Cascade Impactor

Introduction

A cascade impactor is an inertial particle-separation device that separates the particles in an aerosol based on their sizes. It has multiple stages, each of which can trap particles on a flat collector plate and is connected to the other stages by small nozzles. The nozzles become finer as the particles progress from the top stage to the bottom stage. Particle-laden air enters from the top inlet and passes through nozzles. The air jets from these nozzles impact on collector plates and each stage collects finer particles than its predecessor.

Model Definition

The geometry consists of an inlet at the top attached to a preseparator stage followed by five stages with collector plates. These stages and collector plates are numbered in the range 0–4 starting from the top. Each collector stage has two parts: an upper part that contains the collector plates and a hollow lower part to let the aerosol flow downward. The diameters of collector plates at each stage are identical. The collector plates at stages 0 and 1 have a hole in the center which allows the aerosol with smaller particles to flow through the center and periphery of collector plates. The preseparator stage and collector stages are connected by number of cylindrical nozzles. The top two stages and preseparators are connected by larger nozzles while the bottom three stages are connected to each other by more finer nozzles. The outlet is located at the bottom of impactor. All the dimensions mentioned here can be changed by editing the parameters of geometry.

The aerosol containing spherical particles of density 2200 kg/m^3 and sizes ranging from 1 μm to 5 μm enters the inlet at the flow rate of 3 l/min. The **Gravity** feature is used to apply gravitational force. The geometry is axially symmetrical so only a sector of geometry of 30° is built as shown in Figure 1 and the **Symmetry** feature is used which highly reduces the computational time. Drag and lift forces are applied in entire domain using the **Drag Force** and the **Lift Force** features.

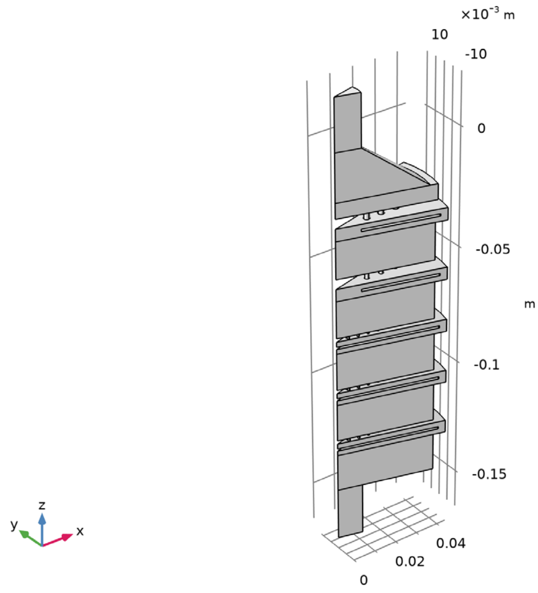

Figure 1: Model geometry of a cascade impactor device.

Notes on COMSOL Implementation

The model is solved in two steps. First, the fluid velocity and pressure are solved using the Laminar Flow interface and **Stationary** study step. Then the particle trajectories are solved for using the Particle Tracing for Fluid Flow interface and **Time Dependent** study step.

A total of 2000 particles are released having random diameter between 1 μm and 5 μm. The particles are released along the inlet boundary with number density proportional to the fluid velocity magnitude. To release particles with random diameters, **Specify particle diameter** needs to be chosen from the **Particle size distribution** options in the **Additional Variables** section of the Particle Tracing for Fluid Flow interface. The **Bounce** wall condition is applied at the boundaries of all nozzles using the **Wall** feature to stop an inordinate amount of particles from getting frozen as they pass through the nozzles.

The relaxation time, τ_p, of particles in the Stokes drag law is given by

$$\tau_p = \frac{\rho_p d_p^2}{18\mu}$$

where

- μ is the fluid viscosity (SI unit: Pa·s),
- ρ_p is the particle density (SI unit: kg/m^3), and
- d_p is the particle diameter (SI unit: m).

For the smallest particles in this model, the relaxation time is approximately 7 μs, which is about thousands times smaller than the maximum output time. However, the default time-dependent solver for most particle tracing models is a second-order implicit method that handles numerically stiff problems rather well, even when taking time steps that are larger than the relaxation time. Nevertheless, the smallest particles are still the main driver of the computational cost of transient inertial particle tracing simulations. In the **Time Dependent** study step, a time step of 0.008 s is used to reduce the number of result outputs. This time step is very large and can be a source of instability in the solution. So, a constant **Maximum step constraint** of 0.1 ms is used in the **Time-Dependent Solver** node under the **Solver Configurations** under the **Study**, which limits the maximum time step size taken by the solver.

