meta4.0

Erasmus Mundus Joint Master in Manufacturing 4.0 by intElligent and susTAinable technologies

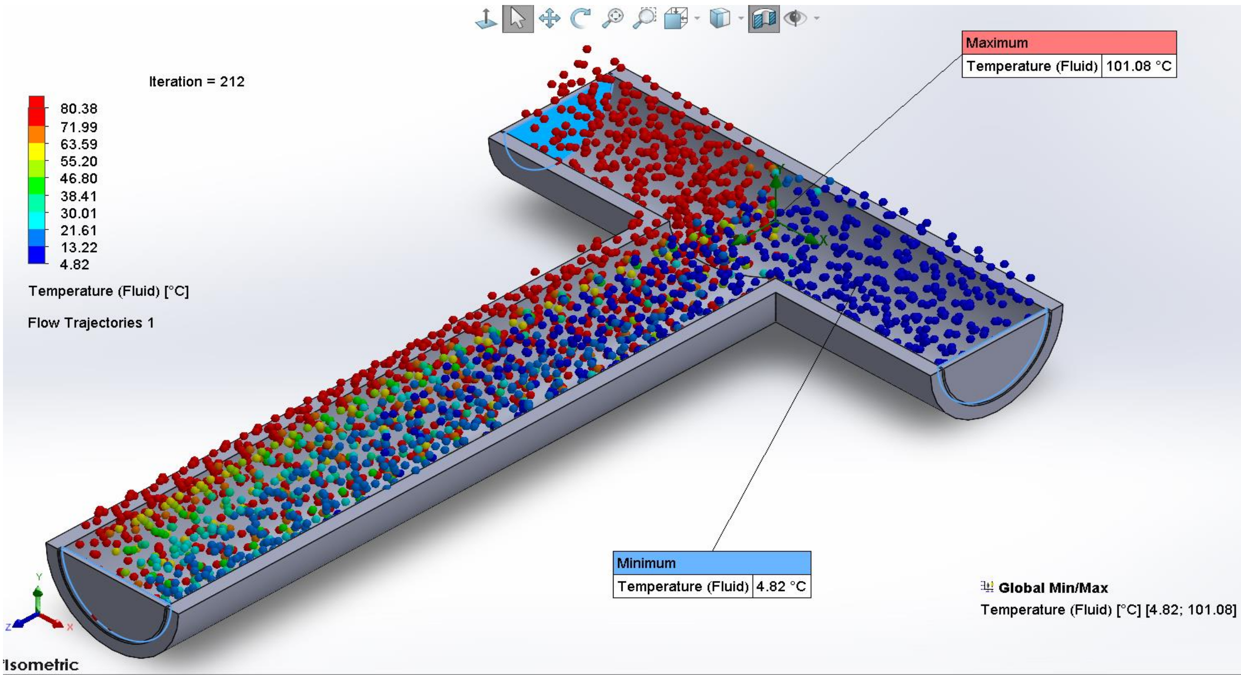
Co-funded by the European Union

SOLIDWORKS

Fluid Flow Simulation
Project Report

Computational Fluid Dynamics (CFD) Analysis of Hot–Cold Water Mixing in a T-Junction Duct Using SolidWorks Flow Simulation

Flow Simulation Project Report



[Learn more about SOLIDWORKS Flow Simulation]

Table of Contents

1 General Information

Objective of the simulation: Efficient fluid flow through ducting systems is a critical requirement in many engineering applications, including HVAC systems, thermal management of equipment, chemical processing, and fluid transport networks. Computational Fluid Dynamics (CFD) tools enable engineers to predict flow behavior, pressure distribution, and thermal characteristics before physical prototyping. This project presents a numerical study of internal duct flow in a T-junction mixing configuration, where two inlet ducts supply hot and cold water that mix and exit through a single outlet, conducted using SolidWorks Flow Simulation. The study focuses on evaluating velocity, pressure, temperature, density, and related flow parameters, supported by quantitative data tables and graphical reports.

1.1 Analysis Environment

Software Product:	Flow Simulation 2025 SP1.0. Build: 6588
CPU Type:	Intel(R) Celeron(R) J4105 CPU @ 1.50GHz
CPU Speed:	1501 MHz
RAM:	12128 MB / 2073 MB
Operating System:	Windows 11 (or higher) (Version 10.0.22631)

1.2 Model Information

Model Name:	Pipe CFD Results.SLDPRT
Project Name:	Pipe CFD Project

1.3 Project Comments:

Unit System:	SI (m-kg-s)
Analysis Type:	External (not exclude internal spaces)

1.4 Size of Computational Domain

Size

X min	-3.880 m
X max	3.880 m
Y min	-3.530 m
Y max	3.530 m
Z min	-3.530 m
Z max	4.880 m
X size	7.760 m
Y size	7.060 m
Z size	8.410 m

