
Particle Shadow and Enhancement Zones on Aircraft

FLUENT CFD Tutorial **by Markus Hermann,**

IFT, Leipzig **January, 2012**

Introduction

The purpose of this tutorial is to illustrate the setup and solution of a CFD simulation to determine the airflow around an aircraft geometry. Moreover, the trajectories of aerosol particles or cloud droplets around that aircraft shall be calculated. For convenience the term “particle” is used for all kinds of aerosol particles, cloud droplets and ice crystals. This tutorial demonstrates how to:

- Set-up a CFD model for an external aerodynamics flow problem
- Model the compressible flow around an aircraft body (flow field)
- Calculate particle trajectories around the aircraft using the calculated flow field

The study of flow around an aircraft is crucial for determining the correct mounting position of an inlet systems or aerosol instruments at an aircraft. The correct position guarantees an undisturbed airflow and hence representative particle measurements, while at the other positions shadow (reduced particle concentrations) and enhancement zones (increased particle concentrations) can occur. These measurement artifacts occur preferably in regions of strong fuselage curvature, e.g., at the aircraft bow.

Problem Description

In this tutorial, we consider the flow around an artificial research aircraft called LIM_4 (Fig. 1) at 472 hPa pressure (6 km altitude), 249 K ambient temperature, an angle of attack of $\alpha = 3^\circ$, and a free stream Mach number (M_∞) of 0.35 - 0.55 (TAS = 111 -174 m/s). The flow field calculations shall be performed with one Mach number only, **please ask the supervisor for the Mach number you should use**. For the simulation, the LIM_4 aircraft, about 20 m long, was embedded in a 340 m x 80 m model domain with about 49,300 grid cells. At the upper LIM_4 fuselage an inlet system should be mounted 5 m behind the aircraft nose. The question is, how large must the inlet pylon be in order to measure well outside the aircraft boundary layer and the particle shadow and enhancement zones.

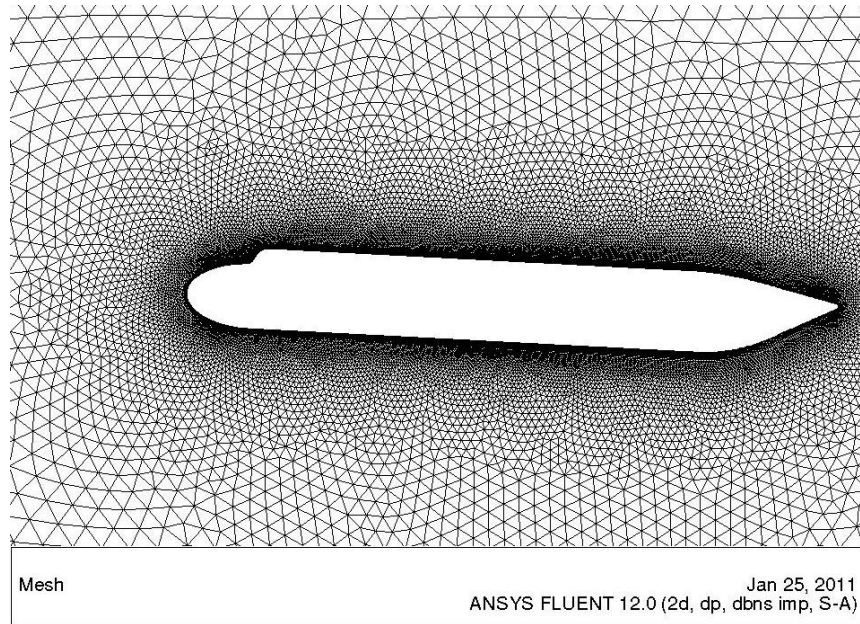


Figure 1: LIM_4 geometry and mesh.

Prerequisites

This tutorial assumes that you read this tutorial and the „CFD Intro“ and „FLUENT graphical user interface“ documents provide with this tutorial well in advance before starting with the tutorial execution. Please spend some minutes to get familiar with the FLUENT graphical user interface.

Preparation

Copy the mesh file „LIM_4.msh“ to your working folder.

Start the 2D double precision (2ddp) version of FLUENT by prompting „fluent13.sh &“.

