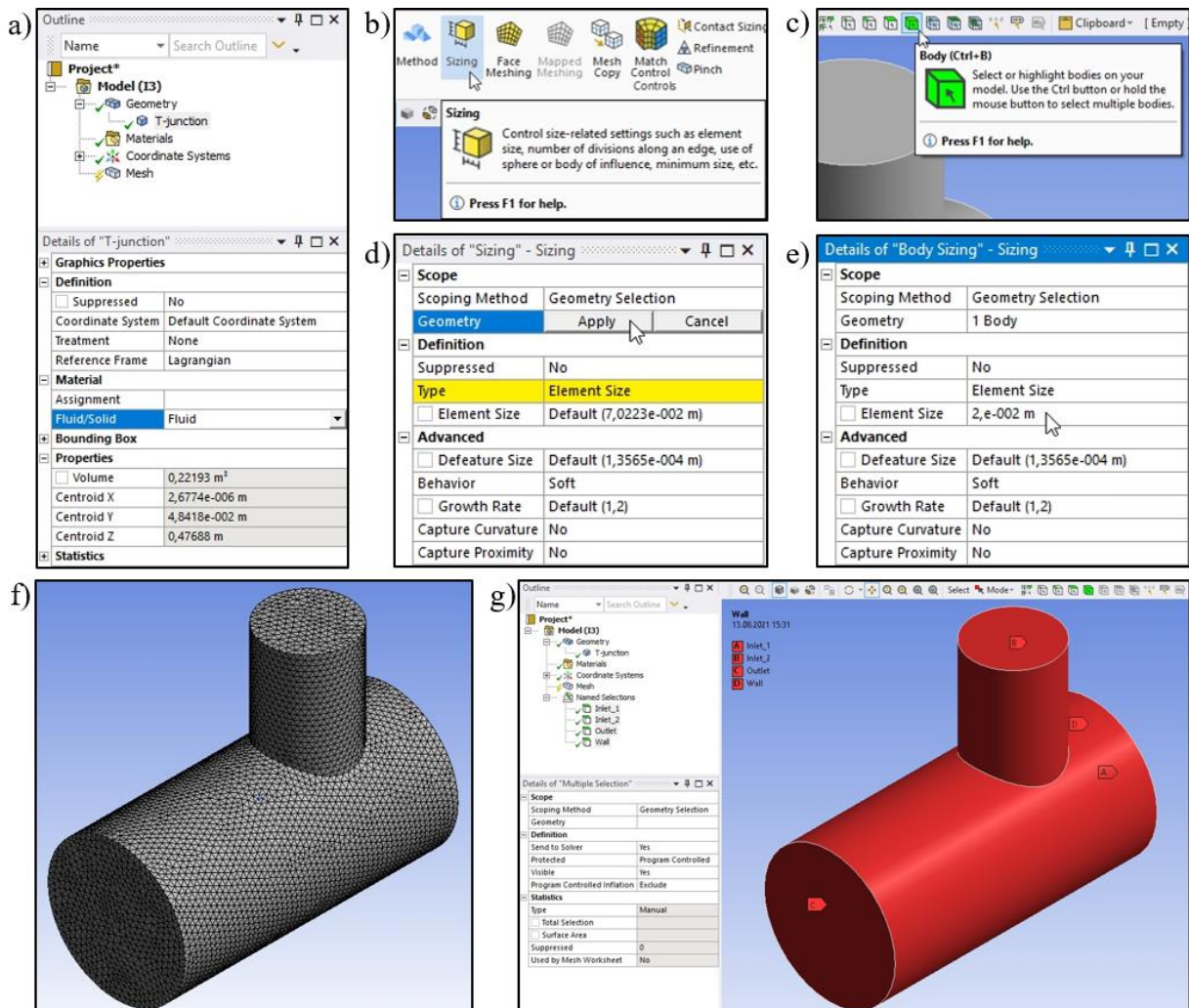


Figure E5-8. Steps of the mesh generation.

## 5.5. Simulations

Open the *Ansys Fluent* with enabled *Double Precision* and *Display Mesh After Reading* options and set the *Solver Processes* as 1. Run down the *Outline View Tree* to edit subsequent parameters and variables:

### A. Setup – General

Examine the following settings: *Mesh* → *Check*, *Mesh* → *Report Quality*, *Solver Type* → *Pressure-Based*, *Solver Velocity Formulation* → *Absolute*, *Time* → *Steady*.

