How to define Operating Density for single phase flow?

Problem/Description:

The definition of the Operating Density plays an important role in cases where free convection / buoyancy is the main driving force for fluid flow. For this it is essential to understand how it is used in Fluent. This Solution shows how the Operating Density is used in Fluent, how to define it and what could happen if doing it wrong.

Solution:

In Fluent the pressure is computed as:

$$p_s' = p_s - \rho_{operating} \cdot gravity \cdot height$$

(Fluent Users Guide, R17.1, “13.2.4.5. Operating Density”)

to avoid round off errors. This means the static pressure head because of gravity is taken out of the calculation of the pressure. When you define pressure boundary conditions you have to set the outside pressure to a value. Often zero is chosen as Gauge Pressure which is identical to ambient conditions outside of the domain. With the Reference Pressure the pressure level for the Absolut Pressure is fixed. E.g. the Reference Pressure is set to 1 atm then a value of zero as pressure BC means that there is 1 atm at the outlet.

But if the pressure BC is not horizontal or if there is more than one pressure BC it is not correct to say that the pressure is just one constant value. Because of gravity there are differences in pressure over the height - even if these differences are often small compared to the absolute pressure level this can have a significant impact on the flow inside of the domain, esp. if it is a buoyancy driven flow.

In Fluent it is implemented that the operating density is used to calculate the pressure head at the outside at any pressure BC. So it is important that this operating density is set correctly.

The default is that the operating density is deactivated in the panel but it is used in Fluent nevertheless. If no value is specified, the mean value of the density inside the fluid domain is used. This is ok if the fluid density is set to constant or for the boussinesq approach. It is incorrect if the density depends on temperature or pressure. With variation of the density inside the domain because of temperature and / or species field (where the values are the result of the simulation) the pressure at the pressure BC becomes something – but normally not the correct values.

© 2016 ANSYS, Inc. All rights reserved.

Important

It is important to understand that the operating density is used by Fluent to compute the pressure at the <u>outside</u> of the domain (as the outside is unknown to the inside of the domain and therefore has to be specified by the user).

So the operating density has to be set to the value at the outside of the fluid domain. This is the density corresponding to the values for temperature and species that are defined at the pressure BC. So e.g. if the outside temperature is 323 K and the species concentration is 79% N2 and 21% O2 the operating density has to be rho(323 K ,79% N2, 21% O2). In this case Fluent will calculate the correct pressure at the pressure BC – if not horizontal then a linear profile will be used.

Another approach is to set the operating density to zero. In this case the term

$$\rho_{operating} \cdot gravity \cdot height$$

vanishes and the pressure hast to be defined “manually” if the pressure BC are not horizontal and / or are on different level of height. For horizontal pressure BC constant values have to be used with different values at every height level. The values can be computed by

$$\rho_{outside} \cdot gravity \cdot height$$

where the zero coordinate hast to be fixed. For non-horizontal pressure boundary conditions profiles have to be used. See Solution 2042660 for defining profiles at boundaries.

Note: If there is more than one pressure BC and the conditions are different (e. g. simulation of a room at winter with an open window and an open door to the inner of the building) then strictly spoken measurement data are needed.

Attached is a project which shows the effects of incorrect operating density.

The case is a simple rectangular 2D domain. Up and down are adiabatic walls. The sides are pressure outlets with 293 K, 0.235 mol% O_2 and 0.765 mol% N_2. Density for this is 1.200295 kg/m^3. In the middle is a heated plate with 500 K. Turbulence is modeled with Realizable k-eps model. Material is incompressible ideal gas with O_2 and N_2. Double Precision Solver is used. Case is calculated transient for numerical stability.

Five cases have been calculated:

1. Operating Density set correct to 1.200295 kg/m3, Gauge Pressure = 0 Pa
2. Operating Density set to Zero and Pressure Profile Used
3. Operating Density set to 1.4 kg / m^3, Gauge Pressure = 0 Pa
4. Operating Density set to 1.0 kg / m^3, Gauge Pressure = 0 Pa
5. Operating Density not explicitly defined (default), Gauge Pressure = 0 Pa

© 2016 ANSYS, Inc. All rights reserved.

