Outgassing Pipes

Introduction

This benchmark model computes the pressure in a system of outgassing pipes with a high aspect ratio. The results are compared with a 1D simulation and a Monte-Carlo simulation of the same system from the literature.

Model Definition

The system consists of a long circular tube with a single change in cross section. A constant outgassing flux of 3×10^{-12} Torr-l/cm^2 is emitted from the walls of the pipes. Two pumps are attached to the system, one directly on the pipe and the other via an additional length of pipe. Both pumps operate at a pump speed of 30 l/s. The model geometry, along with the location of the two pumps, is shown in Figure 1.

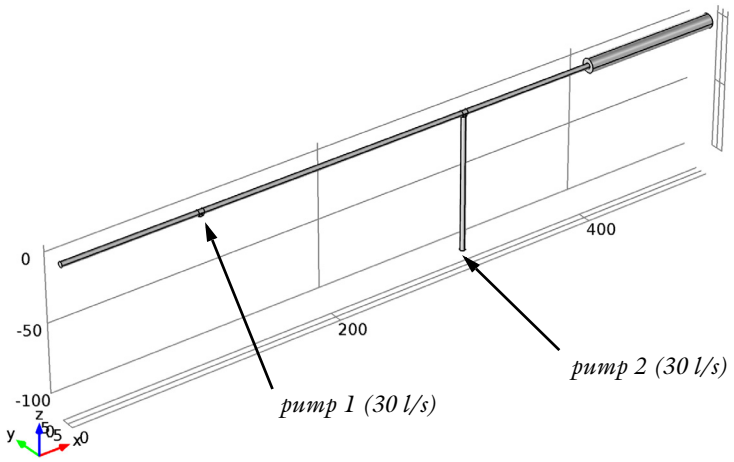


Figure 1: Model geometry. The location of the two pumps is indicated. All other surfaces outgass at a constant rate of 3×10^{-12} Torr-l/cm^2.

Results and Discussion

Figures 2, 3 and 4 show the molecular flux, number density and pressure respectively on the surfaces of the pipes.

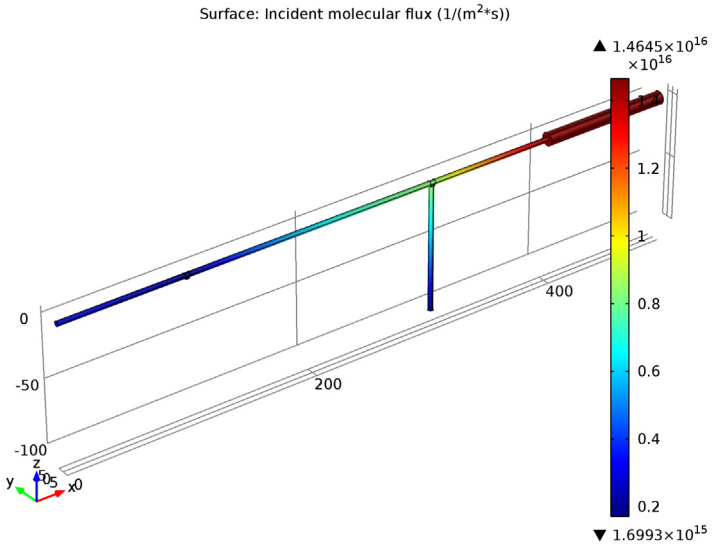


Figure 2: Molecular flux on the surface of the pipes.

