Setting Operating Density for Natural Convection cases when density is modelled using Ideal gas law

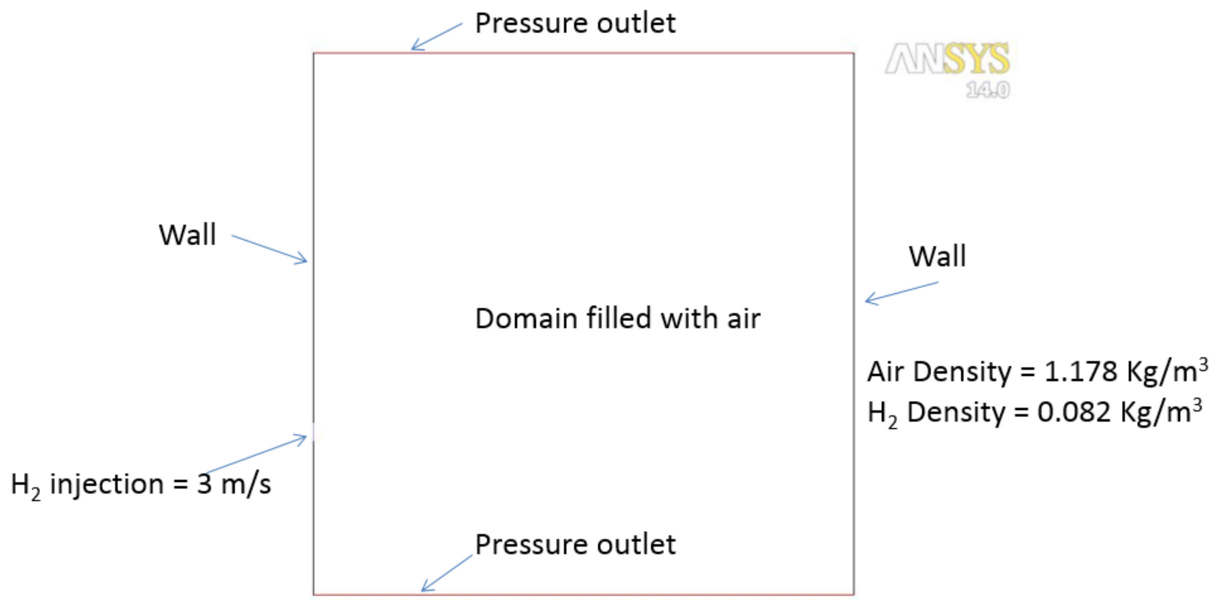
Problem/Description:

Setting Operating Density for Natural Convection cases when density is modelled using Ideal gas law

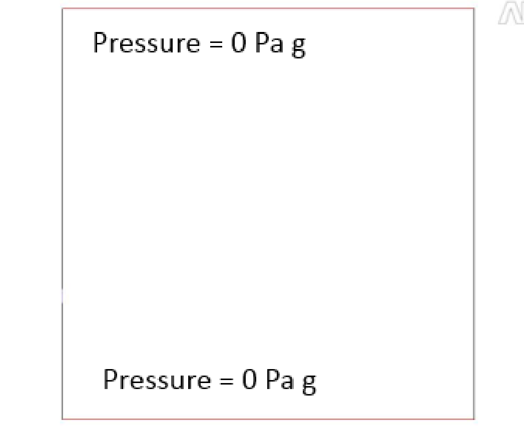
Solution:

In ANSYS Fluent, when density is modelled using ideal gas law, it is recommended to use an operating density value of zero. In such cases, users need to be careful in setting up boundary conditions for pressure outlets/inlets. In natural convection cases, where minor density differences (due to temperature variations) cause the flow, choosing the right operating density would be important for good convergence. This documents provides guidelines in choosing the best operating density values for such cases using a multiple species example involving hydrogen and air whose densities differ significantly.

Hydrogen is released into open air and as hydrogen is lighter than air, it should rise up.