Results and Discussion

The velocity field and the streamlines are shown in Figure 2. As the flow is fully developed, velocity at center of inlet is higher. Fluid velocity further increases at nozzles due to smaller cross-sectional area. Fluid follows a specific path from inlet to outlet so laminar flow assumption works well. Some randomness in fluid path might occur for larger flow rate at inlet. In such situation, Turbulent Flow interface must be used.

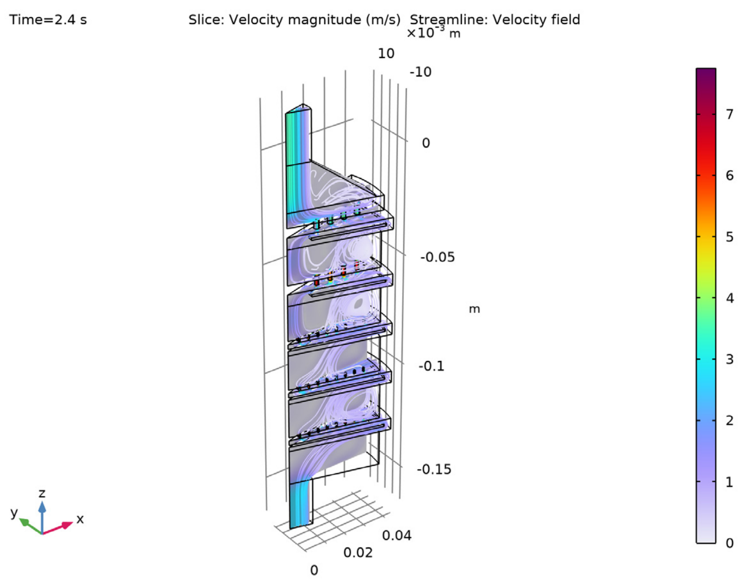

Figure 2: Velocity field and streamlines for the fluid flow.

Larger particles are carried by air for shorter distance and ends up at the collector plates at stages 0 and 1 while smaller particles are carried to the deeper stages (Figure 3). This inertial particle separation is basic principle used in a cascade impactor. Different ranges of particles can be separated by adjusting the geometry and inlet fluid flow rate. Figure 4 is a 2D histogram that shows the particle sizes collected at different collector plates.

Because the model is fully parameterized, a natural extension would be to use a **Parametric Sweep** or some optimization functionality to study effect of different sized nozzles or varying inlet flow rate on the range of particle sizes collected at different stages.

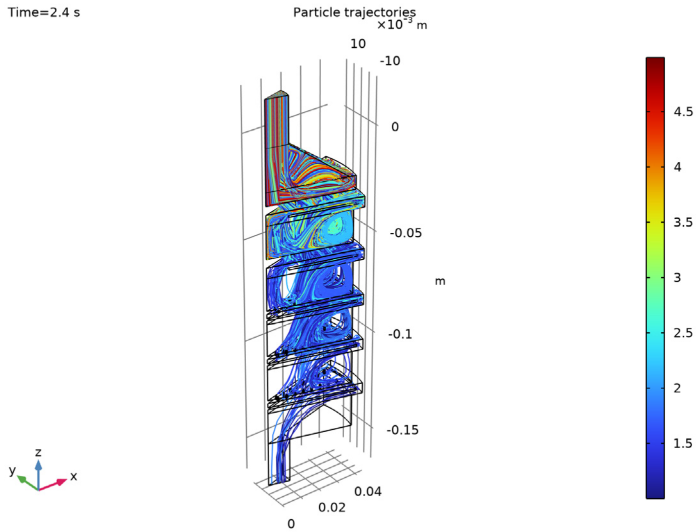

Figure 3: Particle trajectories colored by particle size.

