Created in COMSOL Multiphysics 6.1

Turbomolecular Pump

Introduction

Turbomolecular pumps (TMPs) are widely used in vacuum system design. A TMP is a bladed molecular turbine that compresses gases by momentum transfer from the rapidly rotating blades of the rotor to the gas molecules. In the free molecular flow range (typically less than 10^{-1} Pa), the particles collide primarily with the rotor, rather than each other, resulting in an efficient pumping process. The following assumptions are made:

- Intermolecular collisions are negligible,
- The molecules follow a Maxwellian velocity distribution,
- The gas temperature is constant, and
- The particles experience diffuse scattering at the blade walls, following the distribution of velocity direction called *Lambert's cosine law* or *Knudsen's cosine law*.

This 3D model uses the Mathematical Particle Tracing interface to evaluate the performance of a simple turbomolecular pump in the free molecular flow regime. While the Free Molecular Flow interface provides an efficient and deterministic calculation of gas density in the free molecular flow regime, it uses the angular coefficient method which is only applicable for molecules that are moving much faster than any object in the geometry. For TMPs, where the blades move over similar time scales as the molecules themselves, this requirement is not satisfied, so a Monte Carlo approach is used instead.

The particle trajectories are computed in a rotating frame of reference, attached to the rotor, in which the fictitious centrifugal and Coriolis forces are automatically taken into account. The model shows the effect of the blade velocity ratio on the pumping characteristics, including the transmission probability, maximum compression ratio, and maximum speed factor.

Model Definition

A TMP may include a large number of stages, or rows of blades. For the TMP design considered in this example, each row of blades is arranged in a ring, and each blade is held at an angle. By rotating these rings of blades, a change in gas density in the axial direction is produced because the inclined blades are more likely to permit a gas molecule to cross the stage in one direction than in the opposite direction. The stage is called a rotor if it is allowed to rotate, or a stator if it is held in a fixed position. A single-stage rotor is illustrated in Figure 1; if the blades rotate counterclockwise as indicated by the red arrows, molecules are more likely to cross the stage going downward (as indicated by the blue arrows) than they are to go upward. Due to sector symmetry, the geometry can be simplified by considering only the gap between two blades, such as the region highlighted in Figure 2.

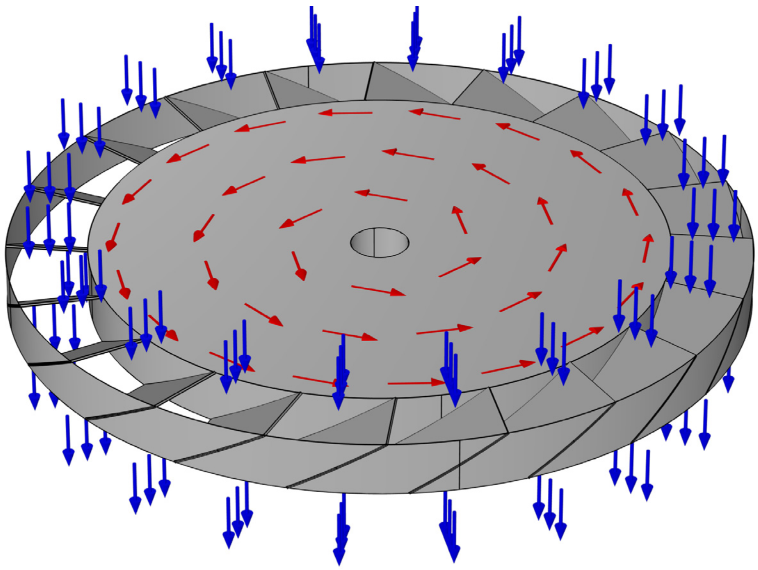

Figure 1: Full geometry of a turbomolecular pump stage.