Setup and Solution

Step 1: Grid

1. Read the mesh file, LIM_4.mesh

File → Read → Mesh...

FLUENT will read the mesh file and report the progress in the console window.

2. Check the mesh.

Mesh → Check

This procedure checks the integrity of the mesh. Make sure the reported minimum volume and face area are positive numbers.

3. Scale the grid.

Mesh → Scale...

Check the domain extents to see if they correspond to the actual physical dimensions. The mesh was created in meters, if this is not indicated, change the length unit and scale the mesh.

4. For a better contrast change to the “Classic” display scheme.

Display → Options... → Color Scheme → Classic → Apply

Click “Close”.

5. Use the magnifying lens button and the left mouse button (hold it and draw a rectangle) to zoom in on the image so that the mesh near aircraft can be viewed. Please note the finer mesh at the aircraft bow and generally at the aircraft fuselage which is necessary for an appropriate flow modeling in the vicinity of a surface.

Step 2: General Settings

1. Set the general flow problem setting.

Define → General → ...

(a) Type: Change the Type setting to “Density-Based”.

This setting is related to the underlying numerics of the model. Generally, the density-based approach is recommended for high-speed compressible flows, which is the case here.

(b) Time: Keep the default “Steady” setting.

The flow problem under investigation is steady, i.e. independent of time.

(c) 2D Space: Keep the default “Planar” setting.

When calculating a flow problem in only two dimensions (2D), which actually saves a lot of computing time compared to a fully 3D calculation, the model must know how to treat the third dimension. The “Planar” setting assumes that in the 2D geometry is expanded to the third dimension up to infinity. This is of course not the case for an aircraft body (it is rather axial symmetric), but for simplification the “Planar” setting is used here. The effect of the aircraft body on the particle trajectories can be shown as well with this setting.

Step 3: Models

1. Set the solver settings.

Define → Models → ...

(a) Energy: Click the “Edit...” button and enable the energy equations in FLUENT.
Click “OK”.

The energy equations are needed when calculating for instance flow problems which require heat transfer.

(b) Viscous: Enable the “Spalart-Allmaras” turbulence model. Use the standard options as indicated when activating the Spalart-Allmaras turbulence model.

Generally, under “Viscous” the kind of turbulent flow treatment in the model is set. The Spalart-Allmaras model is a simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. It was designed for aerospace applications involving wall-bounded flows and has shown to give good results for boundary layers subjected to adverse pressure gradients.

Step 4: Materials

The default working fluid material in this flow problem is air. The default settings need to be modified to account for compressibility (recommended for $MA > 0.3$) and variations of the thermophysical properties with temperature.

1. Define the fluid material properties.

Define → Materials → Fluid → Create/Edit

(a) Density: Select “ideal-gas” from the “Density” drop-down list.

(b) Specific Heat: Set the “Specific Heat” to 1004.8 (constant).

(c) Thermal Conductivity: Set the “Thermal Conductivity” to 0.0222 (constant).

(d) Viscosity: Select “Sutherland” from the “Viscosity” drop-down list.

(e) Click “OK” to accept the default Three Coefficient Method and other parameters.

(f) Click “Change/Create” and close the Materials panel.

Density and Viscosity have been made temperature-dependent. For high-speed compressible flows, it is recommended to allow thermal dependency of the physical properties. But for simplicity, Thermal Conductivity and C_p are assumed to be constant in this case.

Step 5: Operating Conditions

1. Set the Operating Conditions.

Define → Operating Conditions...

2. Set the “Operating Pressure” to 0 Pascal.

FLUENT uses three kinds of pressure definitions. The “absolute pressure” is the static pressure you know and is given by the sum of the “operating pressure” and the “gauge pressure”. The “gauge pressure” is used by FLUENT for the calculations to avoid the numerical problem of

round-off errors. If the flow velocity is low the pressure changes only little with changing velocity. Therefore FLUENT subtracts the “operating pressure” from the “absolute pressure” (two large roughly equal numbers) in order to get these pressure changes (“gauge pressure”) more precisely. For compressible flows, where the velocity and the pressure changes are substantial, it is recommended to set the operating pressure to zero.

