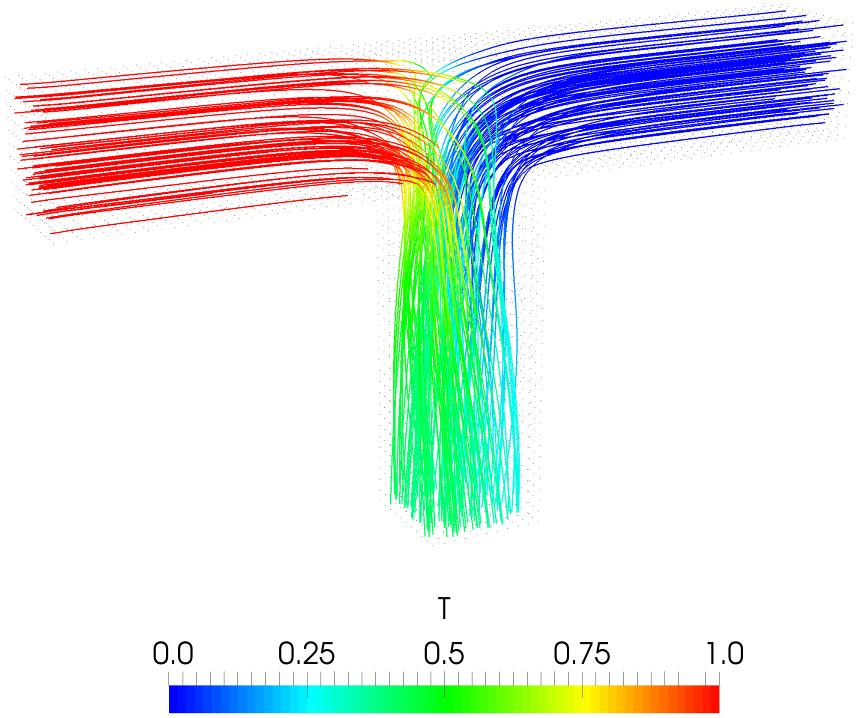
Tutorial Ten

Residence Time Distribution



5th edition, Sep. 2019

Compatibility:

- OpenFOAM® 7
- OpenFOAM® v1906

Cover picture from:

- Bahram Haddadi

ISBN 978-3-903337-00-8

Publisher: chemical-engineering.at

For more tutorials visit: www.cfd.at

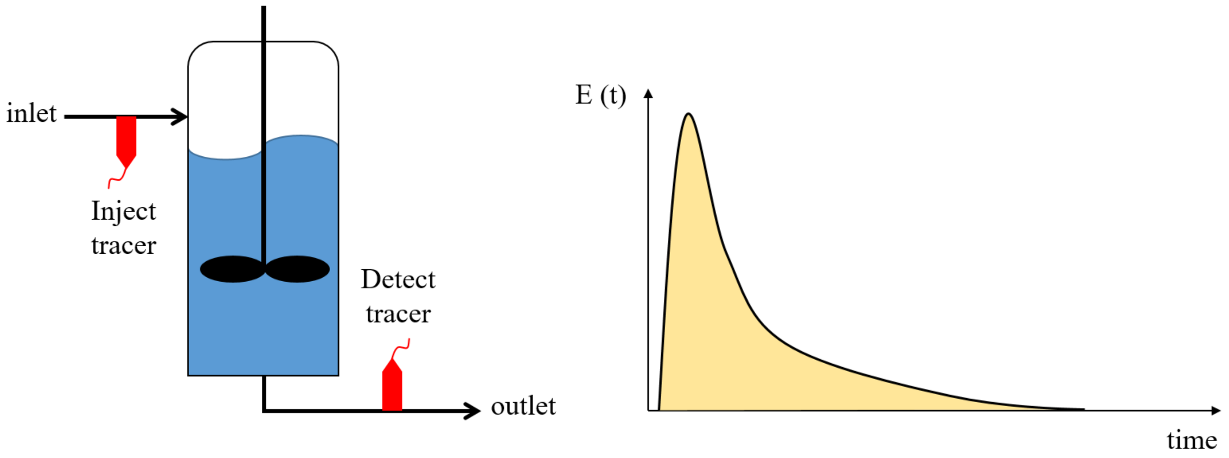
Background

In this tutorial we will carry out Residence Time Distribution (RTD) analysis of fluid flow through a T-junction pipe.

1. Residence Time Distribution (RTD)

Residence time distribution is a probability distribution function that provides information about the amount of time a tracer element spends within a process unit, such as a reactor or a column. RTD analysis is important because in almost all real-life processes, the mixing is not ideal and chemical engineers will need RTD to analyze the real mixing characteristics, for example inside a continuously stirred reactor. They can also use RTD analysis to obtain information about the flow pattern, back mixing and bypassing behavior of a process unit.

