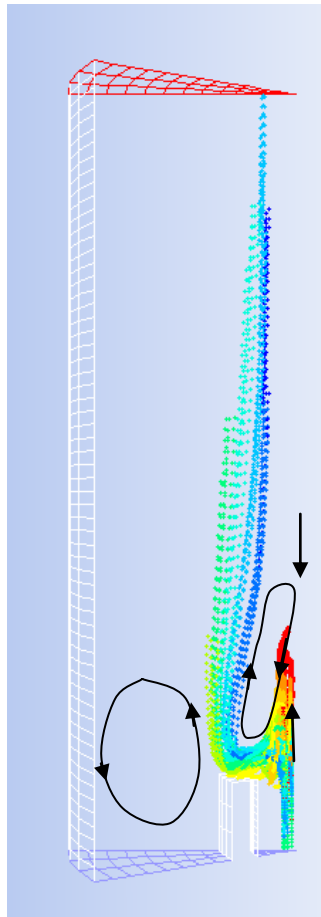

Computational Fluid Dynamics for Engineers

Tutorial 3

Swirl Stabilized Spray Combustor



This tutorial can be printed and distributed freely in its original and complete form conditioned that it is used as supplementary training material to the book *Computational Fluid Dynamics for Engineers*, Andersson B., Andersson R., Håkansson L., Mortensen M., Sudiyo R., van Wachem B., ISBN 978-1-107-01895-2, published 2011 by Cambridge University Press, The Edinburgh Building, Cambridge CB2 8RU, UK.

Additional resources such as *tutorials, project and lecture notes* are available from the authors and at www.cambridge.org/9781107018952

Cambridge University Press and the Authors have no responsibility for the persistence or accuracy of URLs and do not guarantee that any content is, or will remain, accurate or appropriate.

1. Tutorial introduction and objectives

The structure of these instructions are as follows.

In Section 1 the problem description is given along with all required information regarding geometry and boundary conditions. Hence, all information required to solve the problem is found here.

In Section 2 the questions related to the problem, model accuracy and limitations are found, along with instructions on how to prepare a short report.

Section 3-5, contains instructions specific to the Ansys software; how to generate the CAD model, create the mesh, run the simulations and do post-processing. All instructions are written for Ansys Workbench 13 and the software used are: Design Modeler, Meshing Platform, Fluent respectively.

In this tutorial a *swirl stabilized spray combustor* is studied. The purpose is that you, in accordance with best practice guideline, Chapter 7, learn how to:

- *Generate*: CAD and mesh files for the system, 3D model.
- *Analyze*: Mesh quality.
- *Define*: Periodic boundary, Turbulent flow, Lagrangian particle simulation, Rosin -Rammler size distribution.
- *Calculate*: Flow field, turbulence, particle trajectories and drop evaporation rate.
- *Judge*: Convergence.
- *Evaluate*: The effect of injecting drops of different size.
- *Analyze*: Drop size as function of location, drop temperature, residence time and particle velocity.
- *Discuss*: Validity of simulation results with respect to assumptions made in the problem formulation, limitation with the model, possible refinements.

1.1 Tutorial problem description

The system considered in this tutorial is shown in Figure 1. This swirl stabilized combustor has recirculation zones that allows entrainment of drops and hot combustion radicals (species) from the downstream region of the flame, which stabilizes the combustion process. Experiments with combustor operating conditions shows that by decreasing the swirl flow at a fixed dilution flow rate, the flame may lift up and be blown out of the combustor. Furthermore if the swirl is increased to much it may extinguish the flame as well.

There are three inflows as seen in the Figure 1.

- spray air is a small inflow through an air assisted spray that helps forming smaller drops
- swirl inflow around the spray that helps stabilizing the spray and the flame
- the main air is required to burn the fuel and is added in the third layer.

In order to decrease the computational effort only a 30° segment of the combustor is simulated. Further only the spray and not the combustion is simulated.