1.5 Simulation Parameters

1.5.1 Mesh Settings

1.5.1.1 Basic Mesh

Basic Mesh Dimensions

Number of cells in X	82
Number of cells in Y	70
Number of cells in Z	90

1.5.1.2 Analysis Mesh

Total Cell count:	675816
Fluid Cells:	675816
Solid Cells:	99020
Partial Cells:	64924
Trimmed Cells:	0

1.5.1.3 Additional Physical Calculation Options

Heat Transfer Analysis:	Fluid Flow: OnConduction: Off
Flow Type:	Laminar and turbulent
Time-Dependent Analysis:	Off
Gravity:	Off
Radiation:	
Humidity:	
Default Wall Roughness:	0 micrometer

1.5.2 Material Settings

Material Settings

Fluids

Water

1.5.3 Initial Conditions

Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 20.05 °C
Velocity parameters	Velocity vector Velocity in X direction: 0 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m

1.5.4 Boundary Conditions

Boundary Conditions

Inlet Mass Flow 1

Type	Inlet Mass Flow
Faces	Face<1>
Coordinate system	Face Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate: 2.0000 kg/s Fully developed flow: No Inlet profile: 0
Thermodynamic parameters	Temperature type: Temperature of initial components Temperature: 100.00 °C
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m
Boundary layer parameters	Boundary layer type: Turbulent

Inlet Mass Flow 2

Type	Inlet Mass Flow
Faces	Face<2>
Coordinate system	Face Coordinate System
Reference axis	X
Flow parameters	Flow vectors direction: Normal to face Mass flow rate: 2.0000 kg/s Fully developed flow: No Inlet profile: 0
Thermodynamic parameters	Temperature type: Temperature of initial components Temperature: 5.00 °C
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m
Boundary layer parameters	Boundary layer type: Turbulent

Environment Pressure 3

Type	Environment Pressure
Faces	Face<1>
Coordinate system	Face Coordinate System
Reference axis	X
Thermodynamic parameters	Environment pressure: 101325.00 Pa Temperature type: Temperature of initial components Temperature: 20.05 °C
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.003 m
Boundary layer parameters	Boundary layer type: Turbulent

1.5.5 Volumetric Heat Sources

1.5.6 Engineering Goals

1.6 Analysis Time

Calculation Time:	3743 s
Number of Iterations:	212
Warnings:	

2 Results

2.1 Analysis Goals

Goals

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence

2.2 Global Min-Max-Table

Min/Max Table

Name	Minimum	Maximum
Density (Fluid) [kg/m^3]	958.22	1000.63
Pressure [Pa]	101317.01	101331.59
Temperature [°C]	4.82	101.08
Temperature (Fluid) [°C]	4.82	101.08
Velocity [m/s]	0	0.139
Velocity (X) [m/s]	-0.121	0.125
Velocity (Y) [m/s]	-0.066	0.069
Velocity (Z) [m/s]	-0.032	0.135
Velocity RRF [m/s]	0	0.139
Velocity RRF (X) [m/s]	-0.121	0.125
Velocity RRF (Y) [m/s]	-0.066	0.069
Velocity RRF (Z) [m/s]	-0.032	0.135
Vorticity [1/s]	8.89e-27	14.97
Relative Pressure [Pa]	-7.99	6.59
Shear Stress [Pa]	0	0.38
Bottleneck Number [ ]	2.2300139e-37	1.0000000
Heat Transfer Coefficient [W/m^2/K]	0	0
ShortCut Number [ ]	0	1.0000000
Surface Heat Flux [W/m^2]	0	0
Surface Heat Flux (Convective) [W/m^2]	0	0
Total Enthalpy Flux [W/m^2]	-1.553e+08	6.952e+07
Acoustic Power [W/m^3]	0	1.734e-24
Acoustic Power Level [dB]	0	0

2.3 Results

The results indicate a low-velocity internal mixing flow regime, with a maximum velocity of 0.139 m/s. Pressure variations across the system are relatively small, indicating stable flow during the mixing of hot and cold water. Temperature variation from approximately 4.8 °C to 101 °C confirms effective thermal mixing. Density changes correspond directly to temperature differences between the two inlet streams. Increased vorticity near the T-junction indicates localized mixing and flow interaction, which is expected in hot–cold water mixing systems.

2.4 Conclusion

This study successfully demonstrated the use of SolidWorks Flow Simulation to analyze hot–cold water mixing in a T-junction duct with two inlets and one outlet. The results indicate that the duct system exhibits stable and efficient flow behavior, with minimal pressure losses that reflect good overall design performance. The simulation also showed that temperature has a measurable influence on fluid density, affecting the mixing characteristics within the duct. Furthermore, the CFD analysis provided valuable insight into complex flow patterns that are difficult to observe through experimental methods alone. Overall, the findings validate the effectiveness of CFD tools in accurately predicting real-world duct flow behavior.