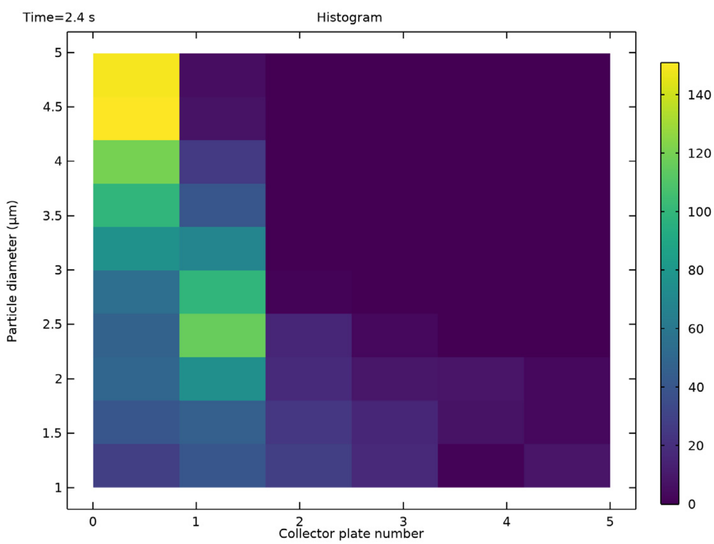

Figure 4: 2D histogram showing the range of particle size collected at each collector plates.

Application Library path: `Particle_Tracing_Module/Fluid_Flow/cascade_impactor`

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>Single-Phase Flow>Laminar Flow (spf)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Particle Tracing> Particle Tracing for Fluid Flow (fpt)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Stationary**.

8 Click **Done**.

GEOMETRY 1

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

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 `cascade_impactor_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**. The geometry should look like Figure 1.

DEFINITIONS

Symmetry Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Symmetry Boundaries` in the **Label** text field.

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

4 Select Boundaries 1, 2, 4–7, 11, 12, 14, 15, 19, 20, 22, 23, 27, 28, 30, 31, 33, 34, 36, 37, 39, 40, and 42–45 only. These are the flat vertical surfaces on either side of the model geometry.

Next define a variable to indicate which plate, if any, each particle hits. The variable will be assigned different values for different boundary selections.

Variables 1

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

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

4 Select Boundary 102 only. This is the upper surface of the first plate.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
pn	0		Collector plate number

Variables 2

1 Right-click **Variables 1** and choose **Duplicate**.

2 Select Boundary 99 only. This is the upper surface of the second plate.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
pn	1		Collector plate number

Variables 3

1 Right-click **Variables 2** and choose **Duplicate**.

2 Select Boundary 25 only. This is the upper surface of the third plate.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
pn	2		Collector plate number

Variables 4

1 Right-click **Variables 3** and choose **Duplicate**.

2 Select Boundary 17 only. This is the upper surface of the fourth plate.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
pn	3		Collector plate number

Variables 5

1 Right-click **Variables 4** and choose **Duplicate**.

2 Select Boundary 9 only. This is the upper surface of the fifth plate.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
pn	4		Collector plate number

Variables 6

1 Right-click **Variables 5** and choose **Duplicate**.

2 Select Boundary 3 only. This is the outlet boundary, which is at bottom of the geometry.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
pn	5		Collector plate number

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Air**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Include gravity** check box.

Inlet 1

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

2 Select Boundary 46 only. This is at the top of the geometry.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

6 In the V_0 text field, type `3[l/min]`.

Outlet 1

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

2 Select Boundary 3 only. This is at the bottom of the geometry.

Symmetry 1

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

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

3 From the **Selection** list, choose **Symmetry Boundaries**.

PARTICLE TRACING FOR FLUID FLOW (FPT)

Allow particles to be released with a distribution of different sizes. The default behavior is to release particles with uniform diameter.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Particle Tracing for Fluid Flow (fpt)**.

2 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Additional Variables** section.

3 From the **Particle size distribution** list, choose **Specify particle diameter**.

Particle Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Particle Tracing for Fluid Flow (fpt)** click **Particle Properties 1**.

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

3 From the ρ_p list, choose **User defined**. Use the default value of the particle density.

Wall 2

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

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

3 From the **Selection** list, choose **All Nozzles**.

4 Locate the **Wall Condition** section. From the **Wall condition** list, choose **Bounce**.

Inlet 1

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

2 Select Boundary 46 only.

3 In the **Settings** window for **Inlet**, locate the **Initial Position** section.

4 From the **Initial position** list, choose **Density**.

5 In the N text field, type `2000`.

6 In the ρ text field, type `spf.U`.

7 Locate the **Initial Velocity** section. From the **u** list, choose **Velocity field (spf)**.

8 Locate the **Initial Particle Diameter** section. From the **Distribution function** list, choose **Uniform**.

9 From the **Sampling from distribution** list, choose **Random**.

10 In the $d_{\mathrm{p,max}}$ text field, type `5[um]`.

For the minimum diameter, the default value is used.