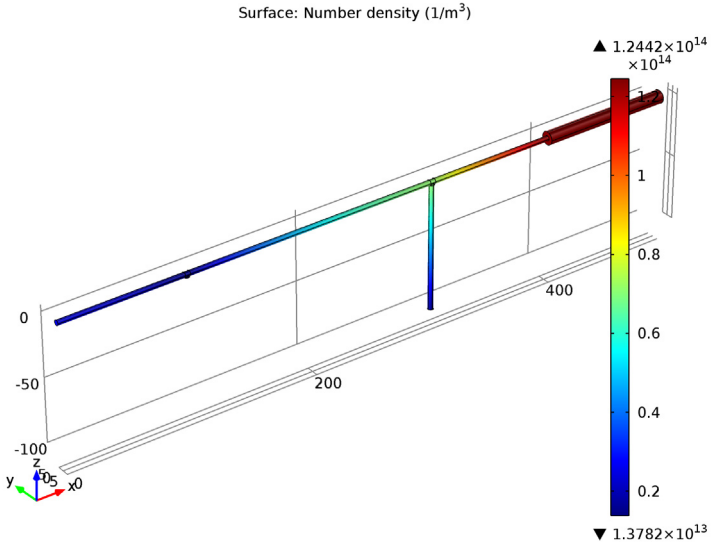


Figure 3: Number density on the pipe surfaces.

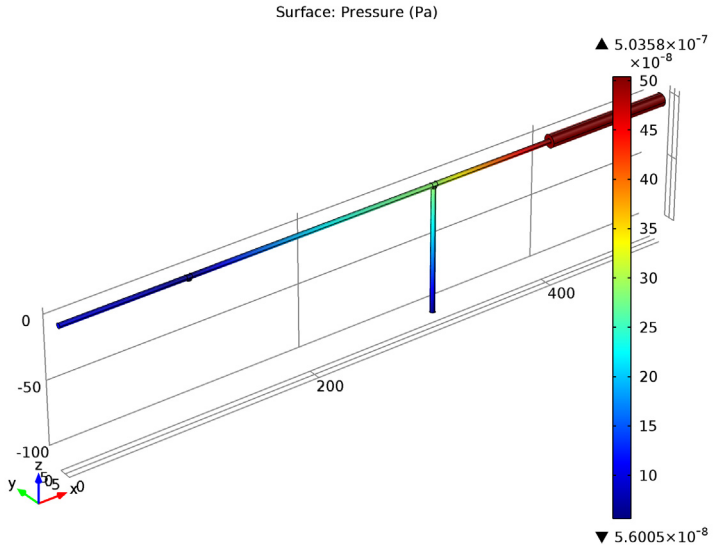


Figure 4: Pressure in the pipes.

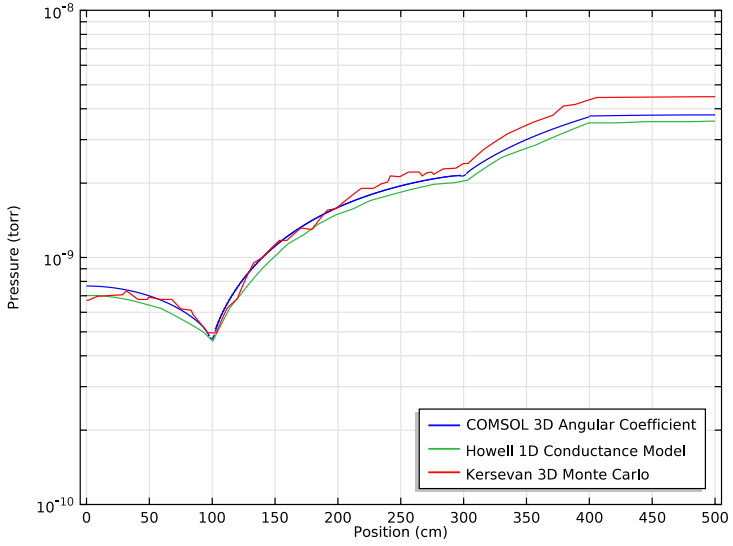


Figure 5: Pressure distribution along the top surface of the pipes. The results are compared with those from Ref. 1 *and* Ref. 2.

The pressure distribution in the pipes is in good agreement with the distributions given in Ref. 1 and Ref. 2.

Model Library path: `Molecular_Flow_Module/Benchmark_Models/outgassing_pipes`

References

1. J. Howell, B. Wherle and H. Jostlein, "Calculation of pressure distribution in vacuum systems using a commercial finite element program", Proceedings of the 1991 IEEE Particle Accelerator Conference (APS Beam Physics). vol. 4, pp.2295-2297, 1991.