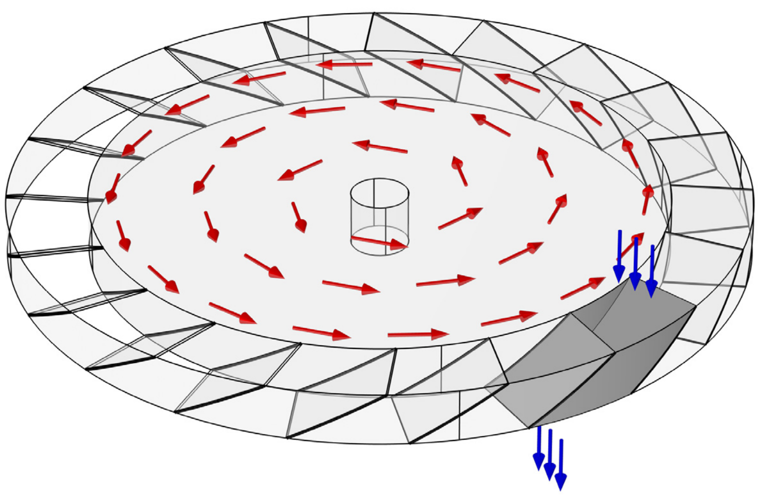

Figure 2: Geometry of a turbomolecular pump stage, where a single unit cell and the molecules entering and leaving it have been highlighted.

As in Ref. 1, the pump stage consists of 36 blades, each inclined at a 30 degree angle. The ratio of the root radius to the tip radius is 0.8. Note that if the root-to-tip ratio is very close to unity, the centrifugal and Coriolis forces can be neglected to some extent, allowing a less computationally intensive quasi-2D model to be used instead.

The relevant pump characteristics are based on the transmission probability of argon atoms from the inlet to the outlet M_{12} and the transmission probability from the outlet back to the inlet M_{21} (both dimensionless). In order to compute these transmission probabilities, the model uses two **Inlet** features to release particles on either side of the pump and two **Outlet** features to capture particles that reach the boundaries. The **Particle Counter** can be used to compute the number of particles from each **Inlet** that reach each **Outlet**.

Each **Inlet** releases argon atoms with a Lambertian distribution of initial directions and a Maxwellian distribution of initial speeds. The most probable speed of particles in the Maxwellian distribution v_p (SI unit: m/s) is

$$v_p = \frac{2k_B T}{m_p}$$

$$m_p = \frac{M}{N_A}$$

where

- $k_B = 1.380649 \times 10^{-23}$ J/K is the Boltzmann constant,
- $T = 300$ K is the ambient temperature,
- $M = 39.948$ g/mol is the molar mass of argon, and
- $N_A = 6.02214076 \times 10^{23}$ 1/mol is the Avogadro constant.

This gives a most probable speed of approximately $v_p = 353$ m/s.

The root wall, tip wall, and blade surfaces use the dedicated **Thermal Re-Emission** boundary condition, in which incident molecules are adsorbed onto the wall and then immediately released back into the modeling domain with a random velocity based on the surface temperature. The root wall and blades are stationary with respect to the rotating frame of reference, while the tip wall is stationary with respect to the inertial (laboratory) frame, and therefore moves relative to the rotating frame.

Modern turbomolecular pumps can operate at very high rotor speeds reaching 90,000 revolutions per minute. In this model, the angular velocity of the pump is specified in terms of the dimensionless blade velocity ratio C, defined as

$$C = \frac{v_b}{v_p} \qquad (1)$$

where v_b (SI unit: m/s) is the blade velocity, or the rms velocity magnitude of the rotating frame. The transmission probability is computed for different values of C from 0 to 4 using a **Parametric Sweep**; the maximum value $C = 4$ corresponds to a blade velocity of approximately $v_b = 1415$ m/s.

ABOUT PARTICLE TRACING IN ROTATING DOMAINS

To model the motion of the argon atoms in a rotating frame, a set of second-order differential equations for the components of the particle position $\mathbf{q}$ (SI unit: m) are solved:

$$\frac{d}{dt}\left(m_p \frac{d\mathbf{q}}{dt}\right) = \mathbf{F}_{cen} + \mathbf{F}_{cor} + \mathbf{F}_{eul}$$

where

- $\mathbf{F}_{cen}$ is the centrifugal force,

$$\mathbf{F}_{cen} = m_p \Omega \times (\mathbf{r} \times \Omega)$$

- $\mathbf{F}_{cor}$ is the Coriolis force,

$$\mathbf{F}_{cor} = 2 m_p \mathbf{v} \times \Omega$$

- $\mathbf{F}_{eul}$ is the Euler force,

$$\mathbf{F}_{eul} = m_p \mathbf{r} \times \dot{\Omega}$$

- Ω (SI unit: rad/s) is the angular velocity of the rotating frame, and
- $\mathbf{r}$ (SI unit: m) is the displacement from the center of rotation to the atom's position.