2. Tracer Analysis



Tracer analysis and RTD distribution of an ideal process

Radioactive tracers are usually used to determine RTD of a process unit. Based on the above diagram, first the tracer is injected into the inlet, and then the exit tracer concentration, $C(t)$, is measured at regular time intervals. This allows the exit age distribution, $E(t)$, to be calculated.

$$E(t) = \frac{C_T(t)}{\int_0^\infty C_T(t)\,dt} = \frac{Tracer\ concentration\ at\ time\ t}{Total\ tracer\ concentration}$$

It is clear from the above equation that the fraction of tracer molecules exiting the reactor that have spent a time between t and $t + dt$ in the process unit is $E(t)dt$. Since all tracer elements will leave the unit at some point, RTD satisfies the following relationship:

$$\int_0^\infty E(t)\,dt = 1$$

simpleFoam & scalarTransportFoam – TJunction

Simulation

Use the simpleFoam and scalarTransportFoam to simulate the flow through a square cross section T pipe and calculate RTD (Residence Time Distribution) for both inlets using a step function injection:

- Inlet and outlet cross sections: 1×1 m^2
- Gas in the system: air at ambient conditions
- Operating pressure: 10^5 Pa
- Inlet 1: 0.1 m/s
- Inlet 2: 0.2 m/s

Objectives

- Understanding RTD calculation using OpenFOAM®
- Using multiple solvers for a simulation

Data processing

Plot the step response function and the RTD curve.

1. Pre-processing

1.1. Copy tutorial

Copy the following tutorial to your working directory as a base case:

```
$FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily
```

1.2. 0 directory

Update p, U, nut, nuTilda, k and epsilon files with the new boundary conditions, e.g. U:

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    inlet_one
    {
        type            fixedValue;
        value           uniform (0.1 0 0)
    }
    inlet_two
    {
        type            fixedValue;
        value           uniform (-0.2 0 0)
    }
    outlet
    {
        type            zeroGradient;
    }
    walls
    {
        type            fixedValue;
        value           uniform (0 0 0)

    }
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//
```

1.3. constant directory

Check turbulenceProperties file for the turbulence model (`kEpsilon`).

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//
simulationType  RAS
RAS
{
       RASModel        kEpsilon;

       turbulence      on;

       printCoeffs     on;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//
```

1.4. system directory

Edit the blockMeshDict to create an appropriate geometry.

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//
convertToMeters 1.0;

vertices
(
```

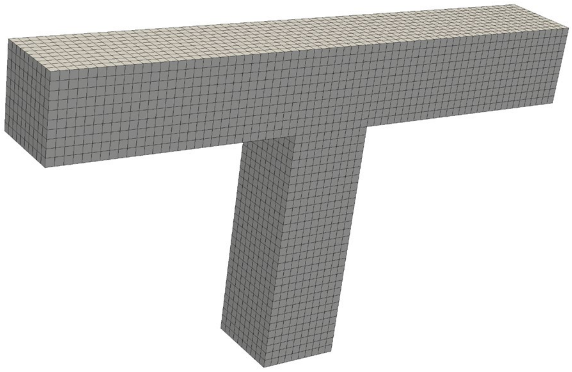
```
    (0 4 0) // 0
    (0 3 0) // 1
    (3 3 0) // 2
    (3 0 0) // 3
    (4 0 0) // 4
    (4 3 0) // 5
    (7 3 0) // 6
    (7 4 0) // 7
    (4 4 0) // 8
    (3 4 0) // 9
    (0 4 1) // 10
    (0 3 1) // 11
    (3 3 1) // 12
    (3 0 1) // 13
    (4 0 1) // 14
    (4 3 1) // 15
    (7 3 1) // 16
    (7 4 1) // 17
    (4 4 1) // 18
    (3 4 1) // 19

);
blocks
(
    hex (0 1 2 9 10 11 12 19) (10 30 10) simpleGrading (1 1 1)
    hex (9 2 5 8 19 12 15 18) (10 10 10) simpleGrading (1 1 1)
    hex (8 5 6 7 18 15 16 17) (10 30 10) simpleGrading (1 1 1)
    hex (2 3 4 5 12 13 14 15) (30 10 10) simpleGrading (1 1 1)
);
edges
(
);
patches
(
    patch inlet_one
    (
        (0 10 11 1)
    )
    patch inlet_two
    (
        (7 6 16 17)
    )
    patch outlet
    (
        (4 3 13 14)
    )
    wall walls
    (
        (0 1 2 9)
        (2 5 8 9)
        (5 6 7 8)
        (2 3 4 5)
        (10 19 12 11)
        (19 18 15 12)
        (18 17 16 15)
        (15 14 13 12)
        (0 9 19 10)
        (9 8 18 19)
        (8 7 17 18)
        (2 1 11 12)
        (3 2 12 13)
        (5 4 14 15)
        (6 5 15 16)
    )
);

mergePatchPairs
(
);
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *//
```

2. Running simulation