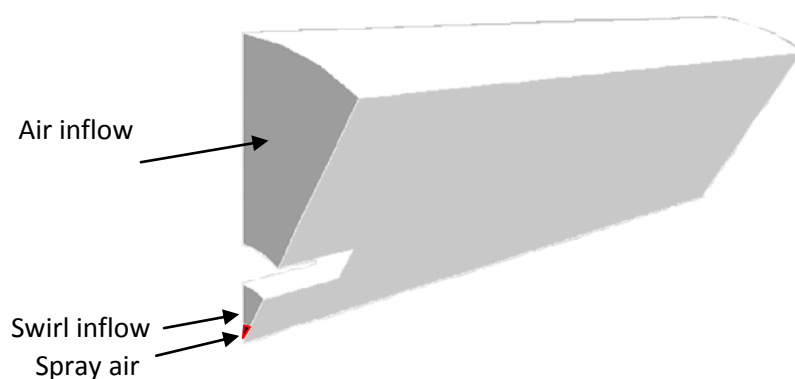


Figure 1. Schematic view of the spray combustor (3D- model with periodic boundary).

The drawing of the combustor is shown in Figure 2. NOTE: all dimensions are centimeter.

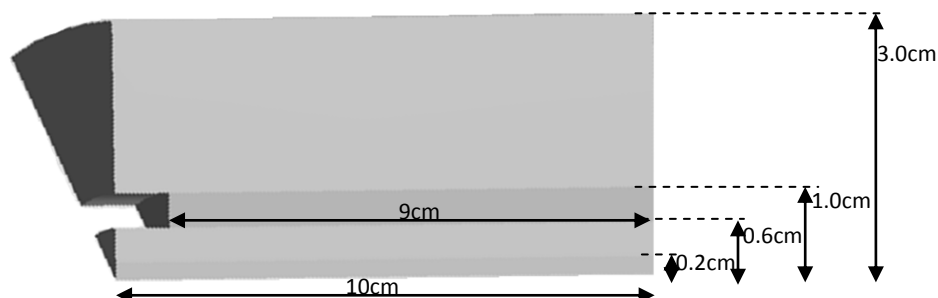


Figure 2. Drawing of the 3D combustor model.

The combustor is operated under the conditions given in Table 1.

Table 1. Boundary conditions.

	Velocity Magnitude	Temperature
Spray air	5 m/s	293 K
Swirl inflow (equal flow in axial and tangential direction)	5 m/s	300 K
Air inflow	20 m/s	320 K
Spray	70 m/s	293 K

Physical parameters:

Methanol vapor pressure (Stepwise linear function of temperature)

All other properties are approximated with constant values

Modeling:

The spray is modeled using Euler-Lagrange approach. As described in Chapter 6 the forces acting on a single particle are given by:

$$m_d \frac{dU_{i,d}}{dt} = F_{i,Drag} + F_{i,Press} + F_{i,Virt} + F_{i,History} + F_{i,Bouy} + F_{i,Lift} + F_{i,Therm} + F_{i,Turb} + F_{i,Brown} \quad (6.9)$$

Here $U_{i,d}$ is the linear velocity of the particle, m_d is the mass of the particle. The right hand side contains the sum of all forces acting on the particle. You find an explanation to all these forces in *Chapter 6.2 -6.4* in the text book.

Application specific models

In spray modeling the Rosin – Rammler distribution of drop sizes is often used

$$f(x; \lambda, k) = \begin{cases} \frac{k}{\lambda} \left(\frac{x}{\lambda} \right)^{(k-1)} e^{-(x/\lambda)^k} & x \geq 0, \\ 0 & x < 0, \end{cases}$$

where $k > 0$ is the distribution shape parameter and $\lambda > 0$ is the spread parameter of the distribution. The shape parameter can be calculated from the mean diameter $\bar{x} = \lambda \Gamma(1 + 1/k)$ where Γ is the gamma function. Different size distributions are shown in Figure 3.

