Molecular Flow Through a Microcapillary

Introduction

Computing molecular flows in arbitrary geometries produces complex integral equations which are very difficult to compute analytically. Analytic solutions are therefore only available for simple geometries. One of the earliest problems solved was that of gas flow through tubes of arbitrary length, which was first treated correctly by Clausing (Ref. 1). Later the integral expressions he derived were computed more accurately by Cole (Ref. 2). These authors derived values for the transmission probability of molecules incident on a tube of arbitrary length, which is independent of the pressure, provided that the Knudsen number is much greater than one (that is in the molecular flow regime).

This model uses COMSOL to compute the transmission probability for molecular flow of molecules through a microcapillary with different ratios of the length to the radius. The results are compared with the accurate solution due to Cole (Ref. 2).

Model Definition

Nitrogen gas flows through a microcapillary tube of diameter 200 μm, from a reservoir at a fixed pressure of 10^{-3} mBar to a reservoir at high vacuum. The flux of molecules entering the tube from the high vacuum end is assumed to be negligible. At this pressure the mean free path of the nitrogen is approximately 100 mm, so the Knudsen number of the flow is much greater than 1. The ratio of the flux of molecules entering the tube to the flux of molecules leaving the tube is computed for tube lengths of 0.4 mm, 0.8 mm, 1.2 mm, 1.6 mm and 2.0 mm. The model is axially symmetric and is built in the 2D axially symmetric interface. The geometry is shown in Figure 1 for the case of the 2.0 mm long tube. The lower boundary is connected to the reservoir at low vacuum, the upper boundary is connected to the reservoir at high vacuum and the non-axial vertical boundary is the capillary wall.

©2013 COMSOL

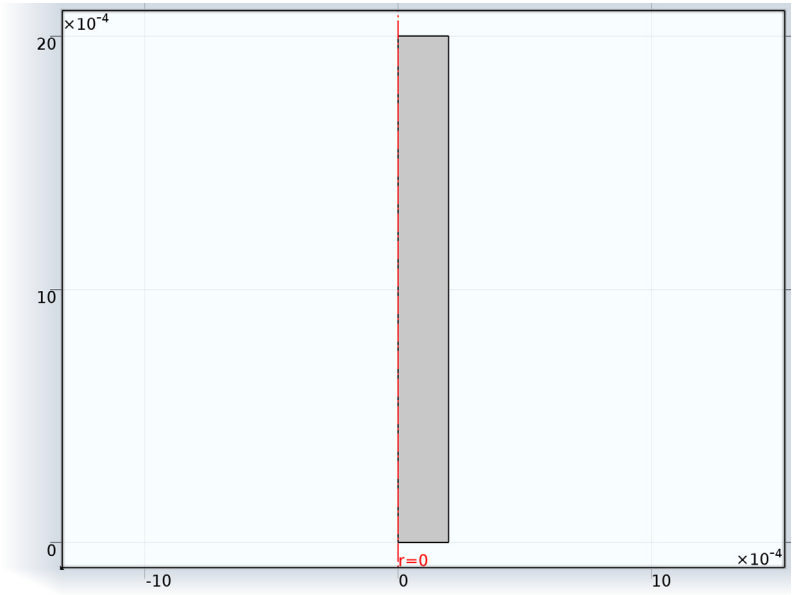


Figure 1: Model geometry.

The probability, χ, of molecules that strike the tube passing through the tube can be computed as the following integral:

$$\chi = \frac{\int_{\mathrm{RS}} 2\pi r J dr}{\int_{\mathrm{HV}} 2\pi r G dr} \tag{1}$$

where J is the outgoing flux, G is the incident flux, r is the radial distance, RS is the boundary adjacent to the reservoir, and HV is the boundary adjacent to high vacuum.

Results and Discussion

The incident flux on the surfaces of the tube is shown in Figure 2 for the case of the 2 mm long tube. A line plot of the pressure along the wall is shown in Figure 3. As expected, the incident flux decreases steadily from the reservoir end to the end at high vacuum, as does the pressure. Note that the incident flux (in contrast with the emitted flux) is not constant along the two reservoir boundaries, as it results from reflections

 ©2013 COMSOL

within the geometry. The transmission probability for the tube is plotted against length to radius ratio in Figure 4. Good agreement is obtained between the calculated results and the probabilities calculated by Cole (Ref. 2).