```
>blockMesh
```



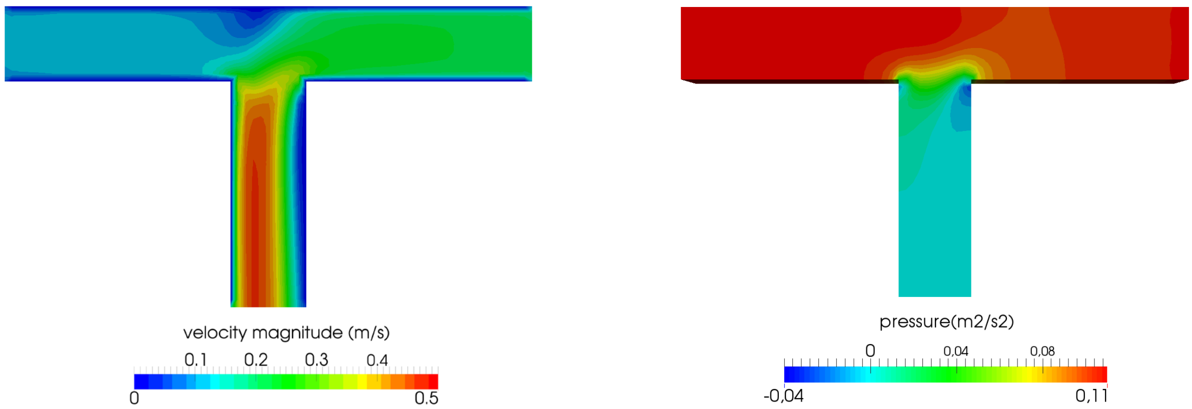
Mesh created using blockMesh

```
>simpleFoam
```

Wait for simulation to converge. After convergence, check the results to make sure about physical convergence of the solution.

```
>foamToVTK
```

The simulation results are as follows (results are on the cut plane in the middle):



Simulation results after convergence (~65 iterations)

3. RTD calculation

3.1. Copy tutorial

Copy following tutorial to your working directory:

```
$FOAM_TUTORIALS/basic/scalarTransportFoam/pitzDaily
```

3.2. 0 directory

Delete the U file and replace it with the calculated velocity field from the first part of the tutorial (use the latest time step velocity field from previous part of simulation to calculate RTD for this geometry). There is no need to modify or change it. The solver will use this field to calculate the scalar transportation.

Update T (T will be used as an inert scalar in this simulation) file boundary conditions to match new simulation boundaries, to calculate RTD of the `inlet_one` set the `internalField` value to `0`, T value for `inlet_one` to `1.0` and T value for `inlet_two` to `0`.

3.3. system directory

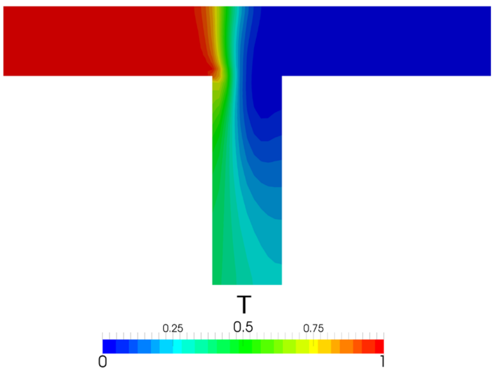
Replace the blockMeshDict file with the one from the first part of tutorial.

In the controlDict file change the `endTime` from `0.1` to `120` (approximately two times ideal resistance time) and also `deltaT` from `0.0001` to `0.1` (Courant number approximately 0.4).

4. Running Simulation

```
>blockMesh
>scalarTransportFoam
>foamToVTK
```

5. Post-processing



Contour plots scalar T at 120 s for inlet 1

5.1. Calculating RTD

To calculate RTD the average T value at the outlets should be calculated first. The “integrate variables function” of ParaView can be used for this purpose.