3. Set the “Reference Pressure Location” x-value to -140 m.

The reference point should be located far away from the geometry of interest, where undisturbed flow conditions prevail.

Step 6: Boundary Conditions

1. Set the boundary condition for pressure-far-field (far-field).

Define → Boundary Conditions...

(a) Select “far_field” from the Zone list.

(b) Click the “Edit...” button.

i. Enter the ambient air pressure for the “Gauge Pressure”.

ii. Enter your given Mach number (0.35-0.55) for the “Mach Number”.

iii. Enter 1 for “X-component of Flow Direction” and 0 for “Y-component of Flow Direction”.

In principle it is possible in two ways to account for the angle of attack in the simulation, either to align the aircraft center line with the x-axis and to apply an x- and y-component of the wind vector here, or to place the aircraft at an angle of attack against the x-axis and to have only an x-component of the wind vector, as it is done in this tutorial.

iv. Select “Turbulent Viscosity Ratio” from the “Specification Method” drop-down list.

v. Retain the default value of 10 for “Turbulent Viscosity Ratio”. For external flows, the turbulent viscosity ratio should be set between 1 and 10.

vi. Click to the “Thermal” folder.

vii. Enter the ambient air temperature for the “Temperature”.

viii. Click “OK” to close the “Pressure Far Field” panel.

2. Close the Boundary Conditions panel.

Step 7: Save the case file

1. Save the case file (LIM_4.cas).

File → Write → Case...

Generally, the case file (.cas) saves the mesh together with the model parameters, while the data file (.dat) saves the flow field data (temperature, pressure, velocity ...) for each grid cell.

Step 8: Solution of the Flow Field

1. Change to first-order-upwind scheme for “Flow” in the “Spatial Discretization” panel.

Solve → Methods...

Normally, second-order upwind schemes are more accurate and are recommended for compressible and high Mach number flows. However in this tutorial to speed-up convergence time at the beginning “only” the first-order-upwind scheme is used for the flow equations.

2. Retain default solver settings for “Solve” → “Controls...”.

Under Solve → Controls numerical variables for the solution of the flow problem can be set. Here the default settings are used, but for other applications different settings might be necessary.

3. Initialize the flow.

The Initialization defines the simulation starting value of each variable in every cell.

Solve → Initialization...

- (a) Select “far_field” from the “Compute from” drop-down list.

For simulations of airflow around an aircraft initialize the flow field using the pressure-far-field boundary as it helps to converge faster.

- (b) Click “Initialize” and close the “Solution Initialization” panel.

4. Enable the plotting of residuals during the calculation.

Solve → Monitors... → Residuals → Edit...

- (a) Enable “Plot” from the “Options” list.

- (b) Note that the “Absolute Criteria” for the residuals are 1E-3.

The “Absolute Criteria” values tell FLUENT when to stop the iteration process, i.e., when all residuals are below the values specified here.

- (c) Click “OK” to close the “Residual Monitors” panel.

5. Create a point monitor at the inlet position.

To check whether the flow simulation is converged or not, the residuals are only on indicator. A very useful second indicator is a point or surface monitor, by which flow variables can be monitored. When the variables have reached a constant value the simulation is likely converged.

If the residuals seem to be converged (showing a “horizontal” line) but the variable point or surface monitor still indicates strictly monotonic changes in the variable, the simulation is not converged, because physics is still changing.

(a) Display only the aircraft hull and the inlet.

Display → Mesh...

- i. Select only the “inlet” and the “aircraft” surface from the “Surfaces” list.
- ii. Click “Display”.
- iii. Click “Close”.

(b) Create a point monitor for the velocity magnitude.

Solve → Monitors...

(c) Click “Create” under “Surface Monitors”.

(d) Click “New Surfaces” → “Point...”.

(e) Click “Select Point with Mouse”.

(f) Click with the right mouse button on the middle of the leading vertical edge of the inlet.

This position of the point monitor will allow you to check the flow properties in the aircraft boundary layer at the point of interest.

(g) Provide a name to the “New Surface Name” input.

(h) Click “Create”.

(i) Click “Close” in the “Point Surface” panel.

(j) Click on the new point monitor under “Surfaces” in the “Surface Monitor” panel.

(k) Set the “Report Type” to “Vertex Average”.