3 Appendix

3.1 Material Data

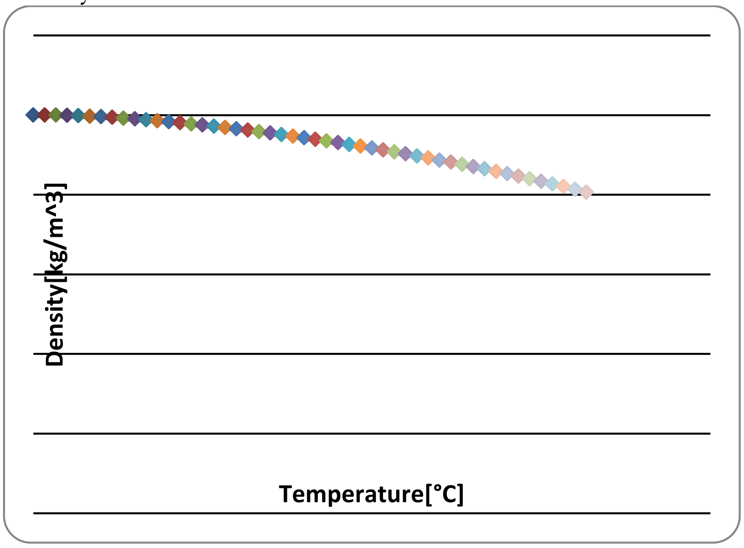
Engineering Database

Liquids

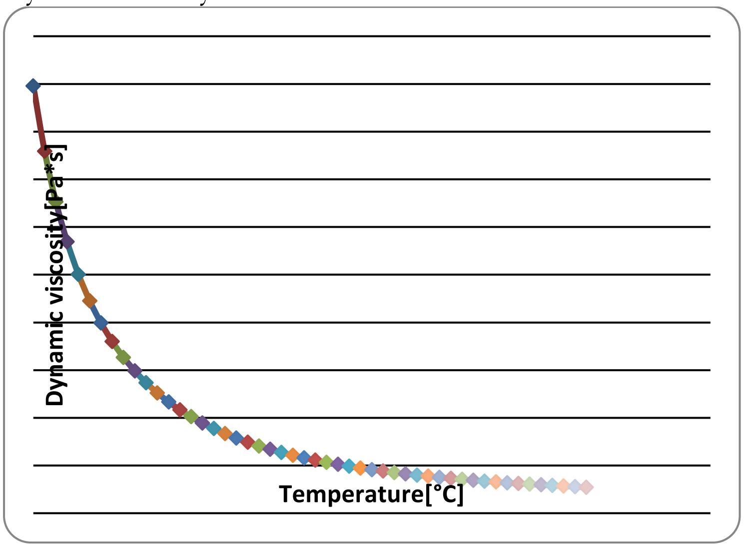
Water

Path: Liquids Pre-Defined

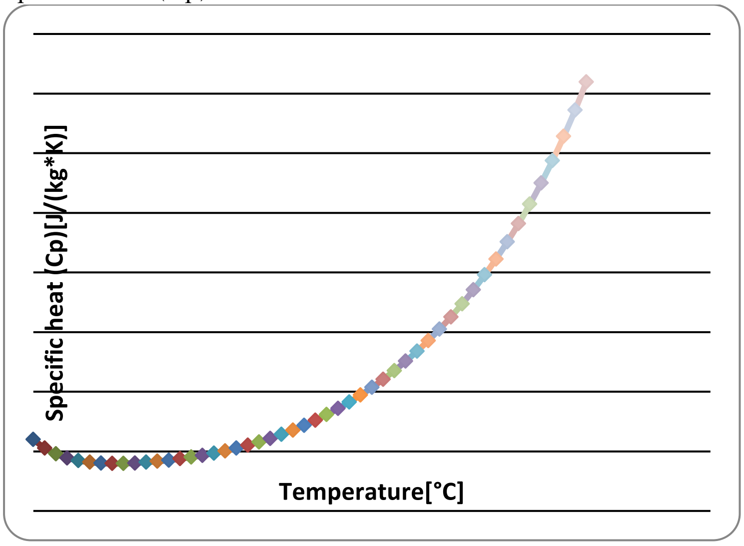
Density



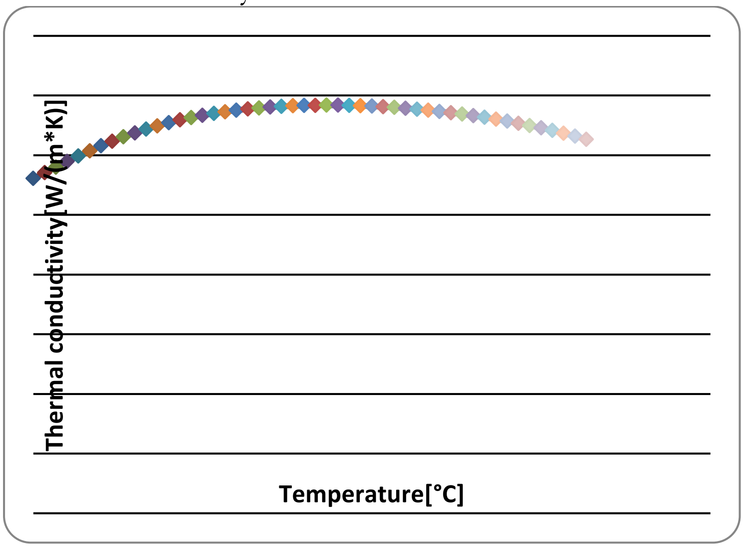
Dynamic viscosity



Specific heat (Cp)