All of these fictitious forces are automatically defined by the **Rotating Frame** feature. In this example, the frame rotates at constant angular velocity, so the Euler force is zero.

Results and Discussion

The **Parametric Sweep** over the velocity ratio C begins at $C = 0$ and ends at $C = 4$ with intervals of 0.5. The final particle positions for six of these values are shown in Figure 3. The green particles begin at the top boundary while the red particles are released at the bottom boundary. As the velocity ratio increases, more green particles are able to cross from the top to the bottom, but fewer red particles can cross from the bottom to the top. Both effects cause the compression ratio of the turbopump stage to increase.

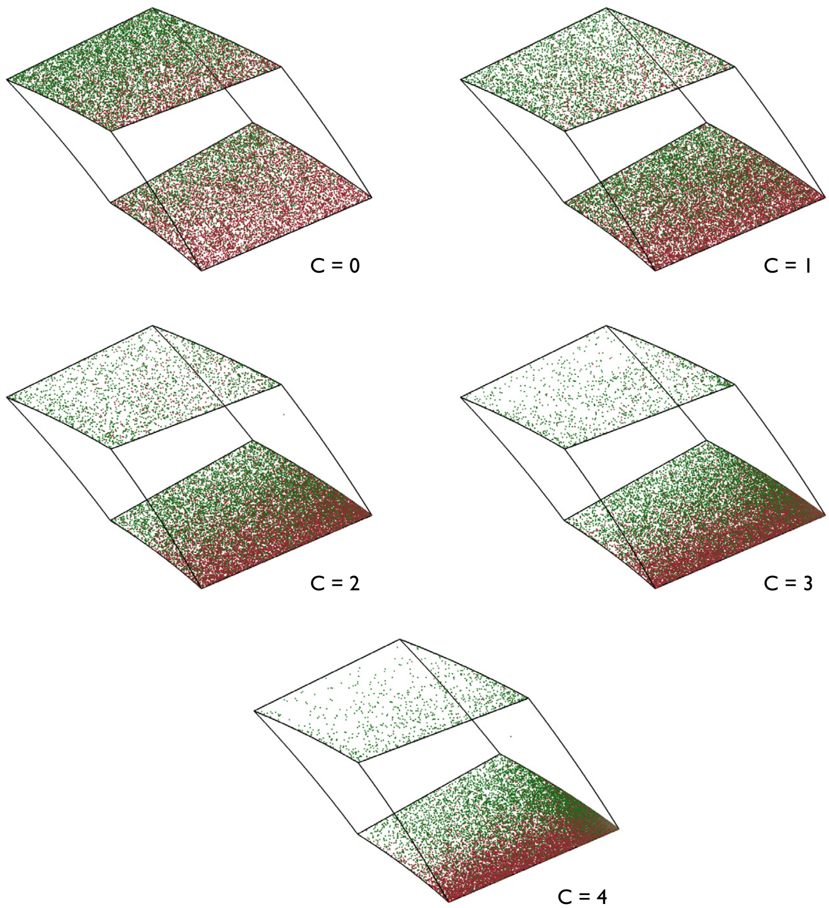

Figure 3: Final particle positions for different values of the blade velocity ratio C.

The transmission probabilities M_{12} and M_{21} for propagation in the forward and reverse directions are shown in Figure 4 and Figure 5, respectively. As expected, the transmission probabilities are almost equal when the blades are at rest ($C = 0$) because there is no distinction between the forward and reverse directions. As the blades begin to rotate faster, particles are more likely to be transmitted in the forward direction due to the momentum transferred to the argon atoms from the rotating walls.