Outlet 1

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

2 Select Boundary 3 only.

Symmetry 1

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

Symmetry in a particle tracing context amounts to specular reflection with the assumption that for every particle that leaves the model geometry, a different particle enters the geometry with equal speed.

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

3 From the **Selection** list, choose **Symmetry Boundaries**.

Drag Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Drag Force**.

2 In the **Settings** window for **Drag Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Drag Force** section. From the **u** list, choose **Velocity field (spf)**.

Lift Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Lift Force**.

2 In the **Settings** window for **Lift Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Lift Force** section. From the **u** list, choose **Velocity field (spf)**.

Gravity Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Gravity Force**.

2 In the **Settings** window for **Gravity Force**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

STUDY 1

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent> Time Dependent**.

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

3 In the **Output times** text field, type `range(0,0.008,2.4)`.

4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Laminar Flow (spf)		Automatic (Stationary)
Particle Tracing for Fluid Flow (fpt)	√	Automatic (Time dependent)

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Maximum step constraint** list, choose **Constant**.

5 In the **Maximum step** text field, type `0.0001`.

The inertial particle tracing problem is numerically stiff, so a small time step is needed to resolve the acceleration of the particles.

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

RESULTS

Slice

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 In the **Planes** text field, type `1`.

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

6 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

7 Click **OK**.

Velocity (spf)

In the **Velocity (spf)** toolbar, click **Streamline**.

Streamline 1

1 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

2 From the **Positioning** list, choose **Starting-point controlled**.

3 In the **Points** text field, type `100`.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice**.

Color Expression 1

1 In the **Velocity (spf)** toolbar, click **Color Expression**.

2 Click **Plot**. The plot should look like Figure 2.

Particle Trajectories 1

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

2 In the **Settings** window for **Particle Trajectories**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `fpt.dp`.

4 From the **Unit** list, choose **µm**.

5 In the **Particle Trajectories (fpt)** toolbar, click **Plot**. The plot should look like Figure 3.

Histogram

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Particle 1**.

Histogram 1

1 In the **Histogram** toolbar, click **More Plots** and choose **Histogram**.

2 In the **Settings** window for **Histogram**, locate the **x-Expression** section.

3 In the **Expression** text field, type `bndenv(pn)`.

4 Select the **Description** check box. In the associated text field, type `Collector plate number`.

5 Locate the **y-Expression** section. In the **Expression** text field, type `fpt.dp`.

6 From the **Unit** list, choose **µm**.

7 Locate the **Bins** section. Find the **x bins** subsection. In the **Number** text field, type `6`.

8 Locate the **Output** section. From the **Function** list, choose **Discrete**.

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

10 In the **Color Table** dialog box, select **Linear>Viridis** in the tree.

11 Click **OK**.

12 In the **Histogram** toolbar, click **Plot**.

13 Click the **Zoom Extents** button in the **Graphics** toolbar. The plot should look like Figure 4.

Appendix: Geometry 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 Click **Done**.

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 `cascade_impactor_geom_sequence_parameters.txt`.

GEOMETRY 1

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** list, choose **xz-plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `Ri`.

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

5 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `Rpsb`.

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

5 Locate the **Position** section. In the **yw** text field, type `-Tps`.

6 Click **Build Selected**.

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

Work Plane 1 (wp1)>Polygon 1 (pol1)

1. In the **Work Plane** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Coordinates** section.
3. In the table, enter the following settings:

xw (m)	**yw (m)**
`0`	`0`
`Ri`	`0`
`0.925*Rpsb`	`Tpsb-Tps`
`0`	`Tpsb-Tps`

4. Click **Build Selected**.

Work Plane 1 (wp1)>Rectangle 3 (r3)

1. In the **Work Plane** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rch`.
4. In the **Height** text field, type `Tch`.
5. Locate the **Position** section. In the **yw** text field, type `-Tps-L0-Tch`.
6. Click **Build Selected**.
7. Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Rectangle 4 (r4)

1. In the **Work Plane** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rc`.
4. In the **Height** text field, type `Tc`.
5. Locate the **Position** section. In the **yw** text field, type `-Tps-L0-Tch-Tc`.
6. Click **Build Selected**.
7. Click the **Zoom Extents** button in the **Graphics** toolbar.