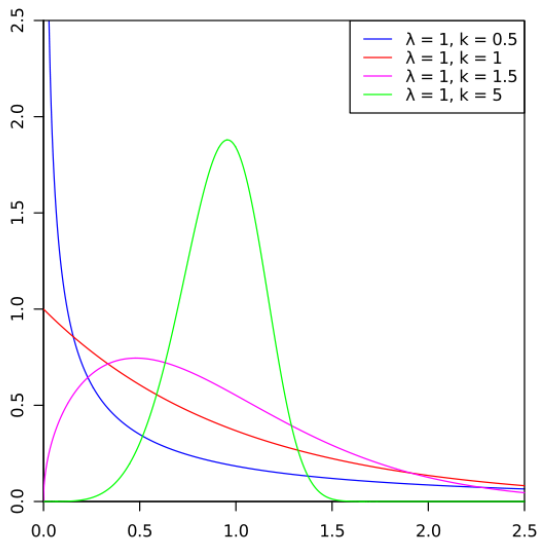


Figure 3. Size distributions.

1.2 Tutorial prerequisites

It is recommended that you have read Chapters 1-4 and 6 in the text book (*Computational Fluid Dynamics for Engineers*) and that you have completed Tutorial 1 (preferably also Tutorial 2). Regardless of what CFD software you use, you will find all required information in Section 1. The questions and tasks related to this tutorial is found in Section 2.

If you use Ansys Workbench you should read Sections 3-5 in this manual to get detailed instructions on how to setup the simulations. However, steps already explained in Tutorial 1 and 2 are not shown with screen dumps. If you use any other CFD software you should use the data given in Section 1 to run the simulations and answer the questions in Section 2.

2. Tasks and report instructions

Turn in a report that focus on analysis of the simulation results and on the following questions:

- What do the different particle forces in Equation 6.9 represent? (*Hint Chapter 6.2*)
- Evaluate and discuss if virtual mass, buoyancy, turbulent and Brownian forces should be included. (*Hint Chapter 6.2*)
- How is the drag force model corrected for the effect of other particles nearby, at higher particle loading? (*Hint Eq. 6.41*)
- For fluid particles the drag coefficient can be corrected for the internal circulation. What physical properties determine how large this correction is compared to solid particles? Evaluate if correction for the internal circulation within the fluid particles should be included in these simulations. (*Hint Chapter 6.4.1 and Eq. 6.44*).
- The drag coefficient should be corrected for Knudsen numbers above 0.02. For what drop sizes can this be neglected? (*Hint $Kn = \lambda / D_p$ and λ is the mean free path of the molecules in the gas i.e. ≈ 80 nm at 300 K 1 atm. Figure 6.9 in the book.*)
- How is the drop evaporation rate i.e. the mass and heat transfer calculated? (*Hint Chapter 6.4.3*)
- What differences between simulation loop nr 2 and loop nr 3, can you see and how do you explain this?

Note: All hints refer to the location in the textbook where you will find an answer.

Simulation loop nr 1

Simulate the flow, according to specifications in Section 1, without spray and analyze the flow pattern using pathlines.

Simulation loop nr 2

Simulate methanol drops with uniform diameter $d_p = 1 \times 10^{-4}$ [m] released from the Spray air surface using total flow rate 1×10^{-8} kg/s. The spray simulation is done using Eulerian-Lagrangian simulations with transient simulation of the drops and a steady state simulation of the continuous phase. Since the number of drops increases initially the simulation is not valid until the number of drops in the volume reach a constant value. The drops are released until steady state is achieved in the system with respect to particle loading. For time being just set a time 100 s and use time step 1×10^{-4} s.

Track

- particle size
- temperature
- residence time
- particle velocity
- particle Reynolds number
- number in parcel

Pressure, Basset, thermophoretic and lift forces should not be included.

Simulation loop nr 3

Decrease the diameter to $d_p=2.5e-5$ [m] and repeat the analysis (as described in Simulation loop nr 2).

Simulation loop nr 4

Use a Rosin-Rammel drop distribution with an average diameter equal to $d_p=2.5e-5$ [m] and spread parameter 3.5. Calculate for 10 diameters. Repeat the analysis described in Simulation loop nr 2.