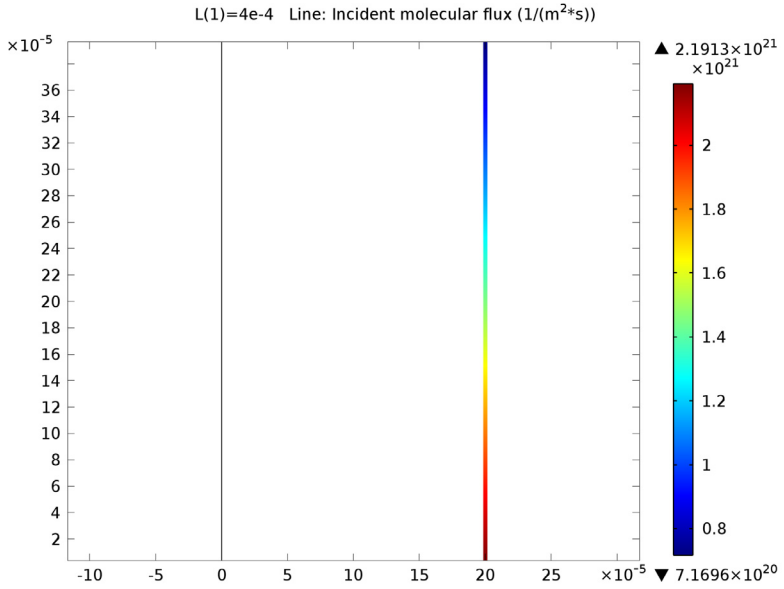


Figure 2: The incident flux inside the tube.

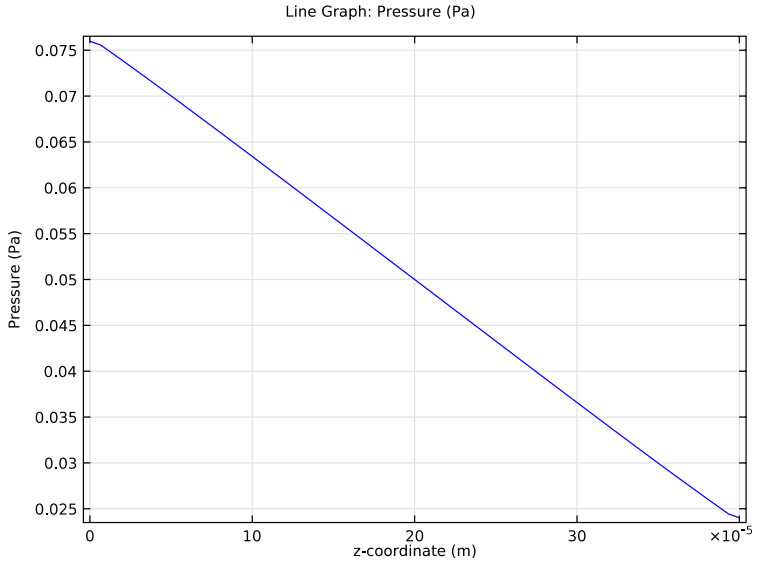


Figure 3: The pressure along the tube wall.

