Created in COMSOL Multiphysics 6.2

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.

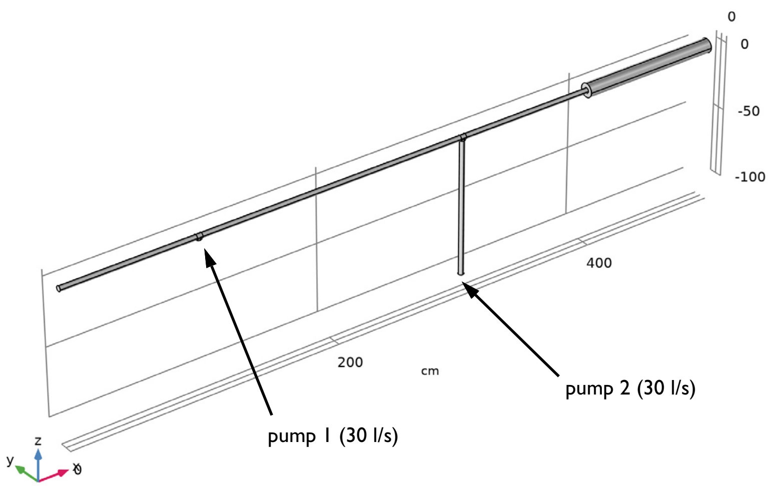

Figure 1: Model geometry. The location of the two pumps is indicated. All other surfaces outgas at a constant rate of 3×10^{-12} Torr-l/cm^2.

Results and Discussion

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

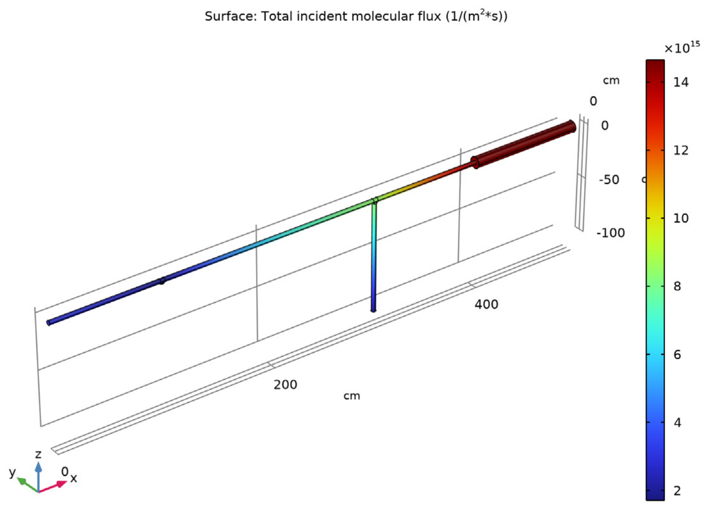

Figure 2: Molecular flux on the surface of the pipes.

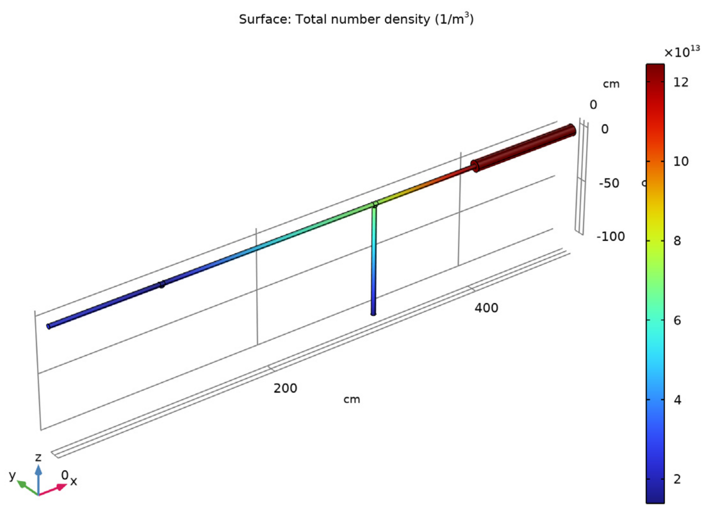

Figure 3: Number density on the pipe surfaces.

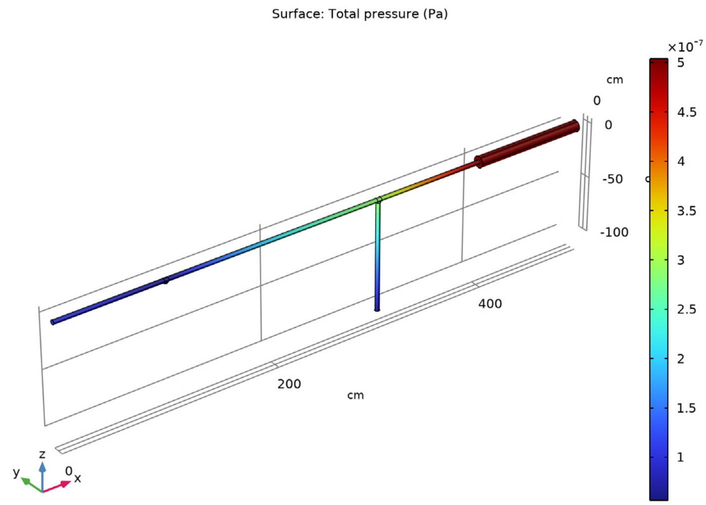

Figure 4: Pressure in the pipes.

