INFLUENCE BOUNDARY LAYER IN THE DIFFERENTIALLY PUMPED CHAMBER

Michal Bílek, Pavla Hlavatá
Doctoral Degree Programme (2),FEEC BUT
E-mail: xbilek07@stud.feec.vutbr.cz

Supervised by: Jiří Maxa
E-mail: maxa@feec.vutbr.cz

Abstract: The goal of this thesis is the analysis of a gas flowing through the drain channels within the apertures of differentially pumped chamber of the Environmental Scanning Electron Microscope (ESEM). This thesis contains a verification of current simulation results of gas flow within the differentially pumped chamber published by D. Danilatos using Monte Carlo method in comparison with simulation results achieved by using simulation program ANSYS Fluent, which uses the mechanism of continuum for its calculations and where it was effect of the boundary layer taken into account.

Keywords: ANYSY Fluent, Environmental Scanning Electron Microscope, ESEM, Monte Carlo, Boundary layer, Wall Functions

1 INTRODUCTION

This article is based on a study and published Dr. Danilatos [1] , which solved pumping differentially pumped chamber in the Environmental Scanning Electron Microscope (ESEM) using statistical Monte Carlo methods.

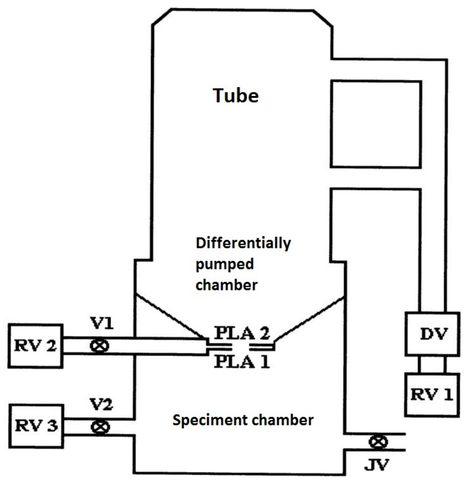

Figure 1: Scheme differentially pumped chamber

On the basic of this article was prepared a comparative study where were obtained comparable results of Monte Carlo method [2] and continuum mechanics of the Ansys Fluent program. In this article will be discuss the influence of the boundary layer. From the results published by Dr Danilatos it is evident that the density of computation mesh did not solve with the issue of the boundary layer Fig.2.

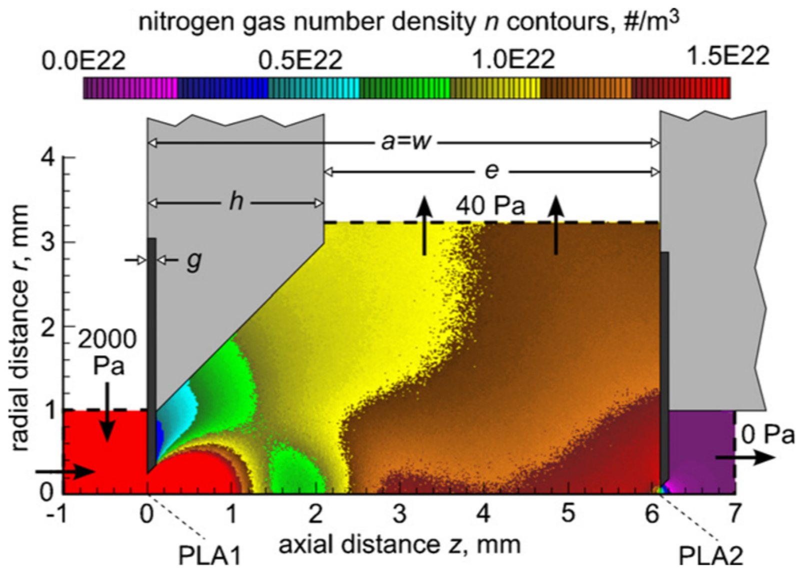

Figure 2: Results published Dr. Danilatos - Distribution of Normalized Number Density [1]

The graphical representation of the distribution of density-normalized number is apparent that the color scale shows a vertical transitions between color scale, which at Boundary layers is unlikely [4].

2 BOUNDARY LAYER

The boundary layer is formed near the wall. The reason for create a fine mesh on the surface of the boundary layer is the law which says the velocity of the fluid is zero. This velocity is non-linear increased with longer distance from the surface. Area cross gradient velocity is the boundary layer Fig.3.