Work Plane 1 (wp1)>Rectangle 5 (r5)

1. In the **Work Plane** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rcol12`.
4. In the **Height** text field, type `Tcol`.

5 Locate the **Position** section. In the **xw** text field, type `Rcol35-Rcol12`.

6 In the **yw** text field, type `-Tps-L0-Gap0-Tcol`.

7 Click **Build Selected**.

Work Plane 1 (wp1)>Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r3** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **r5** only.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **dif1** and **r4** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **yw size** text field, type `2`.

5 Locate the **Displacement** section. In the **yw** text field, type `-L1-Tch-Tc`.

6 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 6 (r6)

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

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

3 In the **Width** text field, type `Rch`.

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

5 Locate the **Position** section. In the **yw** text field, type `-Tps-L0-L1-L24-3*Tch-2*Tc`.

6 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 7 (r7)

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

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

3 In the **Width** text field, type `Rc`.

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

5 Locate the **Position** section. In the **yw** text field, type `-Tps-L0-L1-L24-3*Tch-3*Tc`.

6 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 8 (r8)

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

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

3 In the **Width** text field, type `Rcol35`.

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

5 Locate the **Position** section. In the **yw** text field, type `-Tps-L0-L1-L24-2*Tch-2*Tc-Gap1-Tcol`.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Difference 2 (dif2)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r6** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **r8** only.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Array 2 (arr2)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **dif2** and **r7** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **yw size** text field, type `3`.

5 Locate the **Displacement** section. In the **yw** text field, type `-L24-Tch-Tc`.

6 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 9 (r9)

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

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

3 In the **Width** text field, type `Re`.

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

5 Locate the **Position** section. In the **yw** text field, type `-Tps-L0-L1-3*L24-5*Tch-5*Tc-Te`.

6 Click **Build Selected**.

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

Revolve 1 (rev1)

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

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **Start angle** text field, type `-15`.

5 In the **End angle** text field, type `15`.

6 Click **Build Selected**.

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

Cumulative Selections

In the **Geometry** toolbar, click **Selections** and choose **Cumulative Selections**.

All Nozzles

1 Right-click **Cumulative Selections** and choose **Cumulative Selection**.

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

Thin Nozzles

1 In the **Model Builder** window, right-click **Cumulative Selections** and choose **Cumulative Selection**.

2 In the **Settings** window for **Selection**, type `Thin Nozzles` in the **Label** text field.

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 `R0`.

4 In the **Height** text field, type `L0`.

5 Locate the **Position** section. In the **x** text field, type `14[mm]`.

6 In the **z** text field, type `-Tps-L0`.

7 Click **Build Selected**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **cyl1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `4`.

5 Locate the **Displacement** section. In the **x** text field, type `D12`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **All Nozzles**.

7 Click **Build Selected**.

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 `R1`.

4 In the **Height** text field, type `L1`.

5 Locate the **Position** section. In the **x** text field, type `14[mm]`.

6 In the **z** text field, type `-Tps-L0-Tch-Tc-L1`.

7 Click **Build Selected**.

Array 2 (arr2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **cyl2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `4`.

5 Locate the **Displacement** section. In the **x** text field, type `D12`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **All Nozzles**.

7 Click **Build Selected**.

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 `R2`.

4 In the **Height** text field, type `L24`.

5 Locate the **Position** section. In the **x** text field, type `5[mm]`.

6 In the **z** text field, type `-Tps-L0-L1-2*Tch-2*Tc-L24`.

7 Click **Build Selected**.

Array 3 (arr3)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **cyl3** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `8`.

5 Locate the **Displacement** section. In the **x** text field, type `D35`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Thin Nozzles**.

7 Click **Build Selected**.

Array 4 (arr4)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 In the **Settings** window for **Array**, locate the **Input** section.

3 From the **Input objects** list, choose **Thin Nozzles**.

4 Locate the **Size** section. In the **z size** text field, type `3`.

5 Locate the **Displacement** section. In the **z** text field, type `-L24-Tch-Tc`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Thin Nozzles**.

7 Click **Build Selected**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Thin Nozzles**.

4 Locate the **Rotation** section. In the **Angle** text field, type `-7.5 7.5`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **All Nozzles**.

6 Click **Build Selected**.

Union 1 (uni1)

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

2 Click the **Select All** button in the **Graphics** toolbar.

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

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

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.