“Vertex” is in FLUENT another term for a point.

(l) Set the “Field Variable” to “Velocity...”.

(m) Enable “Plot” from the “Options” list.

(n) Click “OK”.

FLUENT now knows that there is one point at the inlet leading edge where it should report the velocity magnitude to you at each iteration step.

6. Save the case file (LIM_4.cas).

File → Write → Case...

7. Allow for two monitors (one for the residuals and one for your point monitor) to be displayed during the iteration process by changing the graphics window layout (right button in the second menu row).

8. Start the calculation.

Solve → Run Calculation...

(a) Set the “Number of Iterations” to 2000.

(b) Set the “Reporting Interval” and the “Profile Update Interval” to 3.

FLUENT will now update the graphics and the plotted numbers only every third iteration step. This saves a little bit of iteration time.

(c) Click “Calculate” and wait until the simulation is converged.

Iterate till the solution converges for default convergence criteria. The solution will converge in about 1300 iterations. While converging you can see in the point monitor window, how the velocity at your point monitor changes from the initial value to the final value. Please observe the amplitudes of the velocity while FLUENT is searching for the correct solution.

9. Save the case and data file (LIM_4.cas and LIM_4.dat).

File → Write → Case & Data...

Overwrite existing files.

10. Although the residuals have all reached values below 1E-3 the point monitor indicates that the simulation is not finally converged. Go back to Point 4 of this Step and set the “Absolute Criteria” for the residuals to 1E-5.

11. Change now to second-order-upwind scheme for “Flow” in the “Spatial Discretization” panel.

Solve → Methods...

12. Restart the calculation with 4500 iterations and wait until convergence is reached.

... which might take a while.

13. Save the case and data file (LIM_4.cas and LIM_4.dat).

File → Write → Case & Data...

Overwrite existing files.

Step 9: Postprocessing

1. Display velocity vectors.

Display → Graphics and Animations... → Vectors → Set Up...

(a) Enter 3 for the “Scale”.

(b) Click “Display”.

(c) Zoom into the graphics display. In particular have a closer look at the aircraft boundary layer (how the velocity is reduced when approaching the aircraft skin) and the zone of flow recirculation just upstream the cockpit window.

2. Display the pressure distribution.

Display → Graphics and Animations... → Contours → Set Up...

(a) Select “Pressure...” and “Static Pressure” from the “Contours of ” drop-down lists.

(b) Enable “Filled” in the Options list.

(c) Click “Display”.

Zoom in and out to see different regions around the aircraft more in detail. Note the regions of different static pressure, indicating that at the point of measurement the static pressure is not necessarily equal to the ambient pressure far away from the aircraft.

3. Display the temperature distribution.

(a) Select “Temperature...” and “Static Temperature” from the “Contours of ” drop-down lists.

(b) Click “Display”.

Note the regions of different static temperature around the aircraft. In particular note that the aircraft body influences the static temperature already several fuselage diameters upstream the aircraft, thus potentially influencing particle size and chemical composition.

4. Check the Y^+ values around aircraft.

Display → Plots → XY Plot → Set Up...

(a) Select “Turbulence”... and “Wall Yplus” from the “Y Axis Function” lists.

(b) Select “aircraft” in the “Surfaces” list.

(c) Click “Plot”.

The values of Y^+ are dependent on the mesh and the Reynolds number of the flow, and are meaningful only in boundary layers. The value of Y^+ in the wall-adjacent cells indicates if the mesh is correctly chosen (too fine or too coarse mesh) for the chosen turbulence model. For the Spalart-Allmaras model, Y^+ should either be very small (in the order of $Y^+ = 1$) or greater than 30. In this case, the Y^+ maximum values is slightly high with a maximum at about 400, however the majority of values is in an acceptable range of below 250.

At this point call your supervisor in order he can check if the simulation is converged and the flow field is ok, before you start with the particle trajectory calculations.

Step 10: Particle Trajectories

For the calculations of the particle trajectories FLUENT uses the Discrete Phase Model (DPM).

1. Set the number of particle iteration steps in the DPM.

Define → Models → Discrete Phase... → Edit...

- (a) Set the “Max. Number of Steps” to 3000.
- (b) Click “OK” to close the “Discrete Phase Model” panel.