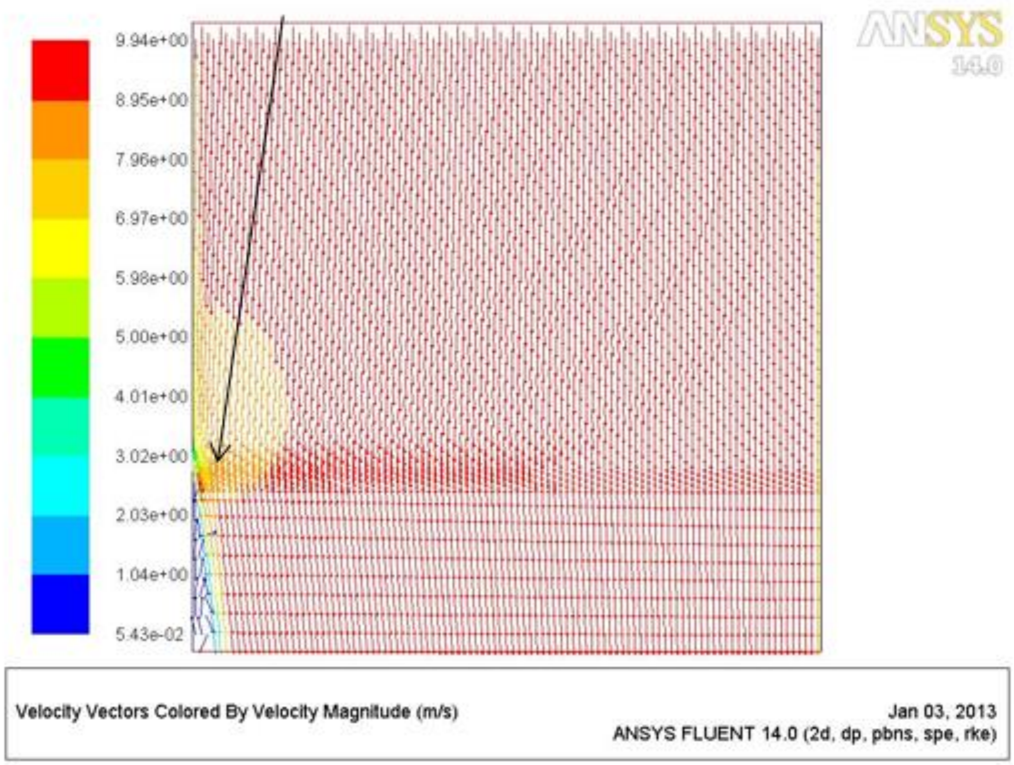


If we specify zero operating density and 0 Pa for top and bottom boundaries, Zero – Operating density



we end up getting wrong results as shown in the below figure

H_2 is found moving towards bottom, which is incorrect. As H2 is lighter, it is expected to rise



When the Boussinesq approximation is NOT used, the operating density rho_0 appears in the body-force term in the momentum equations as (rho-rho_0)*g . This form of the body-force term follows from the redefinition of pressure in ANSYS FLUENT as

p's = ps - rho_0*g*y (where y is the height)

If you are solving a natural-convection problem with a pressure boundary conditions, it is important to understand that the pressure you are specifying is p's in the above equation and not ps.

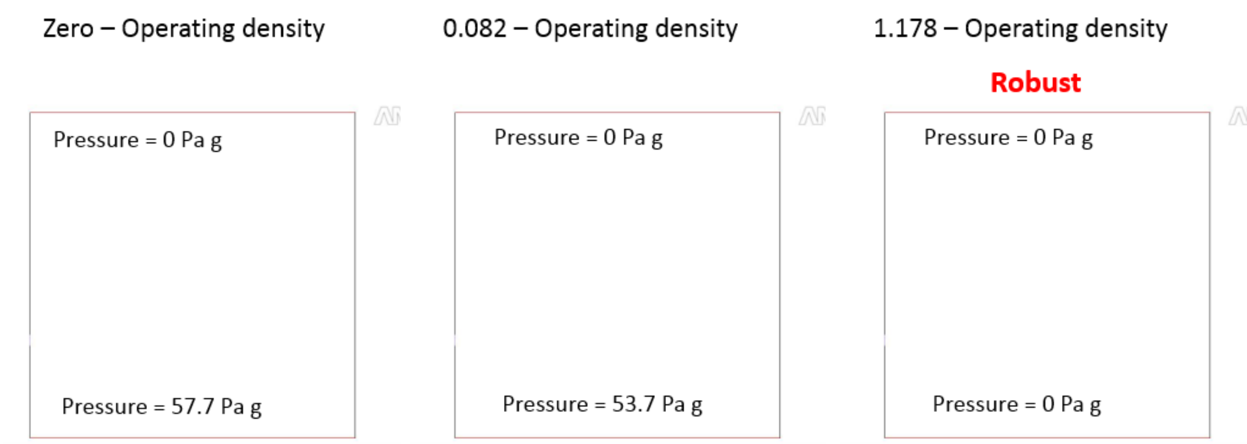
Now let us consider various scenarios:
1. If you are specifying zero as operating density (as recommended for ideal gas density cases by solver)
p's = ps