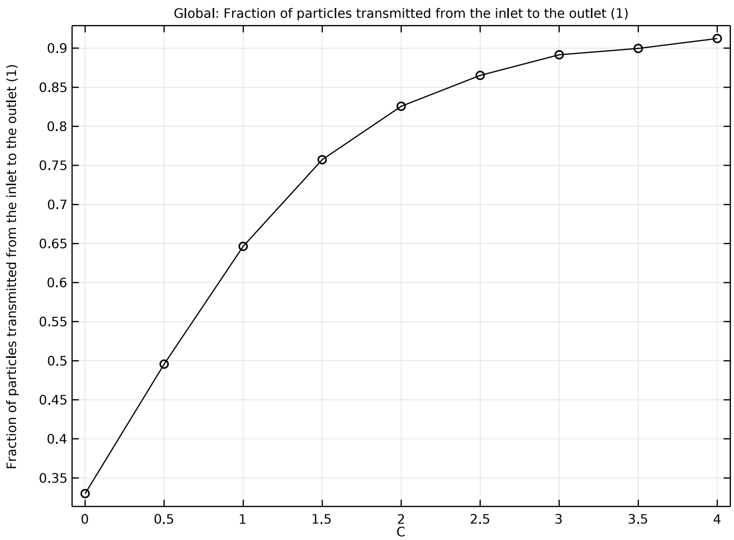

Figure 4: Fraction of particles transmitted from the inlet to the outlet M_{12}*, as a function of the blade velocity ratio* C*.*

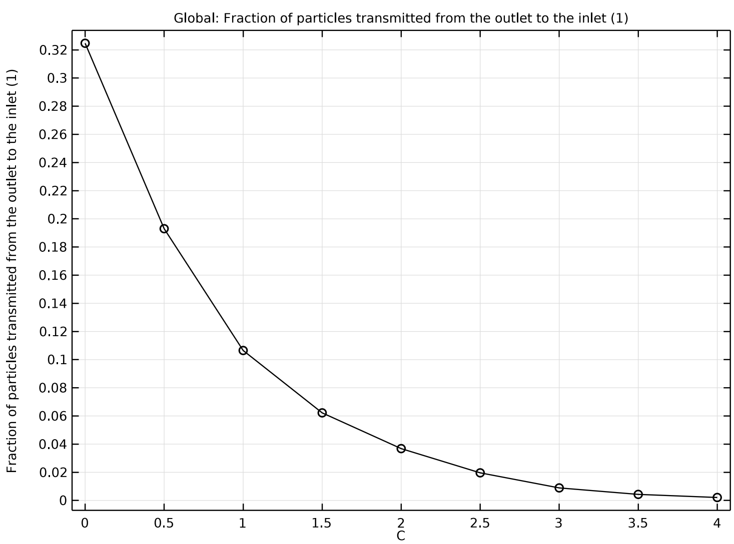

Figure 5: Fraction of particles transmitted from the outlet to the inlet M_{21}*, as a function of the blade velocity ratio* C*.*

Multiple-bladed structures with several disks provide adequate compression and speed to make a functional pump. The blades nearest the inlet are usually designed to have a high pumping speed and a low compression ratio. The blades nearest the outlet are designed to have a high compression ratio and a low pumping speed.

The maximum compression ratio K_{max} is defined by

$$K_{max} = \frac{M_{12}}{M_{21}} \tag{2}$$

and the maximum speed factor W_{max} by

$$W_{max} = M_{12} - M_{21} \tag{3}$$

Figure 6 and Figure 7 show the variation of these values as the blade velocity is increased. Compare these results to the corresponding plots in Ref. 1. The compression ratio is expected to increase monotonically. If C is further increased, some statistical noise is expected as the denominator M_{21} becomes very small. In such cases, consider increasing the number of model particles for a more statistically converged solution.