2. Define a particle injection (injection-0).

Define → Injections...

- (a) Click the “Create” button to open the “Set Injection Properties” panel.
- (b) Select “group” from the “Injection Type” list
- (c) Increase the “Number of Particle Streams” to 30.
30 particle trajectories will be calculated.
- (d) Set the “X-Position” of the “First Point” and “Last Point” to -20 m.
Particles will start 20 m upstream of the aircraft bow.
- (e) Set the “Y-Position” of the “First Point” to -1.2 m and for the “Last Point” to 1.6 m.
- (f) Set the “X-Velocity” of the particles for both points to the free stream air velocity.
- (g) Set the “Diameter” for both points to 100 μm .
- (h) Set the “Temperature” of the particles for both points to the free stream temperature.
- (i) Click “OK”
- (j) Close the “Injections” panel.

3. Define particle boundary conditions.

Define → Boundary Conditions... → aircraft → Edit...

- (a) Click on the “DPM” folder.
- (b) Set the “Boundary Cond. Type” to “trap”.
FLUENT will treat and report all particles which hit the aircraft skin as “trapped”, i.e. sticking to the aircraft fuselage and lost for further trajectory calculations.
- (c) Click “OK”.

4. Save the case file (LIM_4.cas).

File → Write → Case...

5. Display the particle tracks.

Display → Graphics and Animations... → Particle Tracks... → Set Up...

(a) Click on “injection-0” in the “Release from Injections” panel.

(b) Click “Track”.

FLUENT calculates how many particles are trapped on the aircraft skin and how many particles leave the domain to the right domain boundary (“escaped”).

(c) Click “Display”.

FLUENT “tracks” the particles once again but in addition also displays the particle trajectories in the graphics window.

(d) Zoom in and have a closer look onto the particle trajectories.

(e) Calculate particle trajectories for at least 12 particle diameters in the range 1 μm to 0.1 mm.

i. Go back to Point 2 of this Step and create additional injections.

ii. Use the possibility to make copies of “injection-0” to reduce the input effort.

iii. Click the “Set” button to modify a highlighted injection.

(f) After calculating the particle trajectories zoom in to have a closer look at the shadow and enhancement zones and how they change with particle size.

Summary

In this tutorial, you have learned how to set-up and solve an external aerodynamics flow problem in a CFD program, to determine the flow field around an aircraft geometry, and to calculate particle (aerosol particle, cloud droplet, or ice crystal) trajectories around that aircraft. Of course, there are some simplifications in the current model, e.g., the pure 2D model geometry, ideally the calculations would be carried out in 3D. But the principle steps and considerations are always the same. Actually, you have now more experience in CFD modeling for airborne measurements than 95% of the research scientists conducting aircraft-borne measurements. 😊

Discussion

The following issues and questions shall be addressed in the elaboration to this tutorial:

(a) Please describe the most important steps of the solution procedure in **your own** words.

(b) Consider you want to mount an inlet system or particle measurement device at the position of the “inlet” in the model, how large (approximate distance in direction normal to the aircraft fuselage) must the inlet be in order to measure undisturbed particle concentrations?

Fluent does not allow for a direct measurement of distance, however, you can use the

Surface → Point... → Select Point with Mouse

option to click with the right mouse button at the point of interest. The “Point Surface” window will then indicate the coordinates of the point to you. To improve the trajectory density and thus the

accuracy of your measurement you can adjust the “Y-Position” range of the trajectory start position as described under Step 10, Point 2.

(c) As a large inlet is critical with respect to aircraft safety (dynamic loads, icing, bird strike, etc.) there has always to be made a compromise between the desire to measure fully undisturbed air and the height of the inlet. Which particle diameters still reach the “inlet” of the LIM_4 undisturbed, if the aviation certification engineer limits your inlet to 20 cm height?

(d) Below you find two photos of a BAe-146 and a CASA-212 aircraft, these aircraft types are used by two European countries as measurement platforms for atmospheric research. Which are the desirable mounting positions for particle inlets/instruments on both aircraft, which are undesirable positions? Please give reasons.



Figure 2: Photo of a BAe-146 aircraft.



Figure 3: Photo of a CASA-212 aircraft.