Created in COMSOL Multiphysics 6.3

Turbulent Flow Around a Factory Chimney

Introduction

The role of a chimney is to disperse the flue gases, typically from a household fireplace or a process in a factory, above the roof of the building, in order to get the potentially harmful flue gases dispersed and diluted in the air. The further away from people, the better. As a result, factory chimneys can be several tens of meters high. While being very tall structures, the chimney will be subjected to large and varying wind loads over time. Such wind loads are important factors to take into account when designing a factory chimney. If a cylinder is placed in a uniform flow field, vortexes will form as the flow separates in the region downstream of the chimney. These vortexes will start shedding and a von Kármán vortex street will be established. The vortexes of a tall cylinder may be shed uniformly along the length of the chimney, and there will be a significant transverse force component that acts periodically on the chimney. If the frequency of these transverse forces are close to the structural eigen-frequencies of the chimney, catastrophic failure can occur. To remedy such simultaneous shedding, one can add helical strakes to the chimney. These are thin structures that coil in a helical manner along the exterior of the chimney. They help to change the vortex shedding frequency and distribute the vortex shedding forces, as vortexes will not be shed simultaneously along the length of the chimney due to the ever changing geometry in the vertical direction.

The shedding frequency of a cylinder subjected to external uniform flow can be found with the equation

$$\mathrm{St}(\mathrm{Re}) = \frac{fD}{V}$$

where St(Re) is the Strouhal number which is dependent on the Reynolds number, Re, f is the shedding frequency, D is the diameter of the cylinder and V is the flow velocity. The Strouhal number usually takes a value between 0.18 and 0.22. Thus, with a diameter of 3 m and a flow velocity of 25 m/s, the shedding frequency can be calculated to be around 1.8 Hz. For a cylinder with strakes, the effective diameter is increased, and also the geometry is constantly changing. Thus, the location of flow detachment for the vortexes will vary along the length of the chimney. The transverse forces on the chimney will also cancel out when integrated over the whole chimney.

The geometry of the helical strakes has been studied since the 1950s, and it has been found that three strakes separated by 120 degrees with a pitch of five times the diameter of the chimney is the best configuration.

Model Definition

The chimney model consists of a factory building which is 50 m long, 30 m wide, and 8 m high. In the middle of the factory, a chimney of 3 m diameter rises up 25 m above the roof. Three strake fins are modeled using the interior wall boundary condition and they have a width of 20% of the chimney radius. The strakes, whose pitch is five times the chimney diameter, are helically wrapped around the outside of the chimney.

The factory is placed inside a block which is 90 m long, 100 m wide, and 37.5 m high. A few mesh control domains are added so that it is easier to refine the mesh downstream of the chimney. The model geometry is shown in Figure 1.