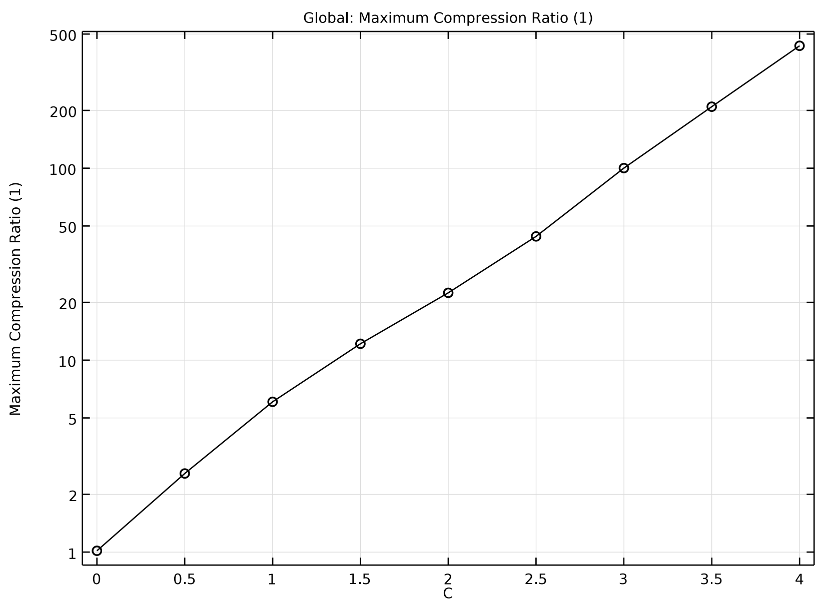

Figure 6: Maximum compression ratio K_{max}, as a function of the blade velocity ratio C.

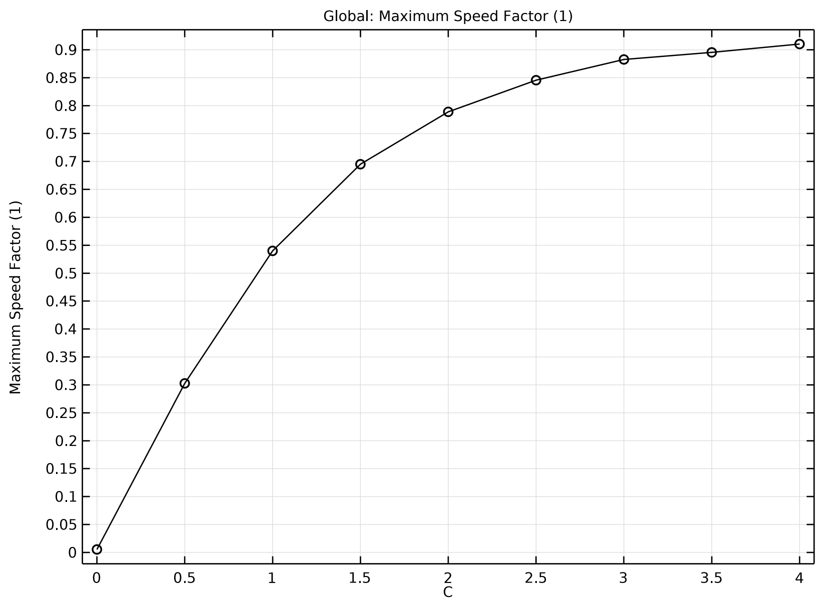

Figure 7: Maximum speed factor W_{max}, *as a function of the blade velocity ratio* C.

Reference

1. Y. Li and others, "Numerical investigation of three turbomolecular pump models in the free molecular flow range," *Vacuum*, vol. 101, pp. 337–344, 2014.

Application Library path: `Particle_Tracing_Module/Vacuum_Systems/turbomolecular_pump`

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 **Mathematics>Mathematical Particle Tracing (pt)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

ROOT

Insert the prepared geometry sequence from file. You can read the instructions for creating the geometry in the appendix.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `turbomolecular_pump_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.
2 In the **Settings** window for **Component**, locate the **Curved Mesh Elements** section.
3 From the **Geometry shape function** list, choose **Linear Lagrange**. Linear geometry shape order is the most robust option for modeling diffuse scattering at convex curved surfaces, such as the tip wall in this geometry.

GLOBAL DEFINITIONS

Geometry Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Physics Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
2 In the **Settings** window for **Parameters**, type `Physics Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
M12	pt.pcnt1.alpha		Transmission probability, forward
M21	pt.pcnt2.alpha		Transmission probability, backward
Kmax	M12/M21		Maximum Compression Ratio
Wmax	M12-M21		Maximum Speed Factor

MATHEMATICAL PARTICLE TRACING (PT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mathematical Particle Tracing (pt)**.