Simulation loop nr 5

Simulate an air blast spray with 10 particle streams. Inner diameter 1 mm and outer diameter 5mm. Use default parameters for the remaining parameters.

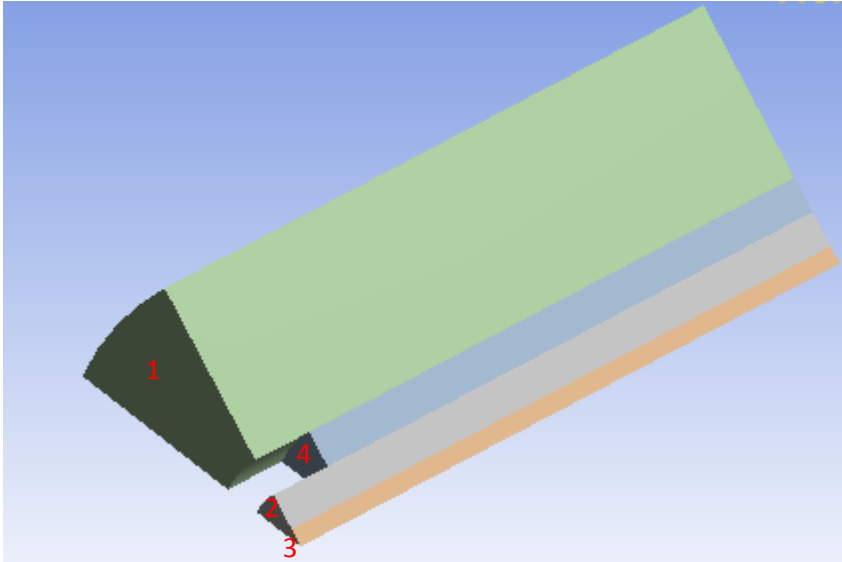
3. Instructions for generating the geometry (Ansys 13 Design Modeler)

The basic CAD operations where explained in Tutorial 1. Please refer to Tutorial 1 if you need to refresh your memory.

Start Workbench and Deign Modeler

Select centimeter

Generate the volume shown in Figure 2. Three of the surfaces start from the same surface and one starts 1 cm downstream. Make the surfaces and extrude them to the desired volumes.



First you need to generate the sketches

Draw Arc by Center in XY-plane (Sketching mode)

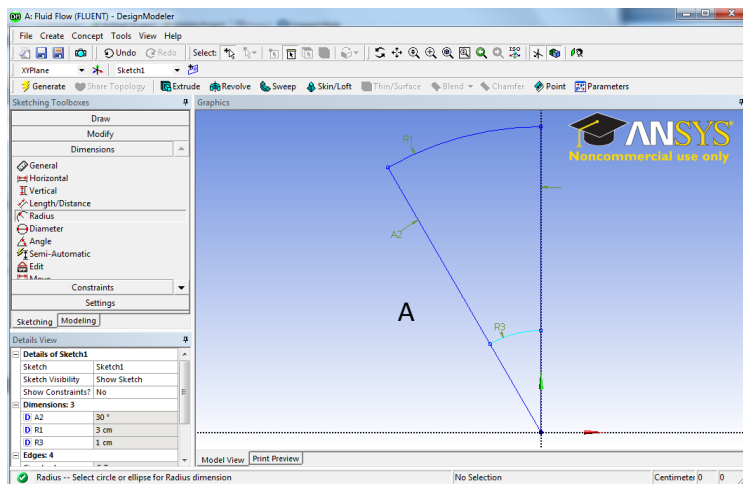
Set Radius to 3 cm (Dimension-Radius)

Draw lines from origo to the arc

Dimension- Angle –(Mark vertical line then A) to 30° .

(Make sure the first line turn yellow when you select it)

Draw the second arc and **Set Radius** to 1 cm.