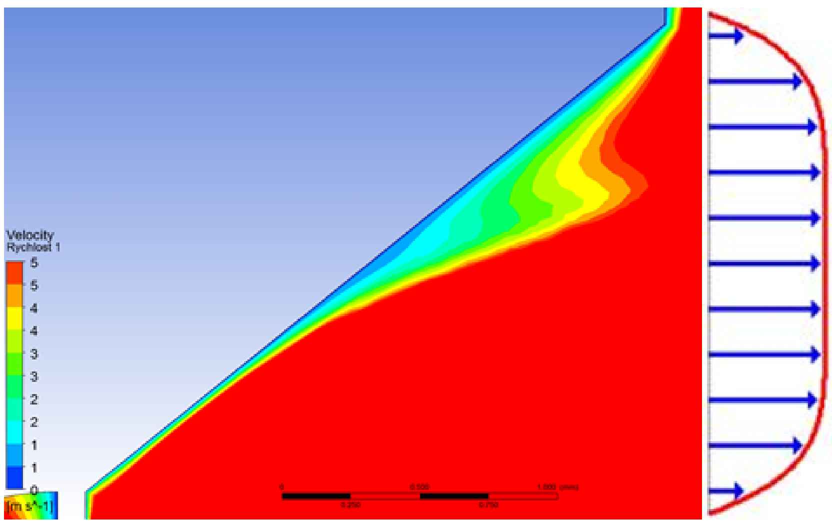

Figure 3: Velocity distribution

If flowing fluid contains a turbulent flow, there was developed appreciably whirls and fluid is mixed. Speed (i.e. both the magnitude and direction) of individual particles with flowing fluid varies irregularly and the fluid is no longer stationary. Velocity profile is no longer stationary as in the laminar flow but speed almost in the whole inner part of the tube is approximately constant. With the exception of a thin layer near the wall in which steeply increases the size of speed depending on distance from the tube wall [3].

3 PROBLEMATICS Y^+

For a good description of the behavior of the flow in the boundary layer using system CFD is necessary use either solution using wall functions or creating precise computational mesh. The correctness of the solution is possible to evaluate by the value of y^+

During implementation, some of the wall function is necessary have function value y^+ in range 30 until 300 for control calculations. In our case, boundary layer is solved by using refinement computational mesh to the wall without using a wall function. The value of y^+ must be 1 in this case.

With this method was calculated appropriate size of the first cell, before the creation of the computational mesh.

For the value of y^+ apply the formula:

$$y^+ = \frac{\rho v_\tau y}{\mu} \tag{1}$$

From this relation we obtain the formula for calculating the size of the first element in the boundary layer:

$$y = \frac{y^+ \mu}{v_\tau \rho} = 2.2e^{-07} \tag{2}$$

Where v_τ:

$$v_\tau = \sqrt{\frac{\tau_w}{\rho}} = 30.8\ [\frac{m}{s}] \tag{3}$$

The wall shear stress can be found from the skin friction coefficient C_f

$$\tau_w = \frac{1}{2} C_f \rho v_m^2 = 1189.9[Pa] \tag{4}$$

It follows that the velocity the middle gas stream in pumped chamber $v_m = 650\ m/s$.

A literature search suggest a formula for internal flows, with Reynolds number based on the pipe diameter is $C_f = 0.079\ R_e^{-0.25}$.

Reynolds numbers for the case involves the relationship:

$$R_e = \frac{\rho v_s D}{\mu} = 94476.7 \tag{5}$$

Where:
Density for Nitrogen $\rho = 1.25\ [kg/m^3]$

Dynamic viscosity $\mu = 1.72e^{-05}\ [kg/ms]$

Hydraulic Diameter $D = 4\ [mm]$

Medium Velocity $v_s = 325\ [m/s]$

4 MESH

According to the previous relationships, in our case at the speed middle stream is based on the size of the first element to the wall 0.0084 *mm*. The boundary layer was modeled in 12 of layers with the choice of Growt rate 1.2 thus enlarging each additional layer of 20% Fig.4.

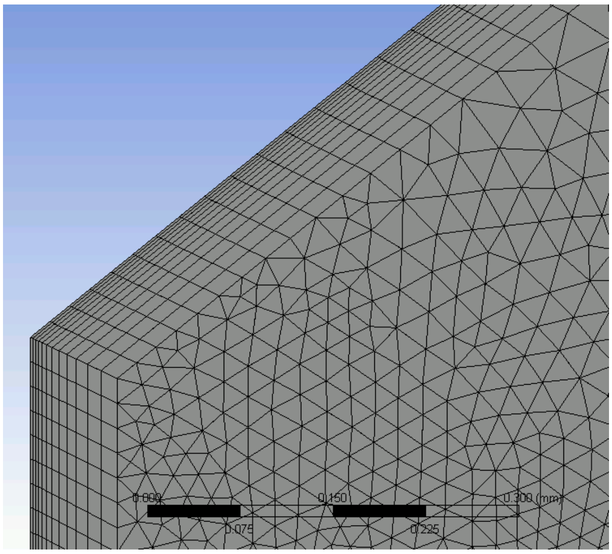

Figure 4: Mesh with a detail at the boundary layer

Total meshing differentially pumped chamber is shown in Figure 5. Verification results values of y^+ after calculation demonstrate the value to 1, therefore corresponding to the conditions for boundary layer.