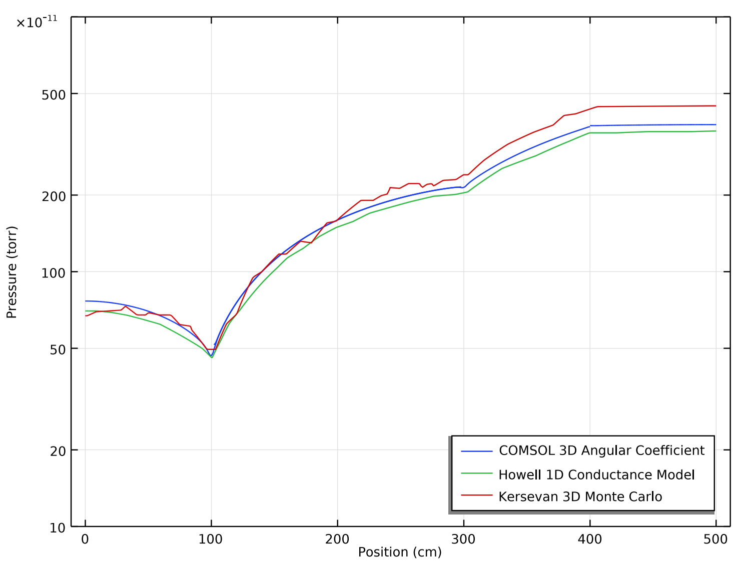

Figure 5: Pressure distribution along the top surface of the pipes. The results are compared with those from Ref. 1 *and* Ref. 2.

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

Application Library path: `Molecular_Flow_Module/Benchmarks/outgassing_pipes`

References

1. J. Howell, B. Wherle, and H. Jostlein, "Calculation of pressure distribution in vacuum systems using a commercial finite element program," *Proc. 991 IEEE Particle Accelerator Conference (APS Beam Physics)*, vol. 4, pp.2295–2297, 1991.

2. R. Kersevan, "Analytical and numerical tools for vacuum systems," *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

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow>Rarefied Flow>Free Molecular Flow (fmf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

Define model parameters.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`T0`	`293.15[K]`	293.15 K	`Temperature`
`Mn0`	`0.028[kg/mol]`	0.028 kg/mol	`Molar mass`
`ps`	`30[l/s]`	0.03 m^3/s	`Pump speed`

Define the geometry.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `2`.

4 In the **Height** text field, type `400`.

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

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `5`.

4 In the **Height** text field, type `100`.

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

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

Cylinder 3 (cyl3)

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `2`.

4 In the **Height** text field, type `2`.

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

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

Cylinder 4 (cyl4)

1 In the **Geometry** toolbar, click **Cylinder**.

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

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

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

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

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

Cylinder 5 (cyl5)

1 In the **Geometry** toolbar, click **Cylinder**.

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

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

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

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

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

7 Click **Build Selected**.

Cylinder 6 (cyl6)

1 Right-click **Cylinder 5 (cyl5)** and choose **Duplicate**.

2 In the **Settings** window for **Cylinder**, locate the **Position** section.

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

4 Click **Build Selected**.

Cylinder 7 (cyl7)

1 In the **Geometry** toolbar, click **Cylinder**.

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

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

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

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

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

7 Click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, locate the **Union** section.

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

4 Click in the **Graphics** window and then press Ctrl+A to select all objects.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Add interpolation functions for benchmark comparisons.

DEFINITIONS

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

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

Interpolation 2 (int2)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

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

Set up the physics and boundary conditions.

FREE MOLECULAR FLOW (FMF)

Molecular Flow 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Free Molecular Flow (fmf)** click **Molecular Flow 1**.

2 In the **Settings** window for **Molecular Flow**, locate the **Molecular Weight of Species** section.

3 In the $M_{n,G}$ text field, type `Mn0`.

Surface Temperature 1

1 In the **Model Builder** window, click **Surface Temperature 1**.

2 In the **Settings** window for **Surface Temperature**, locate the **Surface Temperature** section.

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

Wall 1

1 In the **Model Builder** window, click **Wall 1**.

2 In the **Settings** window for **Wall**, 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 **Thermal desorption rate**.

Vacuum Pump 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Vacuum Pump**.

2 Select Boundaries 10 and 12 only.

3 In the **Settings** window for **Vacuum Pump**, locate the **Vacuum Pump** section.

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

5 In the S_G text field, type `ps`.

Vacuum Pump 2

1 Right-click **Vacuum Pump 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 26 only.

Mesh the geometry.

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Boundary** and choose **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.

This rather long list includes all of the circular edges where different cylinders are connected in the geometry. A shortcut is to copy the selection list from these instructions and then click the **Paste selection** button. An alternative is to create an **Explicit** selection, where you can select the **Group by continuous tangent** check box to reduce the number of clicks required.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

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

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

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

This boundary selection comprises the sides of all of the long cylinders in the geometry. As for the edge selection above, an **Explicit** selection with the **Group by continuous tangent** check box can reduce the number of clicks required.

Distribution 1

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

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

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

Distribution 2

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

2 Select Edge 31 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

5 Click **Build Selected**.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

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

4 Click **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Total Pressure (fmf)

Plot the pressure profile.

1D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Position (cm)`.

6 Select the **y-axis label** check box. In the associated text field, type `Pressure (torr)`.

Line Graph 1

1 Right-click **1D Plot Group 4** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `fmf.ptot`.

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

5 In the **Expression** text field, type `x`.

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

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

8 Click to expand the **Legends** section. Select the **Show legends** check box.

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

10 In the table, enter the following settings:

Legends
`COMSOL 3D Angular Coefficient`

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

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

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

Legends
`Howell 1D Conductance Model`

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

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

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

Legends
`Kersevan 3D Monte Carlo`

Pressure Profile

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

2 In the **Settings** window for **1D Plot Group**, type `Pressure Profile` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

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

5 In the **y minimum** text field, type `1e-10`.

6 In the **y maximum** text field, type `1e-8`.

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

8 In the **Pressure Profile** toolbar, click **Plot**.

Compare the resulting plot with that in Figure 5.