2 In the **Settings** window for **Mathematical Particle Tracing**, locate the **Particle Release and Propagation** section.

3 In the **Maximum number of secondary particles** text field, type `0`.

Particle Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mathematical Particle Tracing (pt)** click **Particle Properties 1**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Mass** section.

3 In the m_p text field, type `m`.

Rotating Frame 1

1 In the **Physics** toolbar, click **Domains** and choose **Rotating Frame**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 In the Ω text field, type `omega`.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet Boundary**.
4. Locate the **Initial Position** section. From the **Initial position** list, choose **Density**.
5. In the N text field, type `10000`.
6. Locate the **Initial Velocity** section. From the **Initial velocity** list, choose **Thermal**.
7. In the T text field, type `T0`.
8. Click to expand the **Advanced Settings** section. Select the **Subtract moving frame velocity from initial particle velocity** check box.

Inlet 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet Boundary**.
4. Locate the **Initial Position** section. From the **Initial position** list, choose **Density**.
5. In the N text field, type `10000`.
6. Locate the **Initial Velocity** section. From the **Initial velocity** list, choose **Thermal**.
7. In the T text field, type `T0`.
8. Locate the **Advanced Settings** section. Select the **Subtract moving frame velocity from initial particle velocity** check box.

Particle Counter (Inlet 1 Transmission)

1. In the **Physics** toolbar, click **Boundaries** and choose **Particle Counter**.
2. In the **Settings** window for **Particle Counter**, type `Particle Counter (Inlet 1 Transmission)` in the **Label** text field.
3. Locate the **Boundary Selection** section. From the **Selection** list, choose **Outlet Boundary**.
4. Locate the **Particle Counter** section. From the **Release feature** list, choose **Inlet 1**.

Particle Counter (Inlet 2 Transmission)

1. In the **Physics** toolbar, click **Boundaries** and choose **Particle Counter**.
2. In the **Settings** window for **Particle Counter**, type `Particle Counter (Inlet 2 Transmission)` in the **Label** text field.
3. Locate the **Boundary Selection** section. From the **Selection** list, choose **Inlet Boundary**.
4. Locate the **Particle Counter** section. From the **Release feature** list, choose **Inlet 2**.

Root and Blade Surfaces

1 In the **Physics** toolbar, click **Boundaries** and choose **Thermal Re-Emission**.

2 In the **Settings** window for **Thermal Re-Emission**, type `Root and Blade Surfaces` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **All Moving Walls**.

4 Locate the **Wall Properties** section. In the T text field, type `T0`.

Tip Wall

1 In the **Physics** toolbar, click **Boundaries** and choose **Thermal Re-Emission**.

2 In the **Settings** window for **Thermal Re-Emission**, type `Tip Wall` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Tip Wall**.

4 Locate the **Wall Properties** section. In the T text field, type `T0`.

5 Locate the **Frame Settings** section. Select the **Subtract moving frame velocity from reflected particle velocity** check box.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
C (Blade velocity ratio at rms radius)	`range(0,0.5,4)`	

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **ms**.

4 In the **Output times** text field, type `range(0,0.01,0.5)`.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories (pt)

The default plot shows the particle positions at the final time step for the highest value of the speed factor. The particles are colored by velocity. Instead, color them by release feature to better visualize the transmission probability across the stage.

1 In the **Model Builder** window, expand the **Particle Trajectories (pt)** node.

Color Expression 1

1 In the **Model Builder** window, expand the **Results>Particle Trajectories (pt)> Particle Trajectories 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Mathematical Particle Tracing>Particle statistics>pt.prf - Particle release feature**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Traffic>TrafficLight** in the tree.

5 Click **OK**.

Particle Trajectories (pt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (pt)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Clear the **Show legends** check box.

4 In the **Particle Trajectories (pt)** toolbar, click **Plot**.

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

The green particles were released from the top boundary while the red particles were released from the bottom boundary. Plot the particle trajectories at different values of the dimensionless speed factor, C, to observe how the rotating frame breaks the symmetry of the red and green particles. Several plots are shown in Figure 3.