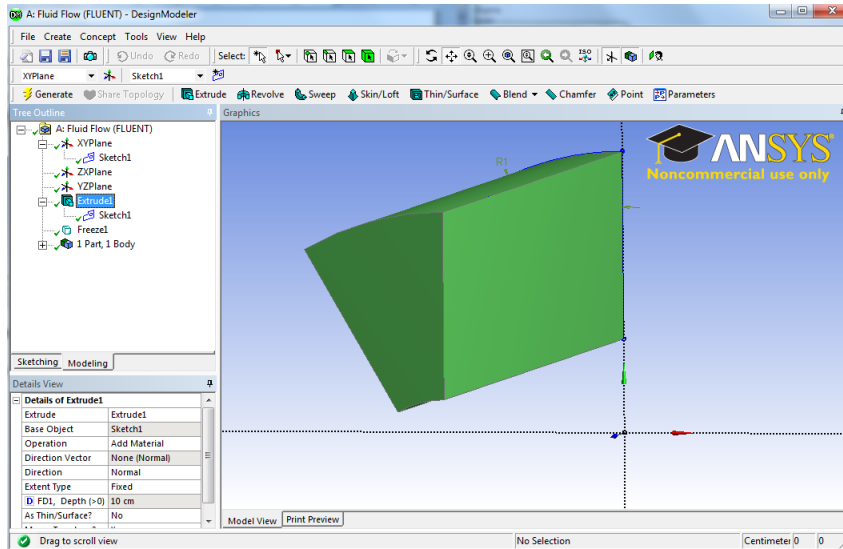


Remove the inner part of the lines by **Modify-Trim**

Mark Sketch 1 in Modeling (modeling mode) and **Generate**

Mark Sketch1 **Extrude** (normal to surface) 10 cm **Generate** and **Freeze**

Now you have generated the first volume as shown in the Figure below.



The next step is to generate the remaining three volumes.

New Sketch

Make the second segment with outer radius 0.6 cm and inner radius 0.2 cm and 30° .

Mark Sketch2 **Extrude** (normal to surface) 10 cm **Generate** and **Freeze**

New Sketch

Make the third segment with radius 0.2 cm and 30° .

Mark Sketch3 **Extrude** (normal to surface) 10 cm **Generate** and **Freeze**

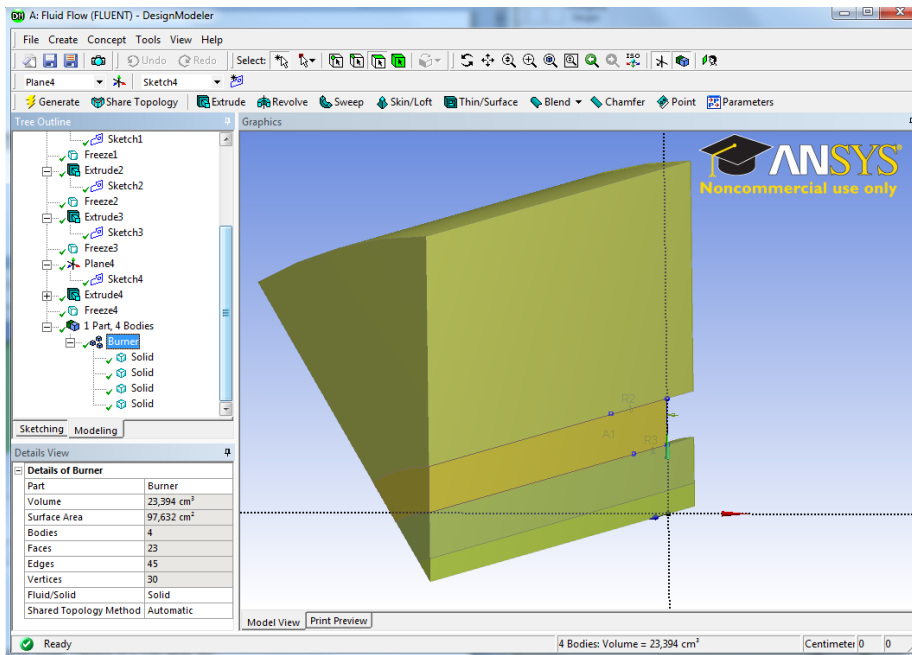
New Plane – Transfrom – Offset Z 1 cm **Generate**

Make the forth segment on the new plane with outer radius 1 cm, inner radius 0.6 cm and 30° . **Generate**
Extrude the new segment 9 cm (**Generate-Freeze**)

You have now all four volumes required, shown in the Figure below. You should now generate a multibody part.