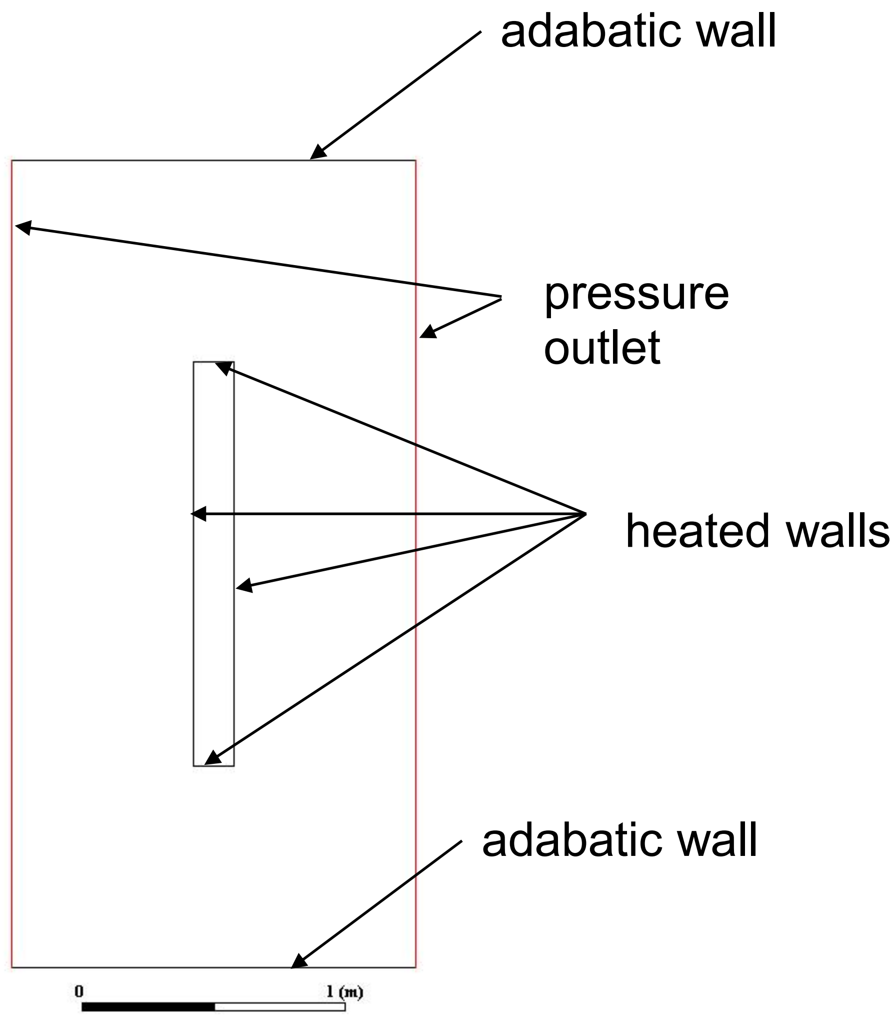

Figure 1: Case setup

© 2016 ANSYS, Inc. All rights reserved.

Results:

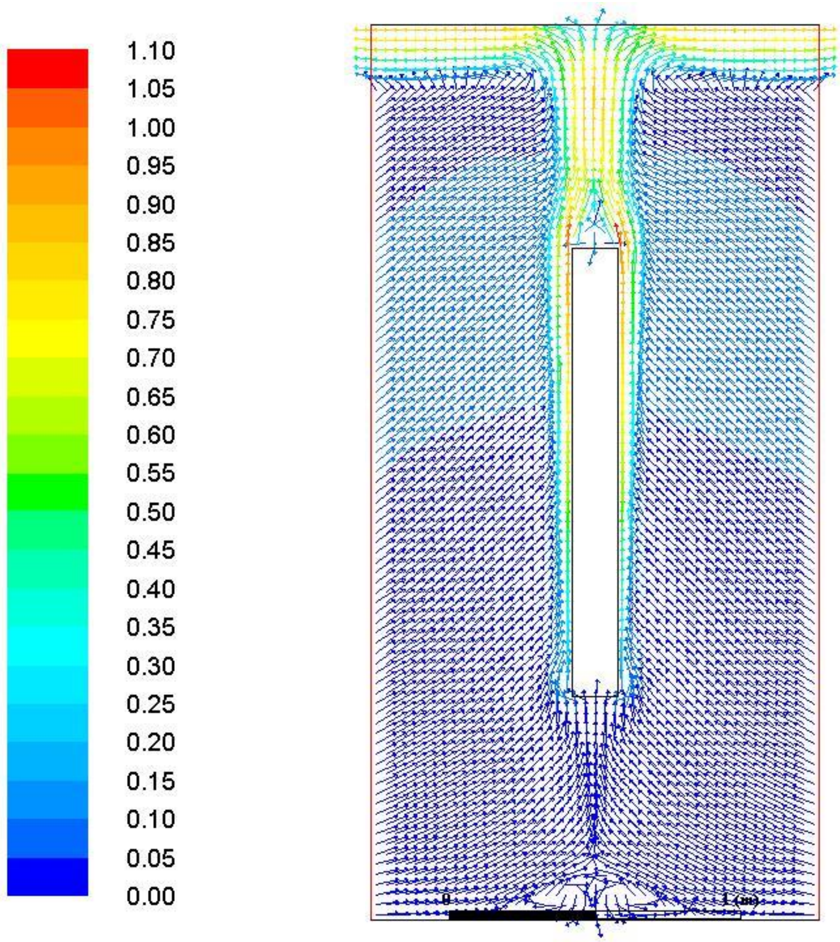

Figure 3 Correct Operating Density

© 2016 ANSYS, Inc. All rights reserved.

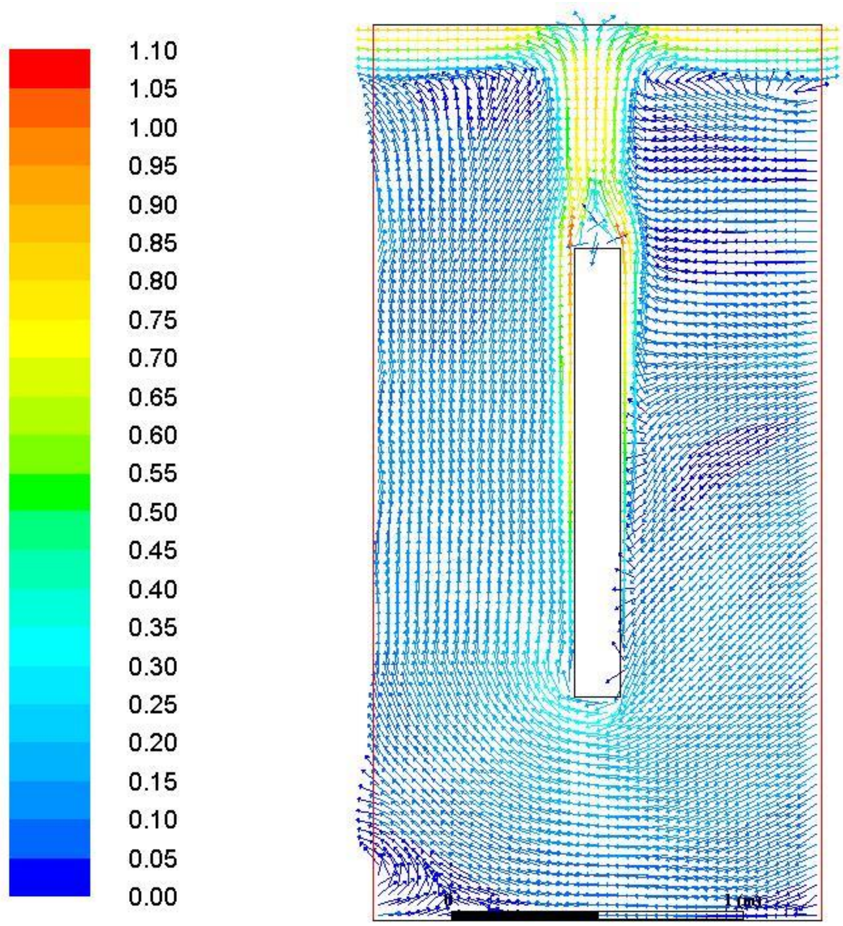

Figure 4 Operating Density set to Zero with Pressure Profile

© 2016 ANSYS, Inc. All rights reserved.