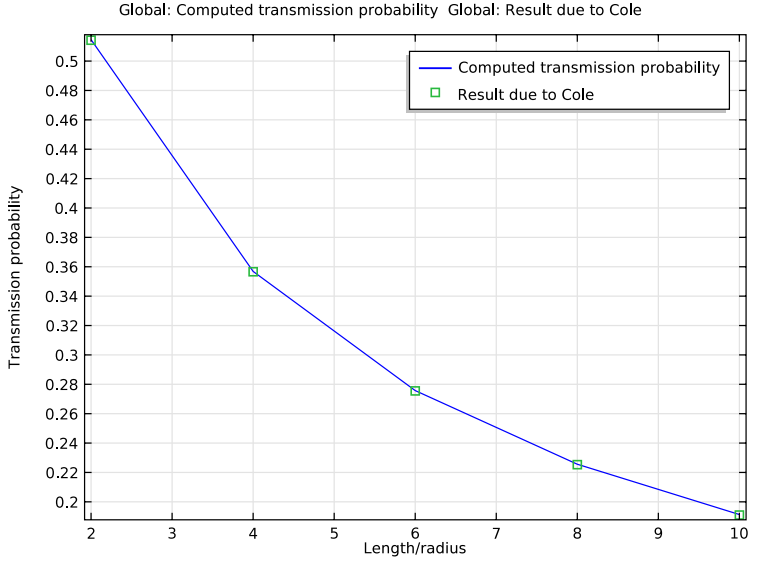


Figure 4: The transmission probability for the pipe as a function of the length to radius ratio. The results due to Cole (Ref. 2) are shown for comparison.

©2013 COMSOL

References

1. P. Clausing, "Über die Strömung sehr verdünnter Gase durch Röhren von beliebiger Länge," *Ann. Physik*, vol. 404, no. 8, pp. 961–989, 1932. English Translation available as: "The Flow of Highly Rarefied Gases through tubes of Arbitrary Length," *Journal of Vacuum Science and Technology*, vol. 8. no. 5, pp. 636–646, 2009.

2. R.J. Cole, "Complementary Variational Principles for Knudsen Flow Rates," *J. Inst. Maths Applics*, vol. 20, pp. 107–115, 1977.

Notes About the COMSOL Implementation

The model is straightforward to set up using the Molecular Flow interface. The capillary walls are assigned the wall boundary condition, the reservoir boundary condition is used for the bottom opening and the top opening is assigned the total vacuum boundary condition. The geometry is parameterized and a parametric solver is used to vary the capillary dimensions.

Model Library path: `Molecular_Flow_Module/Benchmark_Models/vacuum_capillary`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Fluid Flow>Rarefied Flow>Free Molecular Flow (fmf)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

©2013 COMSOL

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
L	2[mm]	Capillary length
R	0.2[mm]	Capillary radius
p0	1e-3[mbar]	Reservoir pressure

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type R.

4 In the **Height** edit field, type L.

5 Click the **Build All** button.

FREE MOLECULAR FLOW

Adjust the integration resolution setting to improve the accuracy of the computed fluxes.

1 In the **Free Molecular Flow** settings window, locate the **Integration Settings** section.

2 From the **Integration resolution** list, choose **1024**.

Set boundary conditions for the tube entrance and exit.

3 In the **Model Builder** window, expand the **Free Molecular Flow** node.

Total Vacuum 1

1 Right-click **Model 1>Free Molecular Flow** and choose **Total Vacuum**.

2 Select Boundary 3 only.

Reservoir 1

1 In the **Model Builder** window, right-click **Free Molecular Flow** and choose **Reservoir**.

2 Select Boundary 2 only.

 ©2013 COMSOL

3 In the **Reservoir** settings window, locate the **Reservoir** section.

4 In the p_0 edit field, type `p0`.

DEFINITIONS

Define an interpolation function for Cole's solution for the transmission probability as a function of the length-to-radius ratio. You will use it later for comparison with the model result.

Interpolation 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Functions>Interpolation**.

2 In the **Interpolation** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

t	f(t)
2	0.5142
4	0.3566
6	0.2755
8	0.2253
10	0.1910

Set up integration operators to compute the total flux of molecules through the system.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 2 only.

Integration 2

1 Right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

MESH 1

Use a mapped Mesh for this simple geometry.