M12

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

2 In the **Settings** window for **1D Plot Group**, type `M12` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Time selection** list, choose **Last**.

5 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **M12** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
M12	1	Fraction of particles transmitted from the inlet to the outlet

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **C**.

5 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Black**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

7 In the **M12** toolbar, click **Plot**. Compare the resulting plot with Figure 4.

M21

1 In the **Model Builder** window, right-click **M12** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `M21` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **M21** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
M21	1	Fraction of particles transmitted from the outlet to the inlet

4 In the **M21** toolbar, click **Plot**. Compare the resulting plot with Figure 5.

Kmax

1 In the **Model Builder** window, right-click **M21** and choose **Duplicate**.

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

Global 1

1 In the **Model Builder** window, expand the **Kmax** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
Kmax	1	Maximum Compression Ratio

4 In the **Kmax** toolbar, click **Plot**.

5 Click the **y-Axis Log Scale** button in the **Graphics** toolbar. Compare the resulting plot with Figure 6.

Wmax

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

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

Global 1

1 In the **Model Builder** window, expand the **Wmax** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
Wmax	1	Maximum Speed Factor

4 In the **Wmax** toolbar, click **Plot**.

5 Click the **y-Axis Log Scale** button in the **Graphics** toolbar. Compare the resulting plot with Figure 7.

Appendix — Geometry Instructions

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

NEW

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

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

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. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `turbomolecular_pump_geom_sequence_parameters.txt`.

GEOMETRY 1

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 `rtip`.
4. In the **Height** text field, type `rh`.
5. Locate the **Position** section. In the **z** text field, type `-rh/2`.
6. Locate the **Rotation Angle** section. In the **Rotation** text field, type `30`.

Cylinder 2 (cyl2)

1. Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.
2. In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `rroot`.

Difference 1 (dif1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Select the object **cyl1** only.
3. In the **Settings** window for **Difference**, locate the **Difference** section.
4. Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.
5. Select the object **cyl2** only.
6. Click **Build Selected**.

Work Plane 1 (wp1)

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. From the **Plane type** list, choose **Coordinates**.
4. In row **Point 1**, set **x** to `wp11x`.
5. In row **Point 1**, set **y** to `wp11y`.
6. In row **Point 1**, set **z** to `wp11z`.

7 In row **Point 2**, set **x** to `wp12x`.

8 In row **Point 2**, set **y** to `wp12y`.

9 In row **Point 2**, set **z** to `wp12z`.

10 In row **Point 3**, set **x** to `wp13x`.

11 In row **Point 3**, set **y** to `wp13y`.

12 In row **Point 3**, set **z** to `wp13z`.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 From the **Plane type** list, choose **Coordinates**.

4 In row **Point 1**, set **x** to `wp21x`.

5 In row **Point 1**, set **y** to `wp21y`.

6 In row **Point 1**, set **z** to `wp21z`.

7 In row **Point 2**, set **x** to `wp22x`.

8 In row **Point 2**, set **y** to `wp22y`.

9 In row **Point 2**, set **z** to `wp22z`.

10 In row **Point 3**, set **x** to `wp23x`.

11 In row **Point 3**, set **y** to `wp23y`.

12 In row **Point 3**, set **z** to `wp23z`.

Partition Objects 1 (par1)

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

2 Select the object **dif1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 From the **Work plane** list, choose **Work Plane 1 (wp1)**.

Partition Objects 2 (par2)

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

2 Select the object **par1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Delete Entities 1 (del1)

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

2 On the object **par2**, select Boundary 7 only.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 On the object **par2**, select Domains 1–3 only.

6 Click **Build All Objects**.

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

Inlet Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inlet Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **del1**, select Boundary 3 only.

Outlet Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outlet Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **del1**, select Boundary 4 only.

Root Wall

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Root Wall` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **del1**, select Boundary 1 only.

Tip Wall

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Tip Wall` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **del1**, select Boundary 6 only.

Blade Surfaces

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Blade Surfaces` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **del1**, select Boundaries 2 and 5 only.

All Moving Walls

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `All Moving Walls` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Root Wall** and **Blade Surfaces**.

6 Click **OK**.