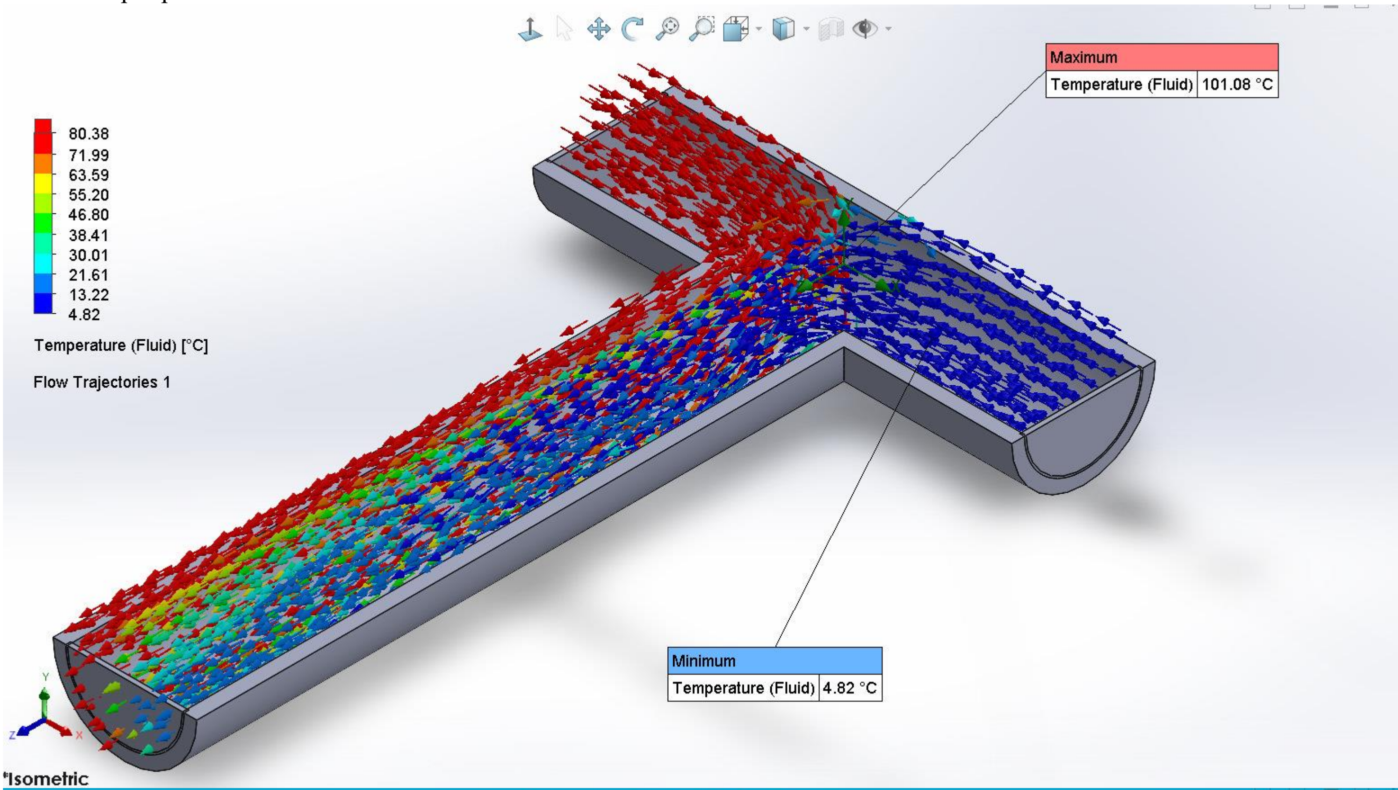


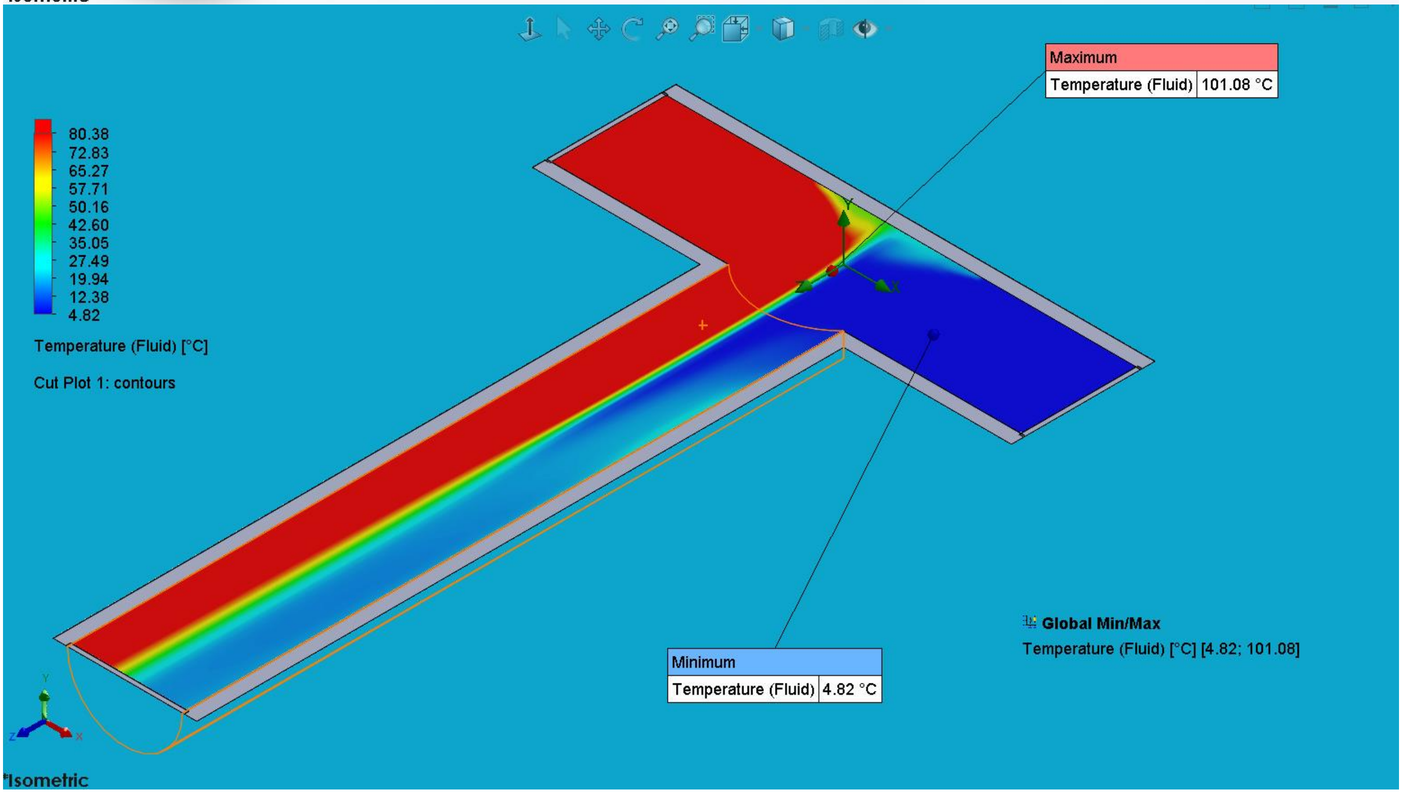