```
>foamToVTK
```

Load the outlet VTK file into paraview using following path:

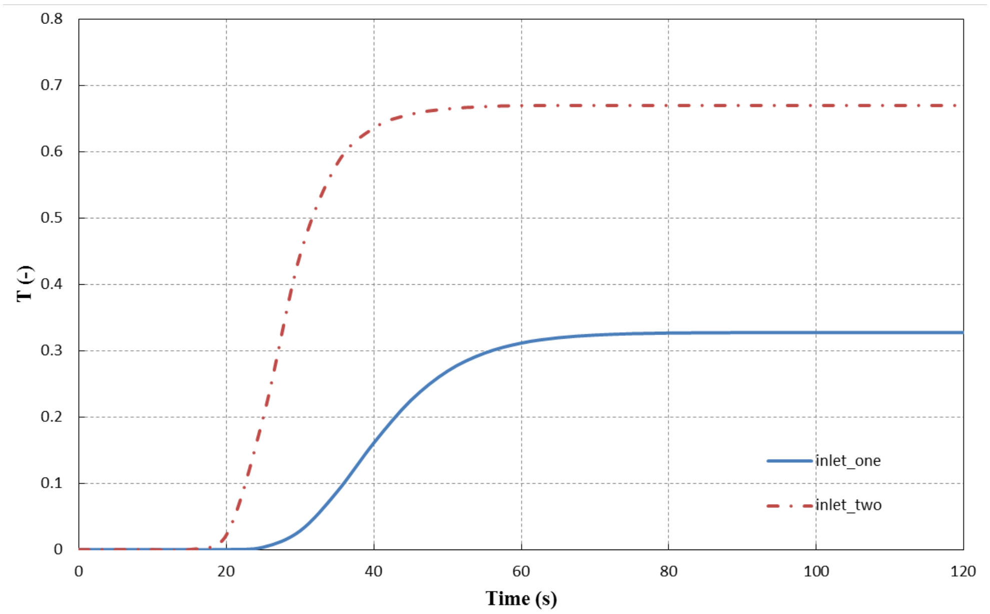
File > Open > VTK > outlet > outlet_..vtk > OK > Apply

Select T from variables menu, and then integrate the variables on the outlet:

Filters > Data Analysis > Integrate Variables > Apply

The values given in the opened window are integrated values in this specific time step. By changing the time step values for different time steps are displayed. As mentioned before, the average value of the property is needed. Therefore, these values should be divided by outlet area to get average values (1m × 1m).

After finishing the RTD calculations for `inlet_one`, the same procedure should be followed for calculating RTD of `inlet_two`, except T value for `inlet_one` should be `0` and for `inlet_two` it should be `1.0`.

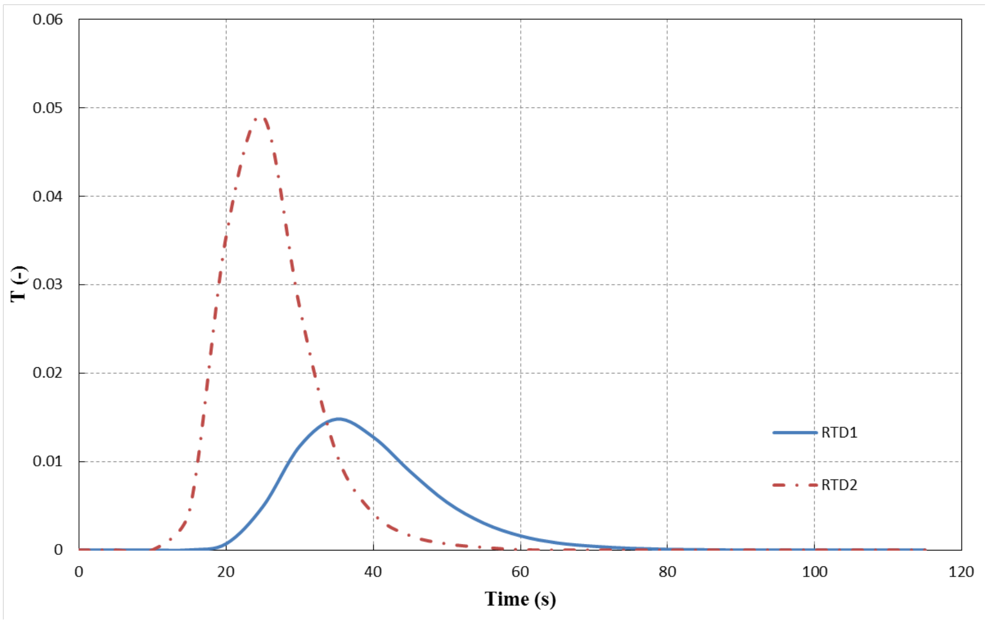


Average value of T on the outlet for two inlets versus time

The average value of T for each outlet approaches a certain constant value, which is the ratio of that scalar mass inlet to the whole mass inlet. For plotting data over time “Plot Selection Over Time” option in ParaView can be used, in the opened SpreadSheetView window (IntegrateVariables) select the set of data which you want to plot over time and then:

Filters > Data Analysis > Plot Selection Over Time > Apply

Next, to obtain the RTD plots, export the data to a spreadsheet program (e.g. Excel), calculate and plot the gradient of changes in average value of T on the outlet from time 0 to 120s for both inlets.



RTD of two inlets

ISBN 978-3-903337-00-8

9 783903 337008