2. R. Kersevan, "Analytical and numerical tools for vacuum simulation", CERN Accelerator School, Silken Park Hotel San Jorge, Platija d'Aro, Spain, 16-24 May 2006 (available at http://cas.web.cern.ch/cas/Spain-2006/PDFs/Kersevan.pdf).

Modeling Instructions

Note: This model may require up to 4.5 GB of RAM to solve.

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click **Next**.
3. In the **Add physics** tree, select **Fluid Flow>Rarefied Flow>Free Molecular Flow (fmf)**.
4. Click **Add Selected**.
5. Click **Next**.
6. Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
7. Click **Finish**.

GLOBAL DEFINITIONS

Define model parameters.

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
T0	293.15[K]	Temperature
Mn0	0.028[kg/mol]	Molar mass
ps	30[l/s]	Pump speed
tdr	3e-12[(torr*l)/cm^2/s]	Thermal desorption rate
Mf	Mn0*tdr/(R_const*T0)	Mass flux

GEOMETRY 1

Define the geometry.

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

Cylinder 1

1 Right-click **Model 1>Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type 400.

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 2

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

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 5.

4 In the **Height** edit field, type 100.

5 Locate the **Position** section. In the **x** edit field, type 400.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 3

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type 2.

5 Locate the **Position** section. In the **x** edit field, type 100.

6 In the **z** edit field, type -2.

Cylinder 4

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type 100.

5 Locate the **Position** section. In the **x** edit field, type 300.

6 In the **z** edit field, type -100.

Cylinder 5

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

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

5 Locate the **Position** section. In the **x** edit field, type 98.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

7 Click the **Build Selected** button.

Cylinder 6

1 Right-click **Model 1>Geometry 1>Cylinder 5** and choose **Duplicate**.

2 In the **Cylinder** settings window, locate the **Position** section.

3 In the **x** edit field, type 298.

4 Click the **Build Selected** button.

Cylinder 7

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

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 2.

4 In the **Height** edit field, type 2.

5 Locate the **Position** section. In the **x** edit field, type 300.

6 In the **z** edit field, type -2.

7 Click the **Build Selected** button.

Union 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

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

3 Clear the **Keep interior boundaries** check box.

4 Click the **Select Box** button on the Graphics toolbar.

5 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

6 Click the **Build Selected** button.

DEFINITIONS

Add interpolation functions for benchmark comparisons.

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 **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `outgassing_pipes_howell.txt`.

Interpolation 2

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

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

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `outgassing_pipes_kersevan.txt`.

FREE MOLECULAR FLOW

Set up the physics and boundary conditions.

Molecular Flow 1

1 In the **Model Builder** window, under **Model 1>Free Molecular Flow** click **Molecular Flow 1**.

2 In the **Molecular Flow** settings window, locate the **Molecular Flow** section.

3 In the M_n edit field, type `Mn0`.

Surface Temperature 1

1 In the **Model Builder** window, under **Model 1>Free Molecular Flow** click **Surface Temperature 1**.

2 In the **Surface Temperature** settings window, locate the **Surface Temperature** section.

3 In the T edit field, type `T0`.

Wall 1

1 In the **Model Builder** window, under **Model 1>Free Molecular Flow** click **Wall 1**.

2 In the **Wall** settings window, locate the **Wall Type** section.

3 From the **Wall type** list, choose **Outgassing wall**.

4 Locate the **Flux** section. From the **Outgoing flux** list, choose **Mass flux**.

5 In the M_f edit field, type `Mf`.

Vacuum Pump 1

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

2 Select Boundaries 10 and 12 only.

3 In the **Vacuum Pump** settings window, locate the **Vacuum Pump** section.

4 From the **Specify pump flux** list, choose **Pump speed**.

5 In the S edit field, type `ps`.

Vacuum Pump 2

1 In the **Model Builder** window, right-click **Vacuum Pump 1** and choose **Duplicate**.

2 In the **Vacuum Pump** settings window, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 26 only.

MESH 1

Mesh the geometry.

Edge 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Edges 9, 10, 12, 13, 15, 18, 22, 26, 29, 30, 32, 34, 37, 38, 43, 44, 46, 48, 54, 60, 63, 64, 67, 69, 72, 73, 75–77, and 79–81 only.

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Extra fine**.