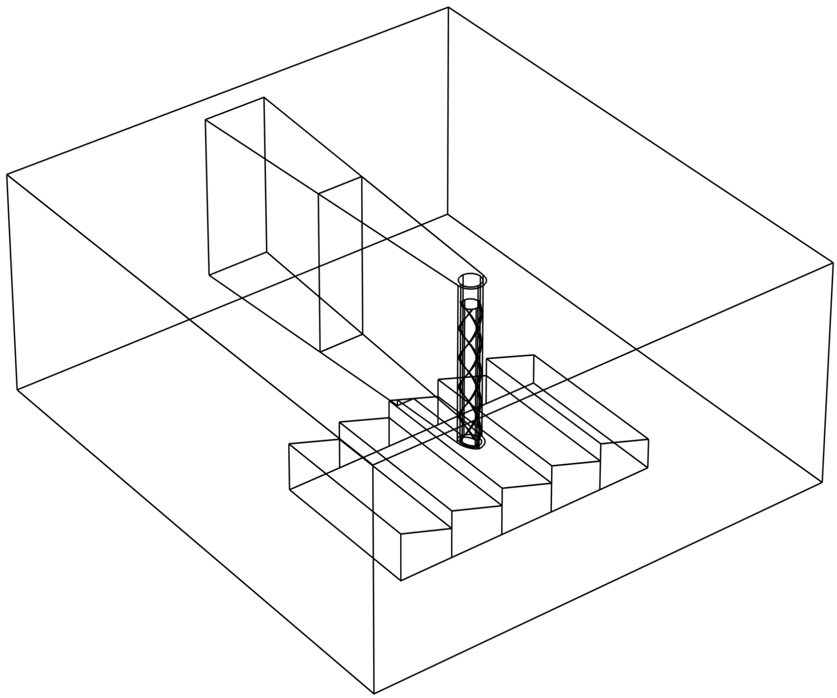

Figure 1: Geometry of the computational domain with mesh domains included downstream of the chimney.

The inlet velocity is set to have a profile that varies in the vertical direction. The velocity profile is given as

$$v_n(z) = 20\left(\frac{z}{15}\right)^{0,27}$$

The factory, the chimney, and the ground surfaces are modeled as wall with automatic wall treatment. Furthermore, all other exterior surfaces are modeled as open boundaries. The RANS SST turbulence model is employed in the current study. This model is capable of resolving the near-wall turbulence that is induced by the chimney. Thus, it reproduces the dynamic motion of the separation point, which gives rise to the vortex shedding phenomena. The first solution of the model gives a stationary solution on a relatively coarse mesh that is not capable of resolving the boundary layer to a sufficient level of accuracy. To solve the model with a finer mesh that resolves the boundary layer to a sufficient degree would require a computer with at least 250 GB RAM. Moreover, using a time-dependent solver leads to longer computational time. In the final part of the results section, some plots of such a model with finer mesh resolution are shown.

Results and Discussion

Figure 2 and Figure 3 show slice plots of the velocity magnitude for the stationary solutions without and with strakes, respectively. Notice the difference in the wakes that form downstream of the chimney. The model with strakes, as shown in Figure 3, has its wake region consisting of several smaller flow structures, resulting in a more complex flow field.

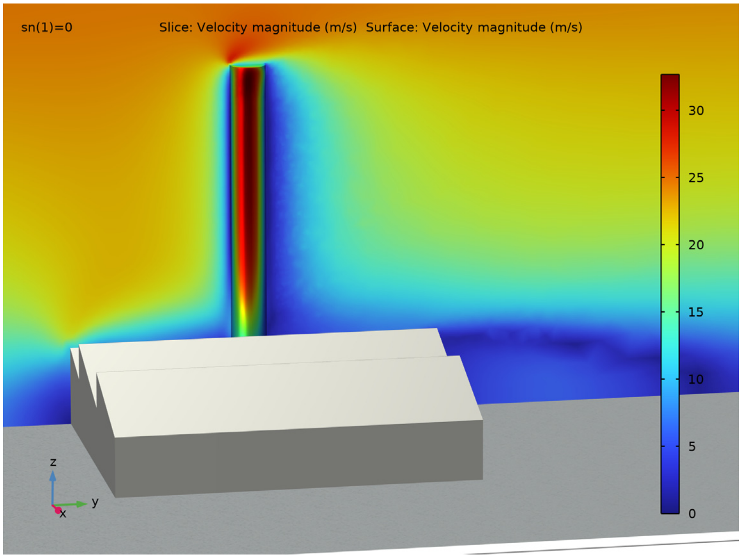

Figure 2: Vertical slice plot of the velocity magnitude for the model without strakes.