### B. Setup – Models

In this example we will consider turbulent heat transfer in a T-junction. Therefore, from the *Models Task Page* select the *Viscous* → *k-epsilon (2 eqn)* model. Use the default *Standard* from the *k-epsilon Model* group and choose *Enhanced Wall Treatment* from the *Near-Wall Treatment*. Do not change the values of *Model Constants*. Click *OK* to accept the model settings and close the *Viscous Model* dialog box.

In order to simulate the heat transfer, the basic set of equations has to be supplemented with an energy equation. Hence, from the *Models* list select also the *Energy* equation. When the energy equation is chosen, its status changed from *Energy – Off* to *Energy – On* (Fig. E5-9).

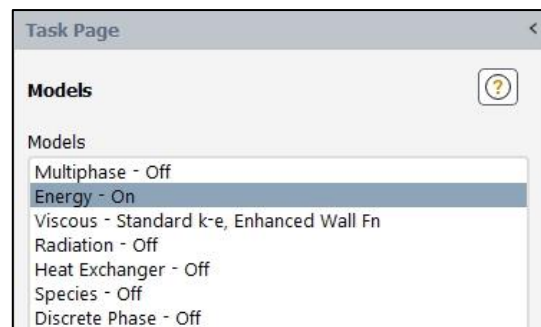


Figure E5-9. Models Task Page with enabled *Energy* equation.

### C. Setup – Materials

Open the *Materials Task Page* and add *water-liquid (h2o<l>)* to the to the *Materials* lists.

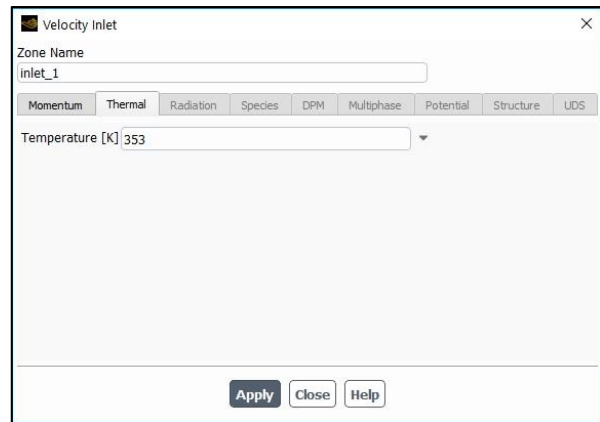
### D. Setup – Cell Zone Conditions

Open the *Cell Zone Conditions Task Page* and select the *water-liquid* as a process fluid that flows in the *t-junction* zone.

### E. Setup – Boundary Conditions

Open the *Boundary Conditions Task Page* and set the boundary conditions as *velocity-inlet* for *inlet\_1* and *inlet\_2* zones, the *pressure-outlet* for the *outlet* zone and the *wall* for the *wall* zone. In the case of both *velocity-inlet* boundary conditions you also need to specify values of

parameters that are characteristic for considered flow streams, such as the velocity and temperature. From the *Problem Description* and using definition of the Reynolds number we know the velocities of the water streams, that are equal to **0.1005** m/s and **0.1675** m/s for *Inlet\_1* and *Inlet\_2*, respectively. In order to enter values of the temperature change the tab in the *Velocity Inlet* dialog box from the *Momentum* for *Thermal* (Fig. E5-10). In the previous examples this tab was not active since the *Energy* equation was not considered. Here, you can enter the temperature of **353** K for the *Inlet\_1* and **283** K for the *Inlet\_2*, respectively.



**Figure E5-10.** *Velocity Inlet* dialog box with opened *Thermal* tab for *Inlet\_1*.

#### F. *Solution – Methods*

Open the *Solution Methods Task Page* and ensure that the options within this page are set up as: *Scheme* → *Coupled*, *Flux Type* → *Auto Select*, *Gradient* → *Least Squares Cell Based*, *Pressure* → *Second Order*, *Momentum*, *Turbulent Kinetic Energy*, *k*, and *Turbulent Dissipation Rate*,  $\epsilon$ , → *Second Order Upwind*, and other options are disabled. Here also the *Energy* is calculated, for which select the *Second Order Upwind* scheme of discretization.

#### G. *Solution – Monitors*

Open the *Residual Monitors* dialog box, make sure that all equations are monitored and their convergence is checked. Then set all *Absolute Criteria* as  $1e-7$ .