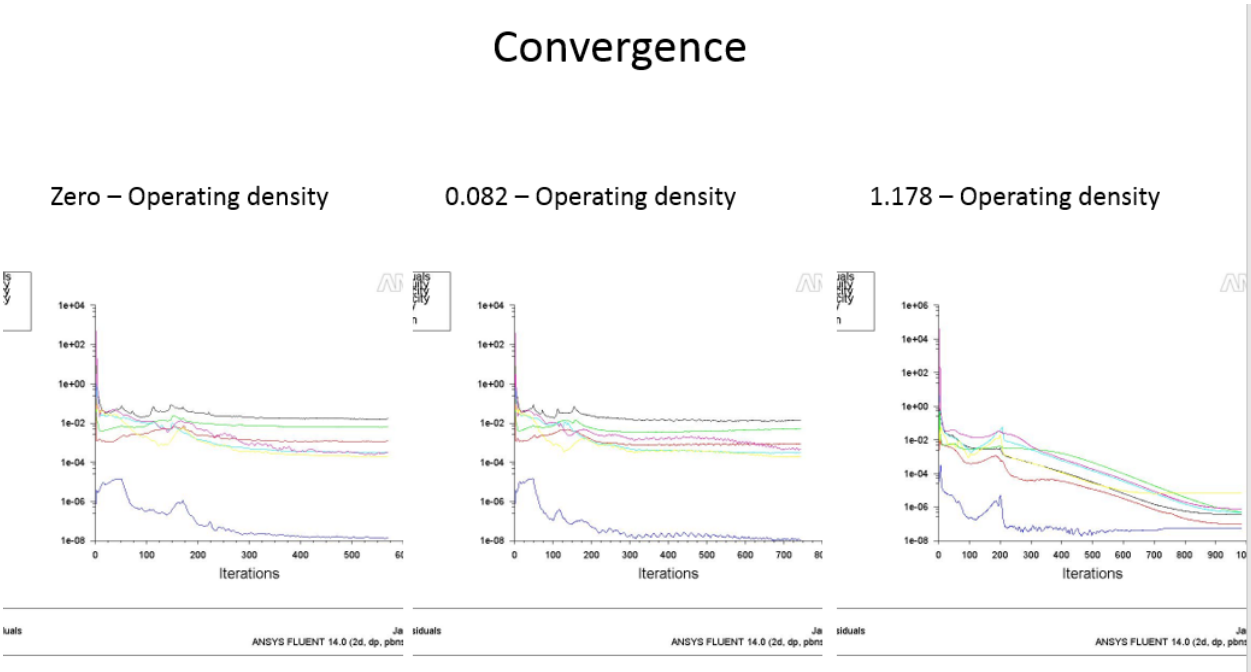
Hence we are directly dealing with static pressure. In this case we need to be careful with the boundary conditions. In your case the height of the domain is 5 m. Hence if the top surface is specified as 0 Pa gauge, the bottom surface has to be specified as = **1.178*9.81*5 = 57.7 Pa g** to establish the correct flow field.

2. In case if you specify hydrogen density (0.082) as the operating density, and for the top boundary 0 Pa, g. we need to specify a pressure boundary condition for the bottom boundary as below:
= (1.178-0.082) * 9.81*5 = 53.7 Pa g