Thermal conductivity

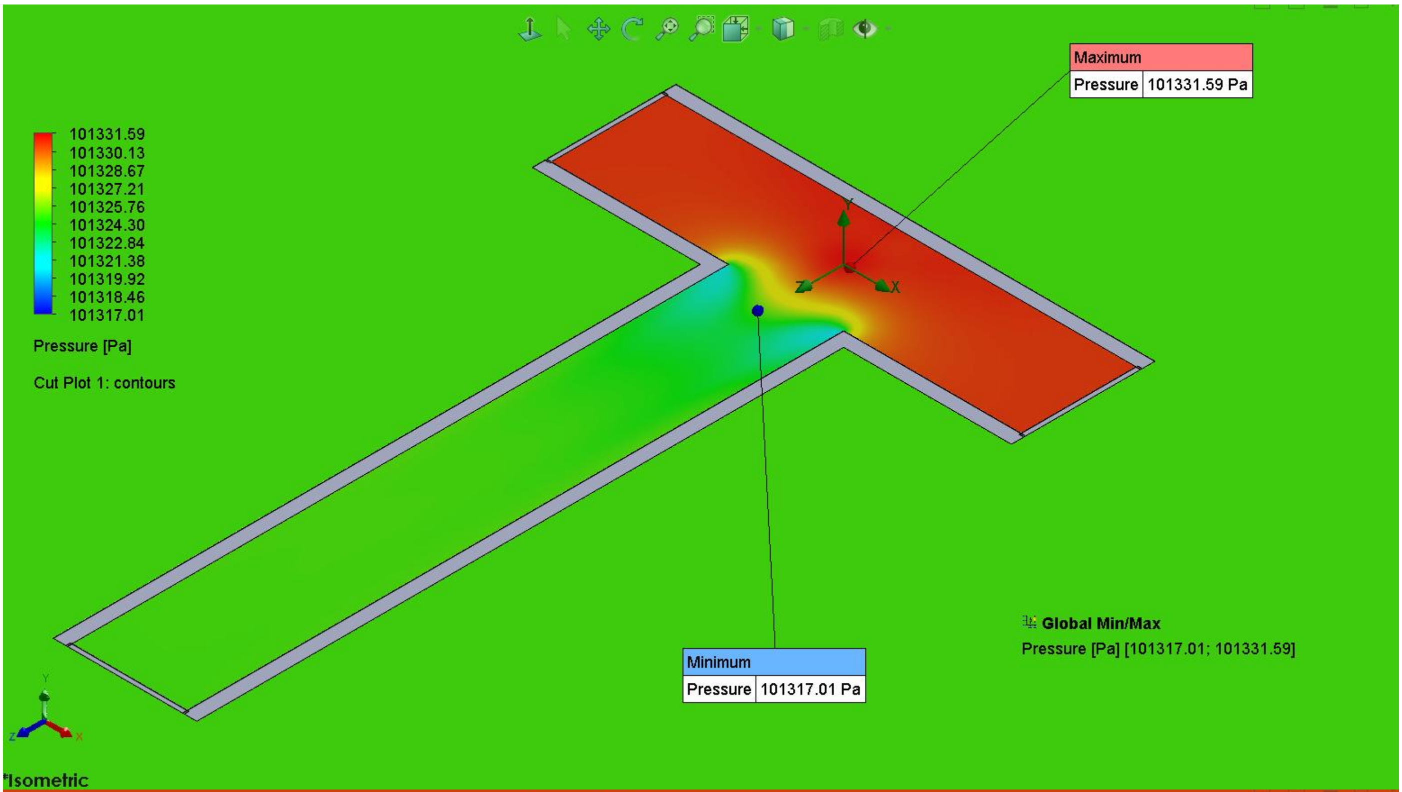


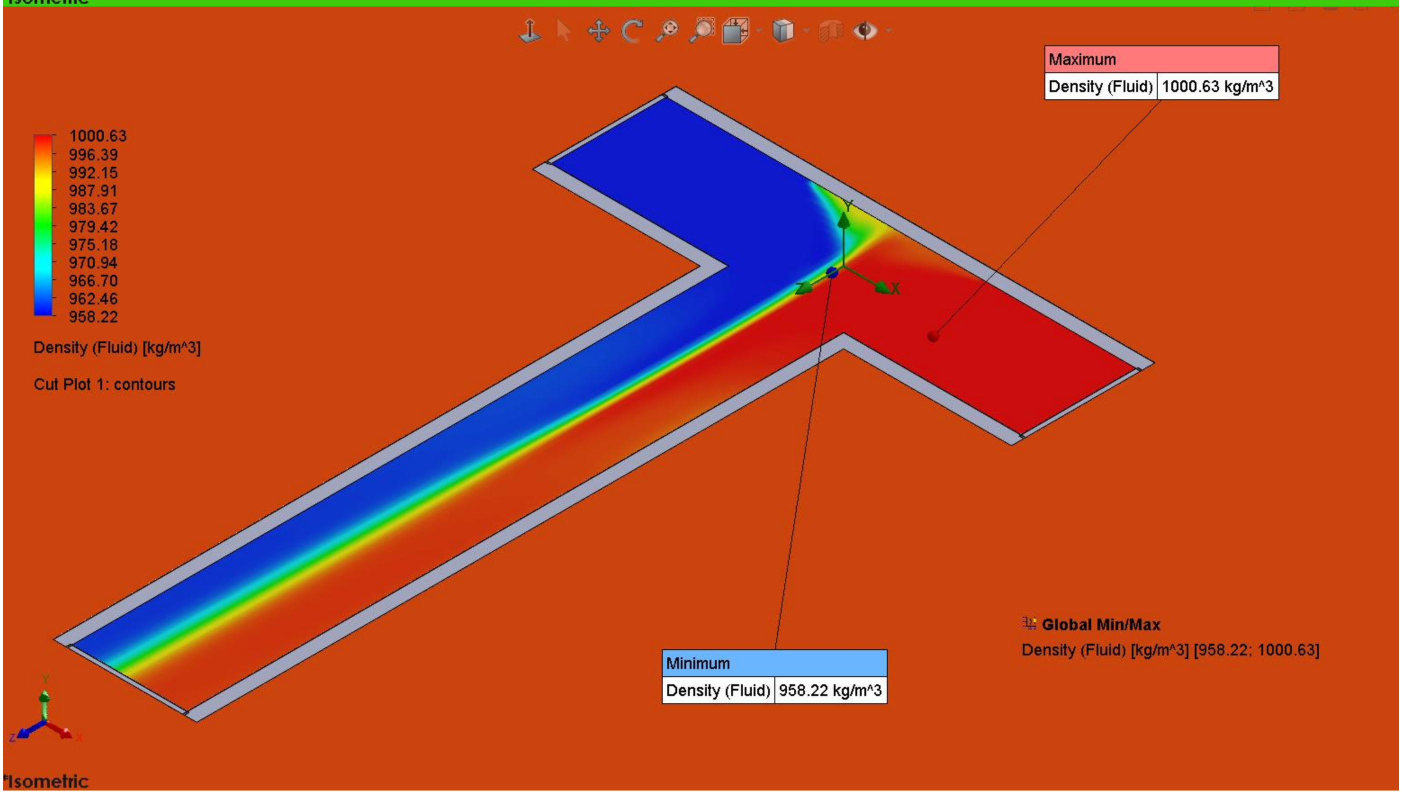
Cavitation effect: Yes
Temperature: -273.15 °C
Saturation pressure: 0 Pa