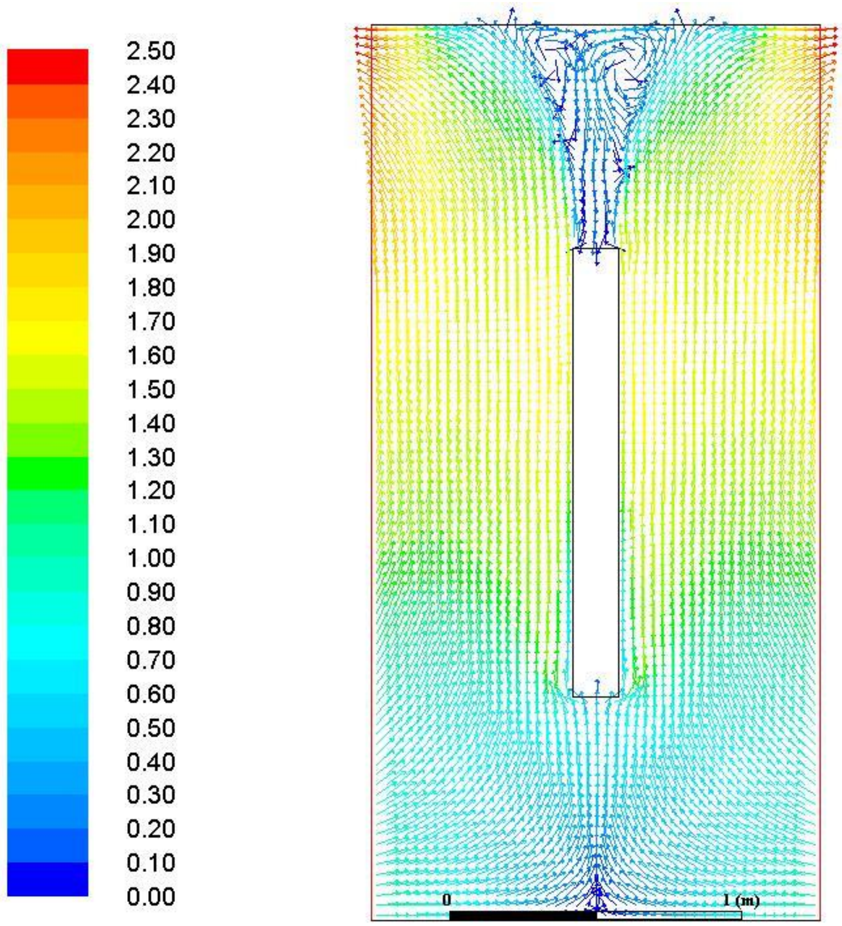

Figure 5 Operating Density set to high value (1.4 kg/m^3)

© 2016 ANSYS, Inc. All rights reserved.