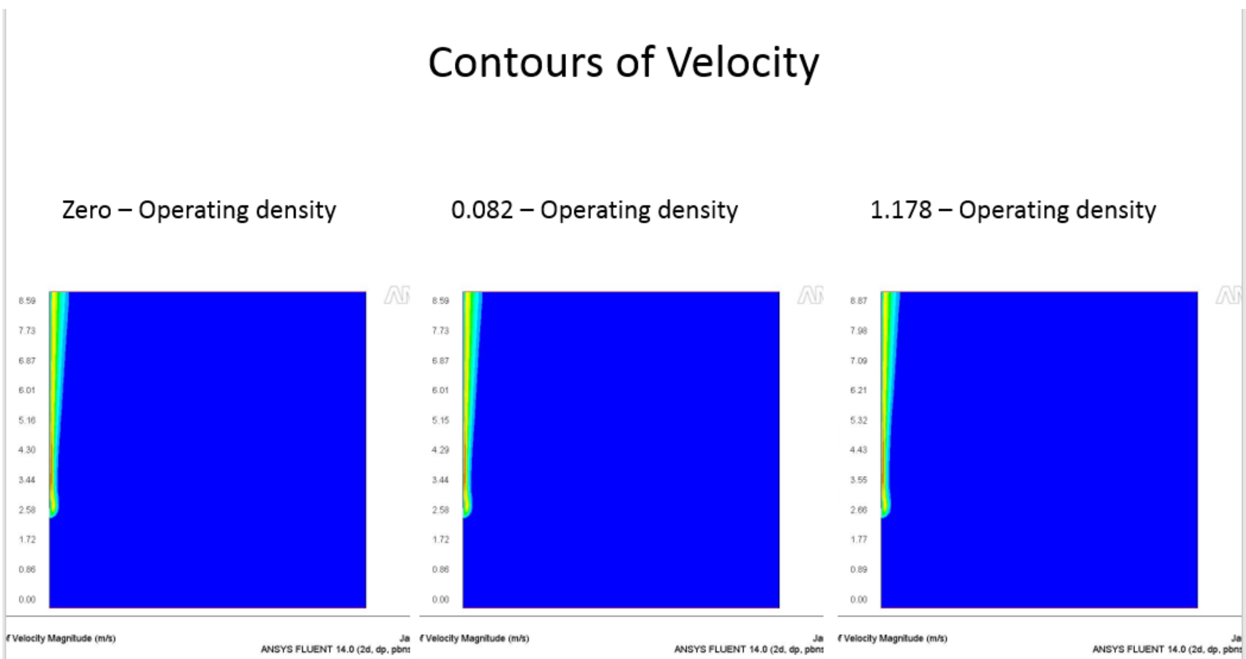
3. In case if you are specifying air density (1.178 Kg/m3) as operating density in the operating conditions panel, we can specify 0 Pa, g for both the top boundary as well as bottom boundary as in this case we are specifying p's. We need not bother about the hydrostatic head. It is taken care by the solver.

Pressure boundary conditions at top and bottom openings



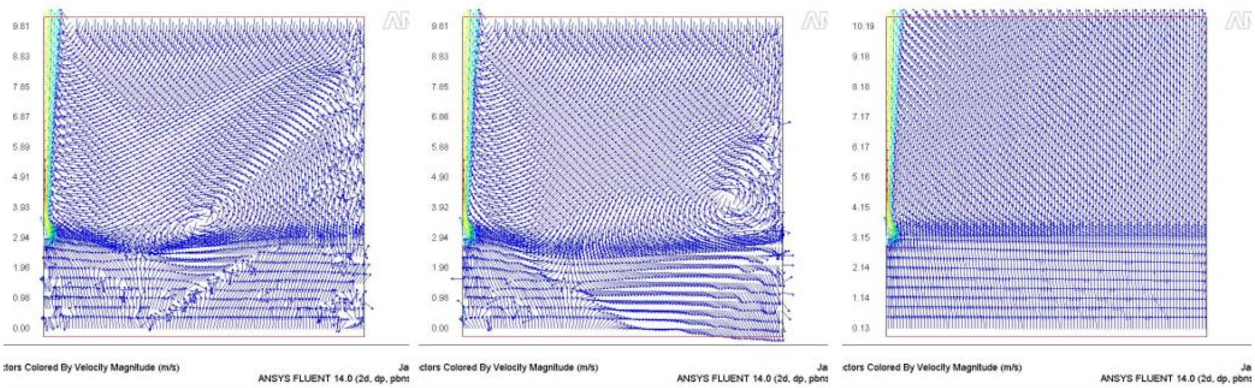


Residuals were not dipping down in the case of zero and 0.08 operating density. Hence the results will not be reliable.



Velocity Vectors

Zero – Operating density | 0.082 – Operating density | 1.178 – Operating density



Though in all the cases Hydrogen goes up, the air velocity vectors are not looking good for zero and 0.08 cases. Also the actual velocity values for air here are also very low. We should rely on 1.178 case, as it has proper convergence.

Summary:

In cases where operating density is zero, pressure values specified for the boundary conditions need to be correctly specified to match with reality

In natural convection cases where in ideal gas law is used for density, choosing higher operating density values aids in better convergence rather than specifying zero.