Mapped 1

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

2 Select Boundaries 2–5, 18–21, 24, 25, 31, 34, 37–40, and 42–45 only.

Distribution 1

1 Right-click **Model 1>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 3, 51, 65, and 74 only.

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

4 In the **Number of elements** edit field, type 80.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edge 31 only.

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

4 In the **Number of elements** edit field, type 160.

5 Click the **Build Selected** button.

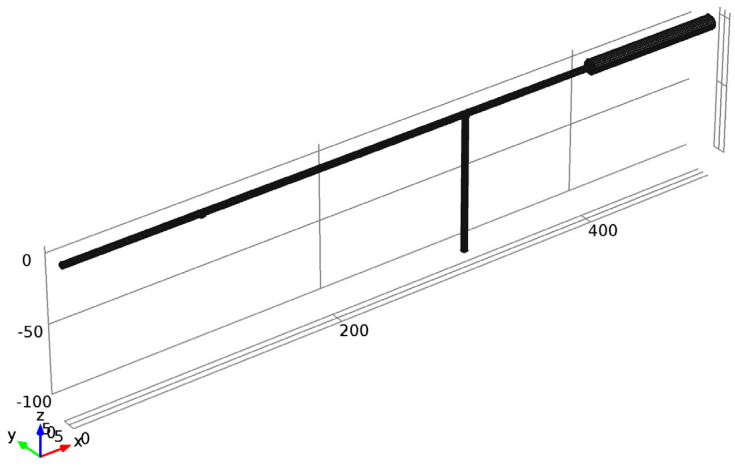
Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Free Triangular** settings window, locate the **Boundary Selection** section.

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

4 Click the **Build All** button.



STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Pressure (fmf)

Plot the pressure profile.

1D Plot Group 4

1. In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2. In the **Model Builder** window, click **1D Plot Group 4**.
3. In the **1D Plot Group** settings window, click to expand the **Title** section.
4. From the **Title type** list, choose **None**.
5. Locate the **Plot Settings** section. Select the **x-axis label** check box.
6. In the associated edit field, type `Position (cm)`.
7. Select the **y-axis label** check box.
8. In the associated edit field, type `Pressure (torr)`.
9. Click to expand the **Axis** section. Right-click **1D Plot Group 4** and choose **Line Graph**.

10 In the **Line Graph** settings window, locate the **y-Axis Data** section.

11 In the **Expression** edit field, type `p`.

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

13 In the **Expression** edit field, type `x`.

14 From the **Unit** list, choose **cm**.

15 Locate the **y-Axis Data** section. From the **Unit** list, choose **Torr**.

16 Select Edges 7, 19, 33, 35, 49, 70, and 82 only.

17 Locate the **Legends** section. Select the **Show legends** check box.

18 From the **Legends** list, choose **Manual**.

19 In the table, enter the following settings:

Legends
`COMSOL 3D Angular Coefficient`

20 Right-click **Results>1D Plot Group 4>Line Graph 1** and choose **Duplicate**.

21 In the **Line Graph** settings window, locate the **y-Axis Data** section.

22 In the **Expression** edit field, type `int1(x/1[cm])`.

23 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Howell 1D Conductance Model`

24 Right-click **Results>1D Plot Group 4>Line Graph 2** and choose **Duplicate**.

25 In the **Line Graph** settings window, locate the **y-Axis Data** section.

26 In the **Expression** edit field, type `int2(x/1[cm])`.

27 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Kersevan 3D Monte Carlo`

28 In the **Model Builder** window, click **1D Plot Group 4**.

29 In the **1D Plot Group** settings window, locate the **Legend** section.

30 From the **Position** list, choose **Lower right**.

31 Locate the **Axis** section. Select the **Manual axis limits** check box.

32 In the **y minimum** edit field, type `1e-10`.

33 In the **y maximum** edit field, type `1e-8`.

34 Select the **y-axis log scale** check box.

35 Click the **Plot** button.

36 Right-click **1D Plot Group 4** and choose **Rename**.

37 Go to the **Rename 1D Plot Group** dialog box and type `Pressure profile` in the **New name** edit field.

38 Click **OK**.