Tutorial 3 - Swirl stabilized spray combustor

Computational Fluid Dynamics for Engineers, Cambridge University Press, 2011.



Form New Part Combine the four bodies into a new part and Rename

Save Project

4. Instructions for generating the computational mesh (Ansys 13 Meshing Platform)

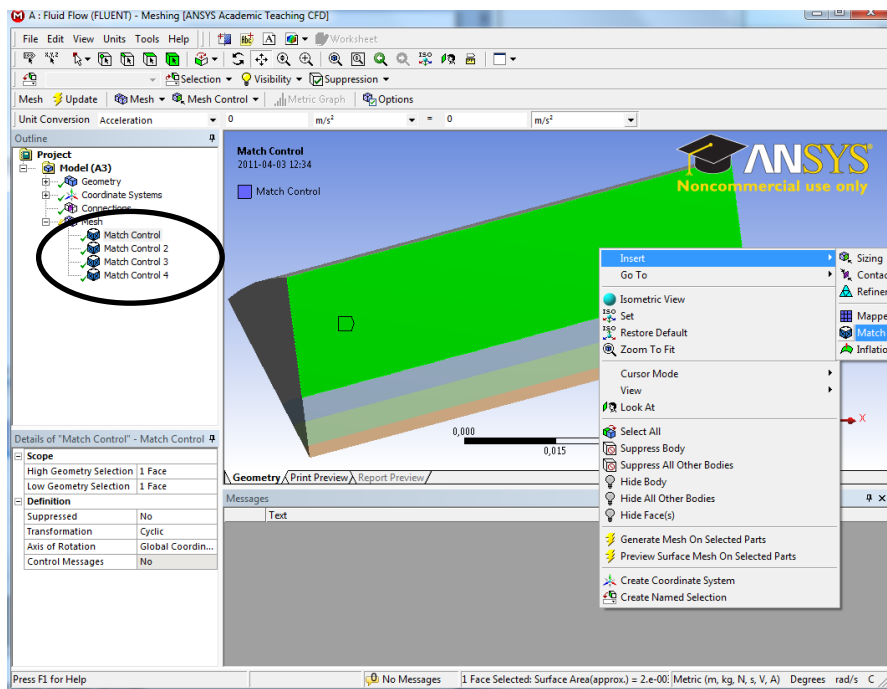
The basic Meshing operations where explained in Tutorial 1. Please refer to Tutorial 1 if you need to refresh your memory.

There are four pair of surfaces that are part of the periodic boundary.

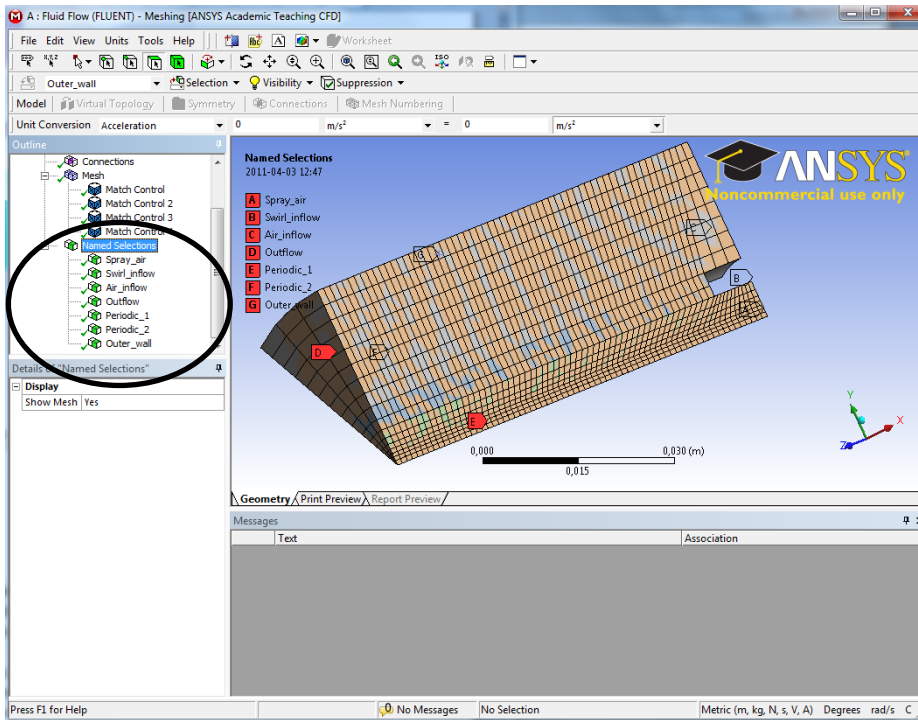
Select the pair of surfaces that are part of angular periodic rotation right click and select **Match-Control**.