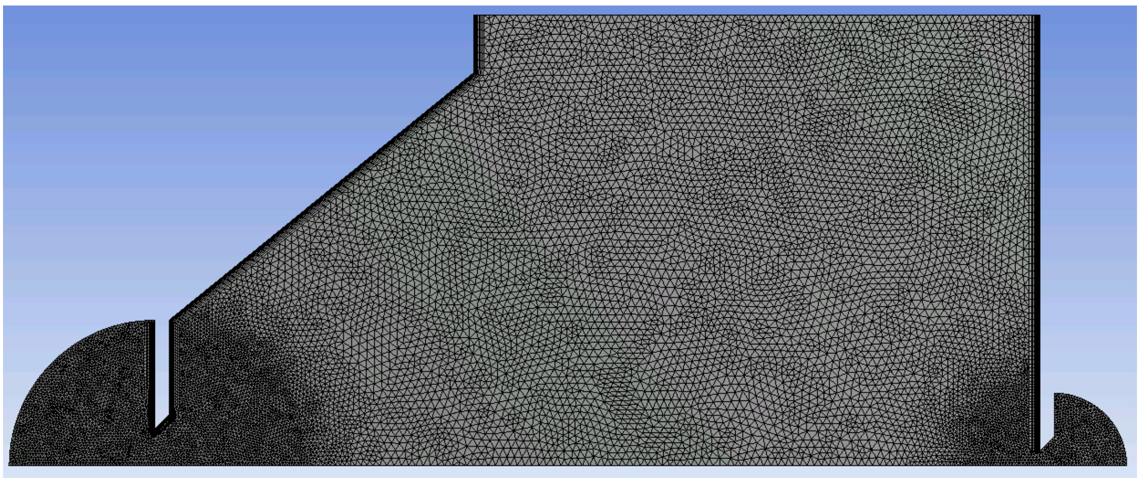

Figure 5: Total meshing differentially pumped chambers

5 RESULTS

In case that modeled boundary layer really leads to produce significantly different results. Figure 6 shows deceleration on the wall of the differentially pumped chamber in relation compared to the results on Figure 2 where is not braked. This has the effect influencing results also in the free stream, but primarily for intended measuring the pressure on the surface of the experimental chamber.

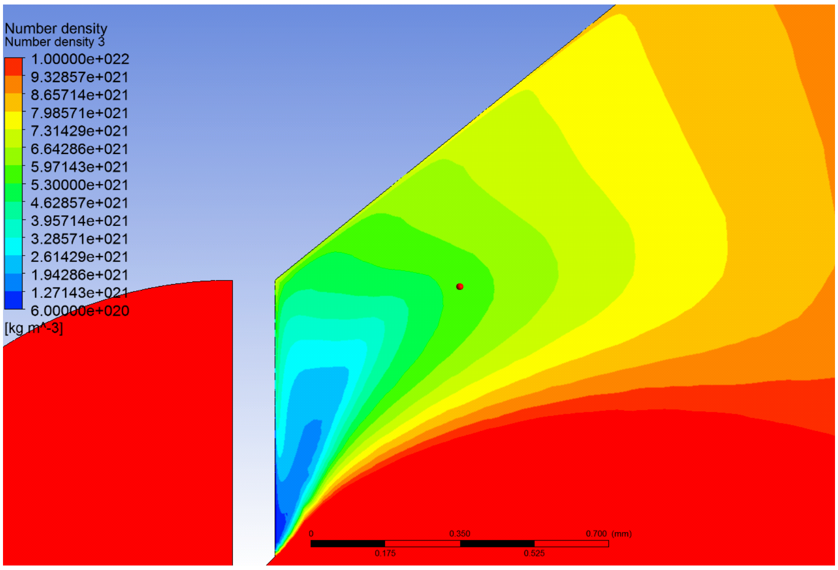

Figure 6: Deceleration on the wall of the differentially pumped chamber

6 CONCLUSIONS

There was done analysis of the effect neglecting the calculation of the boundary layer. An analysis was made of the gas flow in the differentially pumped chamber with modeled boundary layer and evaluating the values of y^+. The results confirmed the need for modeling appropriate mesh to evaluate the boundary layer which has an impact on the results in a free stream but mainly are key upcoming experiment.

ACKNOWLEDGEMENT

This research work has been carried out in the Centre for Research and Utilization of Renewable Energy (CVVOZE). Authors gratefully acknowledge financial support from the Ministry of Education, Youth and Sports of the Czech Republic under NPU I programme (project No. LO1210) and BUT specific research programme (project No. FEKT-S-17-4595).

REFERENCES

[1] DANILATOS, GD. *Velocity and ejector-jet assisted differential pumping: Novel design stages for environmental SEM.* Micron, 2012, vol. 43, no. 5, p. 600-611.

[2] BÍLEK, M.; HLAVATÁ, P.; MAXA, J. *Comparison of Results obtained using Monte Carlo and ANSYS Fluent in Analysis Differentially Pumped Chamber.* In Proceedings of the 22 Conference STUDENT EEICT 2016. 2016. s. 600-604. ISBN: 978-80-214-5350- 0.

[3] URUBA, Václav. *Turbulence. 2.*, revised edition In prague: Czech Technical University in Prague, 2014. ISBN 978-80-01-05600-4.

[4] NEDĚLA, V., KONVALINA, I., LENCOVÁ, B. and ZLÁMAL, J. *Comparison of calculated, simulated and measured signal amplification in variable pressure SEM.* Nuclear Instruments and Methods in Physics Research Section A, 2011, vol. 645, no. 1, p. 79-83.