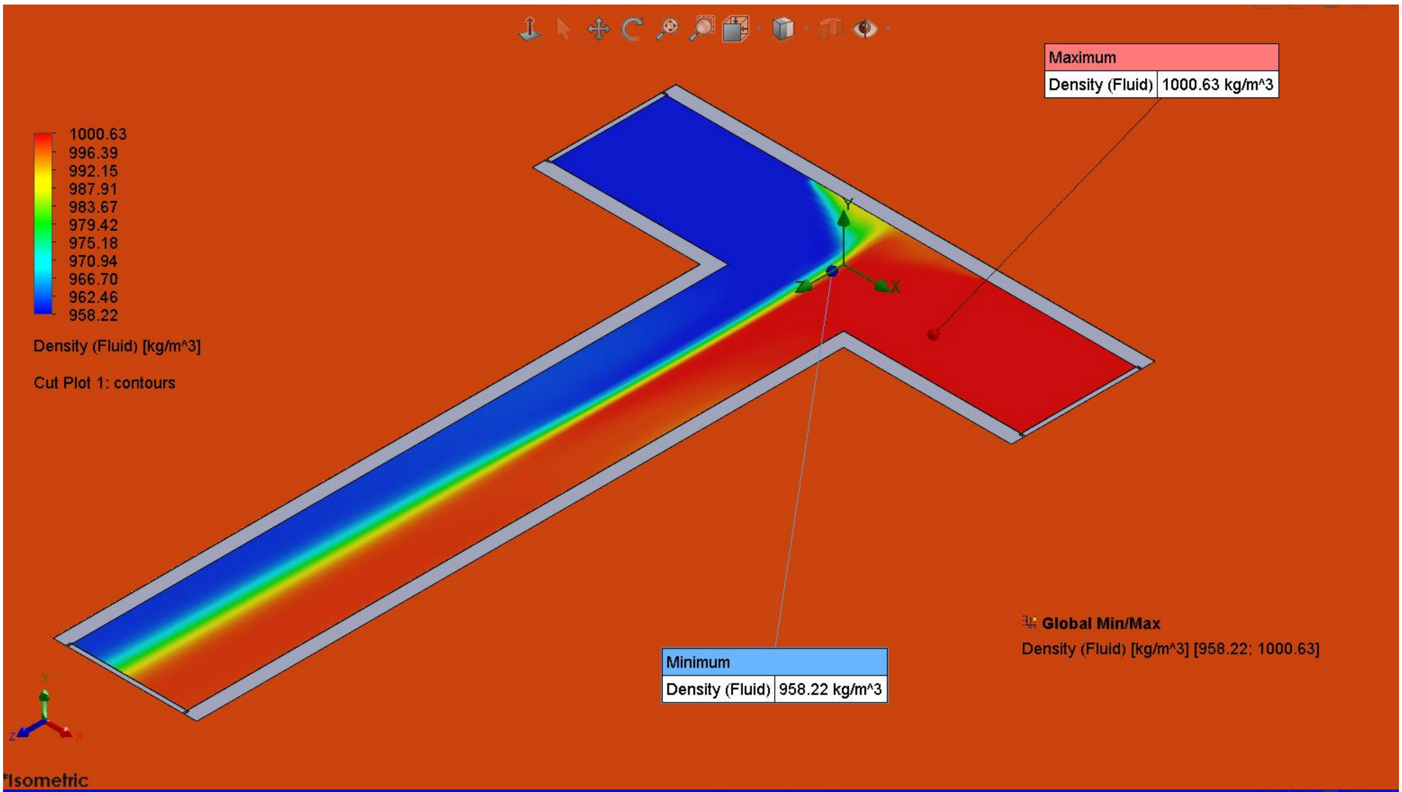
Radiation properties: No

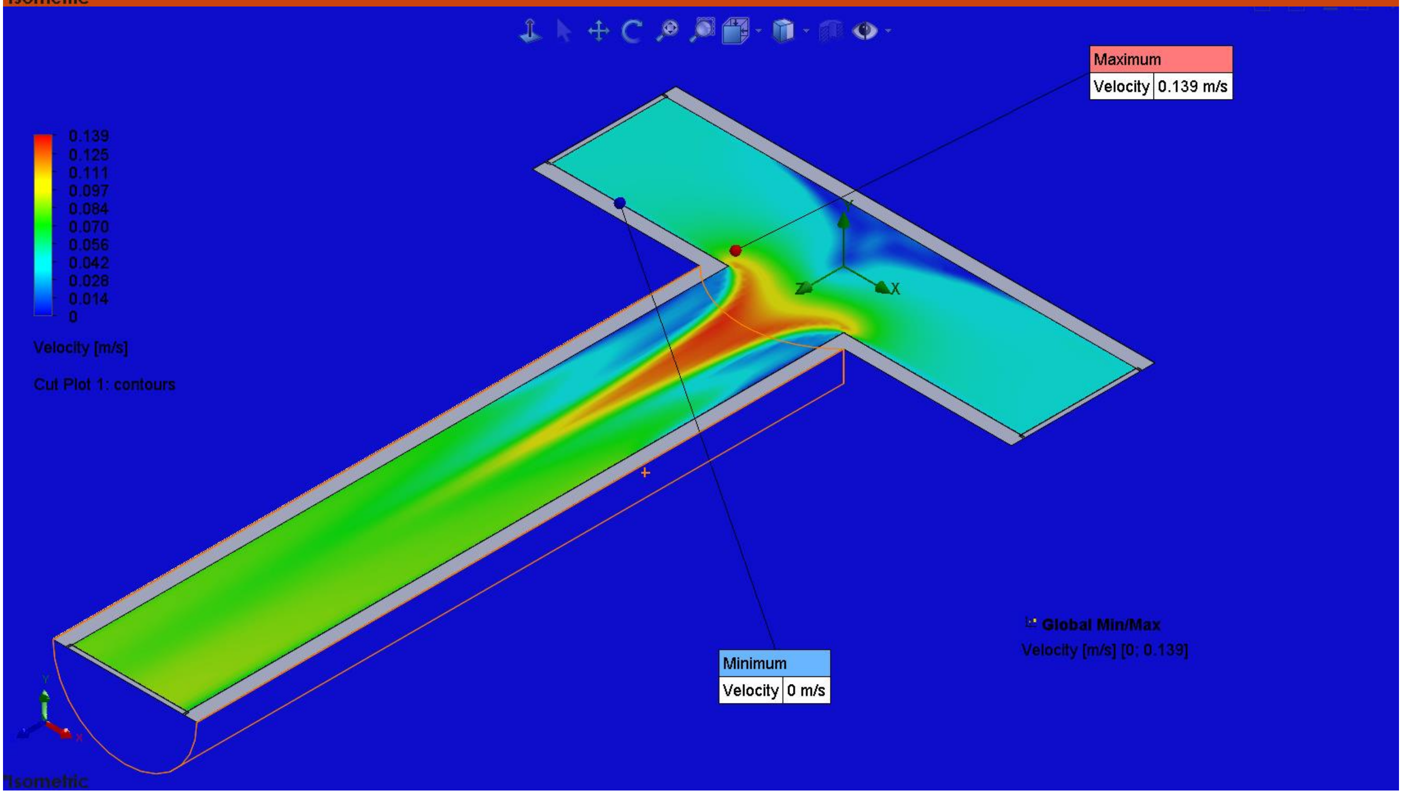





Maximum
Pressure 101331.59 Pa
101331.59
101330.13
101328.67
101327.21
101325.76
101324.30
101322.84
101321.38
101319.92
101318.46
101317.01
Pressure [Pa]
Cut Plot 1: contours
Global Min/Max
Pressure [Pa] [101317.01; 101331.59]
Minimum
Pressure 101317.01 Pa
*Isometric


Maximum
Density (Fluid) 1000.63 kg/m^3
1000.63
996.39
992.15
987.91
983.67
979.42
975.18
970.94
966.70
962.46
958.22
Density (Fluid) [kg/m^3]
Cut Plot 1: contours
Global Min/Max
Density (Fluid) [kg/m^3] [958.22; 1000.63]
Minimum
Density (Fluid) 958.22 kg/m^3
*Isometric


Maximum
Density (Fluid) 1000.63 kg/m^3
1000.63
996.39
992.15
987.91
983.67
979.42
975.18
970.94
966.70
962.46
958.22
Density (Fluid) [kg/m^3]
Cut Plot 1: contours
Global Min/Max
Density (Fluid) [kg/m^3] [958.22; 1000.63]
Minimum
Density (Fluid) 958.22 kg/m^3
*Isometric


Maximum
Velocity 0.139 m/s
0.139
0.125
0.111
0.097
0.084
0.070
0.056
0.042
0.028
0.014
0
Velocity [m/s]
Cut Plot 1: contours
Global Min/Max
Velocity [m/s] [0; 0.139]
Minimum
Velocity 0 m/s
*Isometric