Repeat for the remaining three surfaces.

You now have four match controls as shown in the Figure below.



Mesh the volume using Medium relevance center and default settings



Specify names for all surfaces and the volume, as shown in the Figure above. The two periodic surfaces should contain all four surfaces on the periodic surface.

Update

Save Project

5. Instructions for running the simulations (Ansys 13 Fluent)

The basic Fluent operations where explained in Tutorial 1. Please refer to Tutorial 1 if you need to refresh your memory.

In Workbench - Right click **Setup** and Edit to start Fluent. Start Fluent in double precision.

Find the periodic boundaries in Boundary conditions menu and identify their ID:s (2 numbers)

In the text interface(TUI) write (shown in the Figure below):

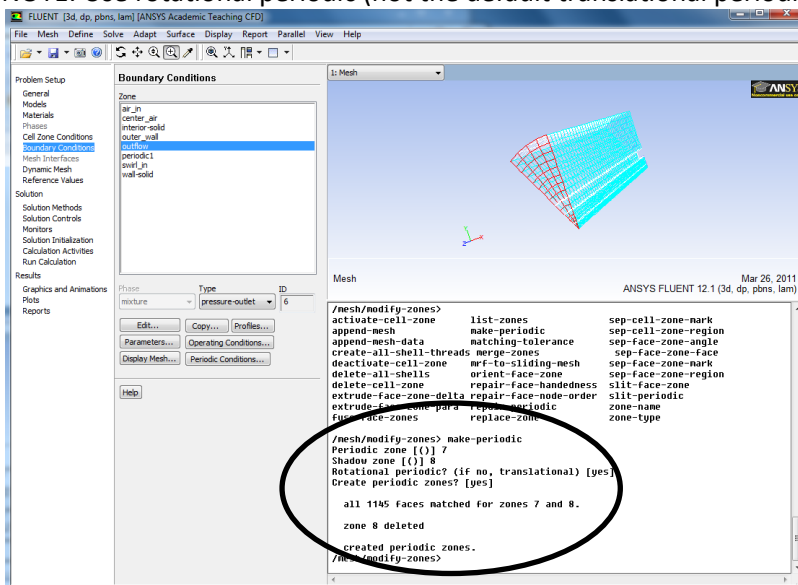
Mesh/modify-zones/make-periodic

Periodic zone [()] write the first ID number

Shadow zone [()] write the second ID number

NOTE: The ID-numbers are not necessary the same number as shown in this Figure.

NOTE: Use rotational periodic (not the default translational periodic)



Ensure that all faces matched for these two zones (is written in the prompt). If not you have to go back and check that you applied Match controls correctly in the Meshing program.

Mesh - Reorder

This instruction rearranges the mesh indexes to optimize the use of cache memory.

Set Energy on and choose a proper turbulence model

Boundary conditions. Set the inflow conditions including 5% turbulence intensity and hydraulic diameter $\sim 1/5$ of the inlet size. The swirl is set by local cylindrical coordinates with equal tangential and axial velocities.