Mapped 1

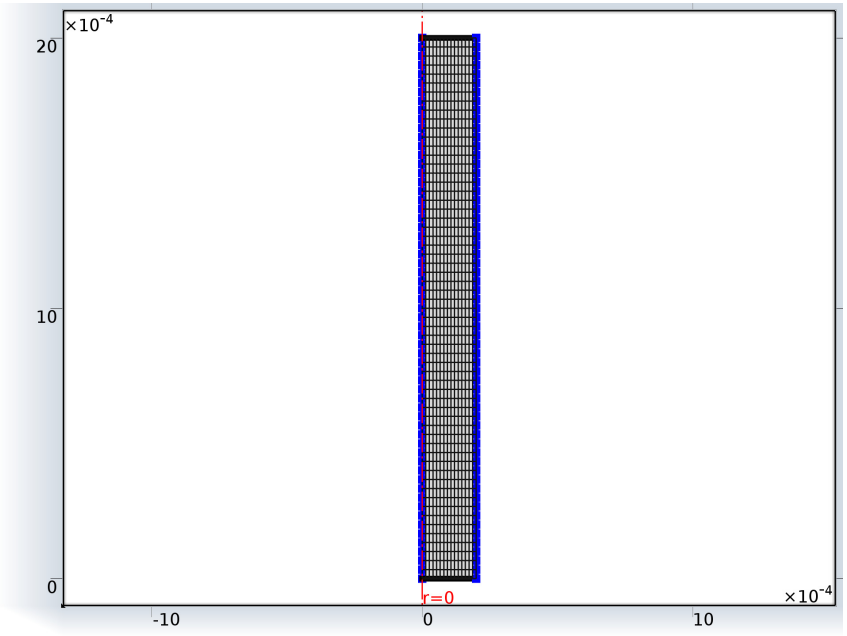
In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Mapped**.

Distribution 1

1. In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 2 and 3 only.
3. In the **Distribution** settings window, locate the **Distribution** section.
4. In the **Number of elements** edit field, type 60.

Distribution 2

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundaries 1 and 4 only.
3. In the **Distribution** settings window, locate the **Distribution** section.
4. In the **Number of elements** edit field, type 80.
5. Click the **Build All** button.



 ©2013 COMSOL

STUDY 1

Set up a parametric sweep over the capillary length.

In the **Model Builder** window, expand the **Study 1** node.

Parametric Sweep

1 Right-click **Study 1** and choose **Parametric Sweep**.

2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names
L

5 Click **Range**.

6 Go to the **Range** dialog box.

7 In the **Start** edit field, type `2[mm]`.

8 In the **Stop** edit field, type `0.4[mm]`.

9 In the **Step** edit field, type `-0.4[mm]`.

10 Click the **Add** button.

11 Right-click **Study 1** and choose **Compute**.

RESULTS

Incident molecular flux

1 Click the **Zoom Extents** button on the Graphics toolbar.

2 In the **Model Builder** window, under **Results** right-click **Incident molecular flux** and choose **Plot**.

The default plot shows the incident molecular flux inside the microcapillary tube. Compare with the plot in Figure 2.

Plot the pressure along the tube wall by following these steps.

1D Plot Group 4

1 Right-click **Results** and choose **1D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Line Graph**.

3 Select Boundary 4 only.

4 In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Free Molecular Flow>Pressure>Pressure (p)**.

Use the z-coordinate for the x axis data.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** edit field, type `z`.

7 Click the **Plot** button.

Compare the resulting plot with that in Figure 3.

Finally, plot the result for the transmission probability versus the length-to-radius ratio and compare with Cole's solution.

1D Plot Group 5

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>1D Plot Group 5** and choose **Global**.

5 In the **Global** settings window, locate the **y-Axis Data** section.

6 In the table, enter the following settings:

Expression	Unit	Description
`intop2(2*pi*r*G)/ intop1(2*pi*r*fmf.J)`		`Computed transmission probability`

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** edit field, type `L/R`.

9 In the **Model Builder** window, click **1D Plot Group 5**.

10 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

11 Select the **x-axis label** check box.

12 In the associated edit field, type `Length/radius`.

13 Select the **y-axis label** check box.

14 In the associated edit field, type `Transmission probability`.

15 Right-click **1D Plot Group 5** and choose **Global**.

16 In the **Global** settings window, locate the **y-Axis Data** section.

 ©2013 COMSOL

17 In the table, enter the following settings:

Expression	Unit	Description
int1(L/R)		Result due to Cole

18 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

19 In the **Expression** edit field, type `L/R`.

Change the plot style to display only the data points.

20 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

21 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

22 From the **Positioning** list, choose **In data points**.

23 Right-click **1D Plot Group 5** and choose **Plot**.

Compare with Figure 4.

©2013 COMSOL