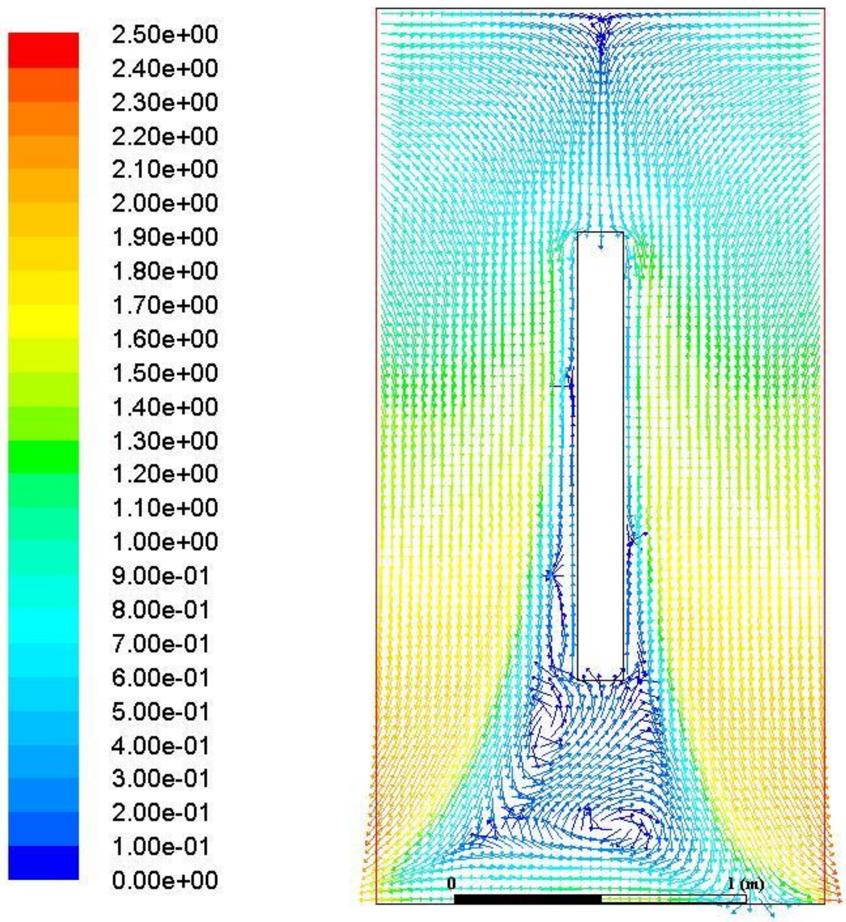

Figure 6 Operating Density set to low value (1.0 kg/m^3)

© 2016 ANSYS, Inc. All rights reserved.

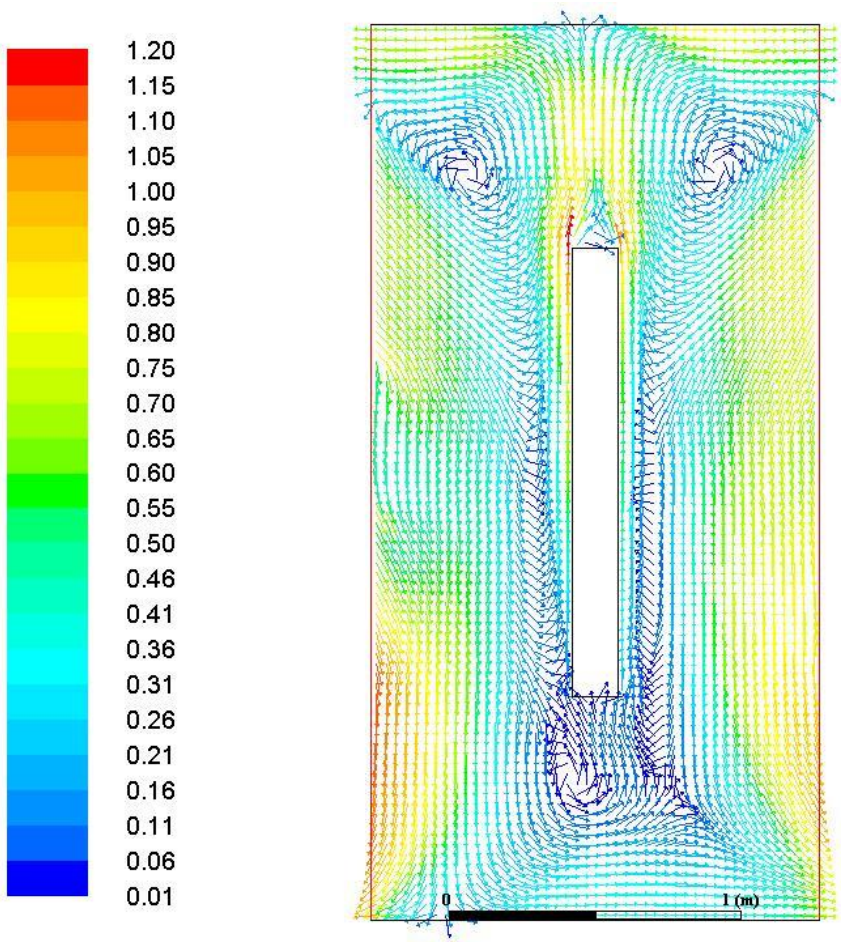

Figure 7 Operating Density set automatically (default)

All cases are not very stable. For case 1 + 2 solution is converging nicely but slow. The other cases do not converge well at all. This is because of the non-physical setup; the pressure boundary conditions are not set correctly.

It can be seen that the results with the wrong Operating Density (even with the default method) differ strongly from the correct setup. For case 3 with the Operating Density set too low the flow field is completely nonsense as the cold flow enters the domain at the top and is falling down at the heated plate to leave the domain at the bottom.

Attachments

1. 2042659_Define_Operating_Density_Single_Phase.wbpz

References

1. Fluent Users Guide, R17.1, “13.2.4.5. Operating Density”
2. Solution 2042660 How to define profiles for pressure boundary conditions

Keywords: Buoyancy, Natural Convection, Operating Density, Gravity

© 2016 ANSYS, Inc. All rights reserved.