#### H. *Solution – Initialization*

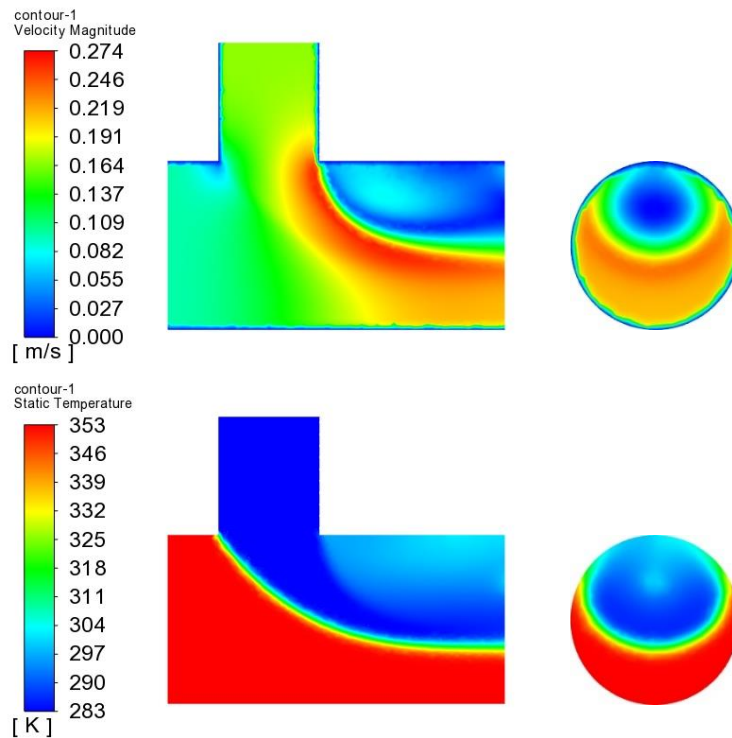
Open the *Solution Initialization Task Page* and initialize the calculation selecting the *Standard Initialization* with enabled *Compute from all-zones* option.

#### I. *Solution – Run Calculation*

Open the *Run Calculation Task Page* and examine your setup selecting *Check Case...* You should see the recommendation to consider *realizable k-epsilon turbulence model*, nevertheless do not apply this model and close the dialog box. You can start the calculation by entering 1000 iterations. During the calculations in the main window you will observe the *Scaled Residuals*.

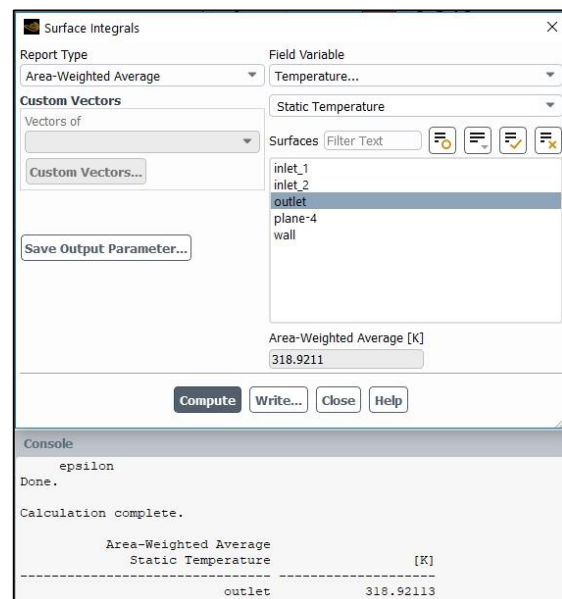
## 5.6. Results

Contours of the fluid velocity magnitude and temperature will be presented at the outflow and in the mid-plane of the T-junction. The outflow surface already exists, but there is no mid-plane. Therefore, before reading the results, it is necessary to create a plane that passes through the centre of the flow domain. The steps of a surface creating were described in *Example 1* and they will not be repeated here. The obtained results are presented in Fig. E5-11.



**Figure E5-11.** Contours of fluid velocity magnitude and temperature in the mid-plane and at the outflow.

Additionally, we will read the average value of temperature at the outflow. In this purpose from the *Outline View* tree select *Results* → *Reports* → *Surface Integrals*. A new dialog box will open. From the *Report Type* dropdown list select the *Area-Weighted Average*. The *Field Variable* set as *Temperature...* and *Static Temperature*. In the last step from the *Surfaces* list select the *outlet* and click *Compute*. Value of the average temperature, here about 318.92 K, will appear both, in dialog box and in the *Console* (Fig. E5-12).



**Figure E5-12.** Average value of temperature at the outflow.