Convergence may be improved if unrealistic values can be avoided. Reasonable pressure range and viscosity ratio should be set in **Solution Controls-Limits**. The minimum and maximum temperatures must be set because the available methanol vapor pressure is restricted to a given temperature range.

Initialize and solve the flow field

For viewing purpose make the following two settings

Make a surface that is in the middle of the segment.

Surface-Iso-Surface-Mesh-Angular Coordinate Iso-Values 15

Name e.g. angle=15

Display the full volume

Display-Graphics and Animations-Views-Define

Select fluid Rotational

12 number of repeats ($12 \times 30 = 360$)

Display the flow pattern by using **Display-Pathlines**

Try the different inlet surfaces and pulse/continuous and obtain a view of the flow.

The drops contain methanol that evaporates and species transport must be included.

Species-Species Transport

Mixture Material

methyl-alcohol-air

Apply

Models-Discrete phase

Interactions with continuous phase

Number of Continuous Phase Iterations per DPM Iteration 10

Unsteady Particle Tracking

Time step 0.001

Number of Time Steps 1

Tracking Parameters

Max Number of Steps 500

Step Length Factor 5

Spherical drag law

Injection-Create (as shown in the Figure below)

Injection Type - Surface Select Spray air

Particle Type Droplet

methyl-alcohol-air –

Evaporation Species ch3oh

Uniform

Point Properties

z-velocity 5 m/s

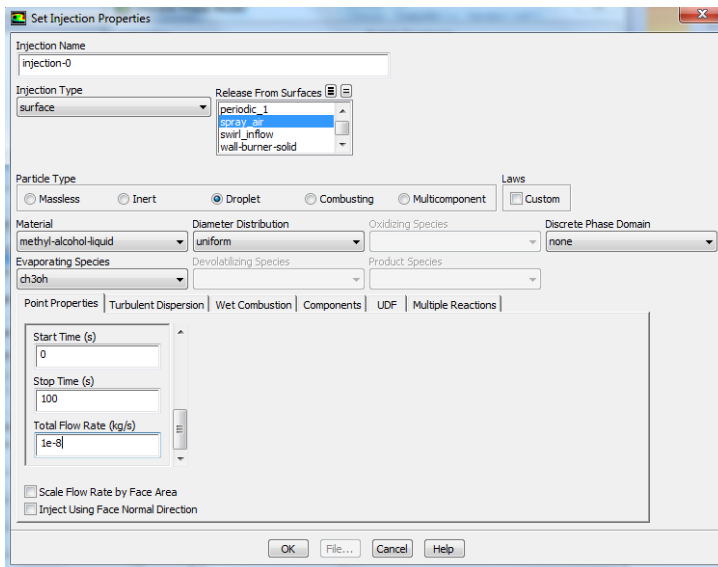
Particle diameter $1e-4$ m

temperature 300 K

Start Time 0

Stop Time 10

Total Flow Rate $1e-8$ kg/s



RANS modeling will provide a time averaged turbulent flow and the injected particles will follow exactly the same trajectory each time. An estimation of the random effects of turbulence can be obtained by including Discrete Random Walk (DRW) and comparing two trajectories of particles with and without DRW released from the same point. Discrete random walk can be found under **Turbulent dispersion**.

Close discrete phase menu

Define the material properties:

Materials

Droplet Particle

Cp - piecewise-linear

Saturation Vapor Pressure - piecewise-linear

Steady state is reached when the concentration of methanol in the outlet or the number of drops in the volume reach steady state

Specify the solution controls:

Monitors – Surface Monitors

Mass weighted average of methanol molar fraction

Monitors – Volume Monitors

Axis exponential

Report type Sum-Discrete Phase model – DPM Mass source

Post-processing:

Display vectors and contours of temperature and methanol concentration in a plane 15° .

Display Particle Tracks

- particle size,
- temperature
- residence time
- particle velocity
- particle Reynolds number
- number of particles in parcel

Report Discrete Phase summary