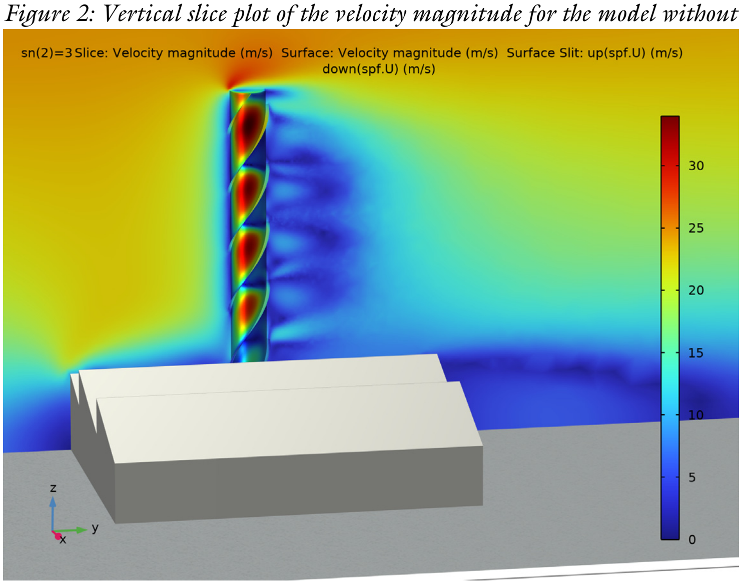

Figure 3: Vertical slice plot of the velocity magnitude for the model with strakes.

Figure 4 and Figure 5 show horizontal cut planes, colored by velocity magnitude, for the stationary solutions without and with strakes. Notice the shorter downstream extent of the wake for the chimney equipped with strakes.

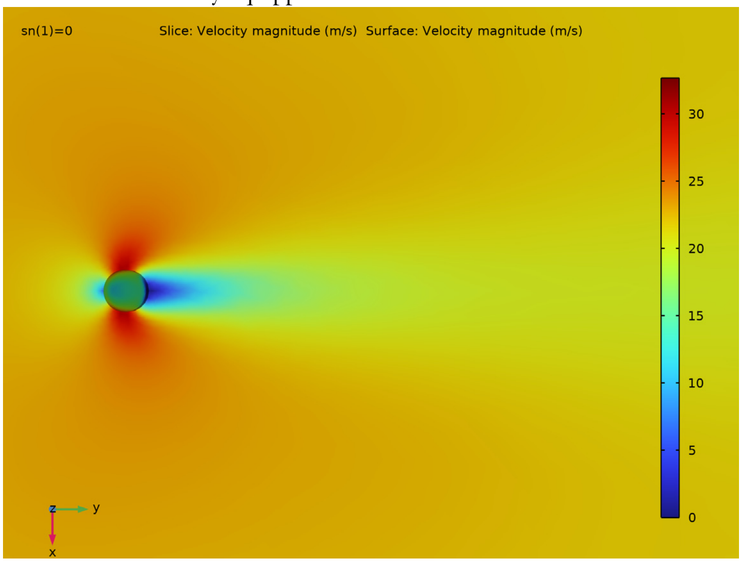

Figure 4: Horizontal slice plot of the velocity magnitude for the model without strakes.

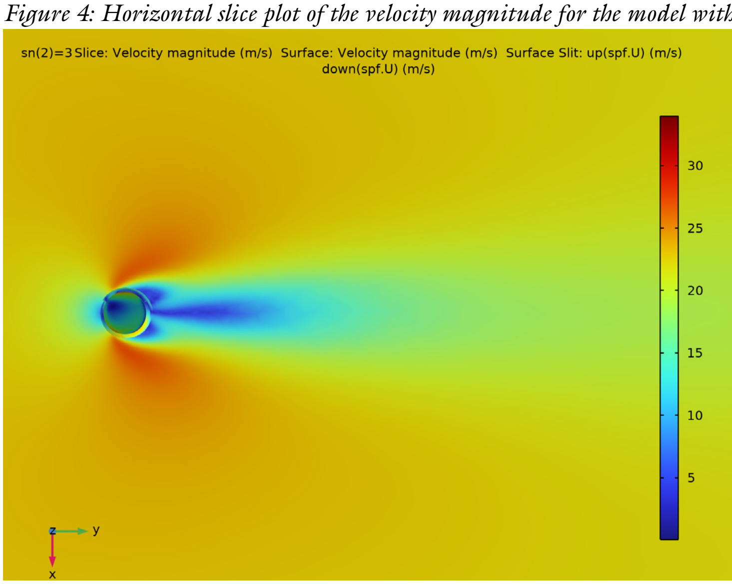

Figure 5: Horizontal slice plot of the velocity magnitude for the model with strakes.

Figure 6 shows the pressure distribution on the chimney and in a vertical cut plane.

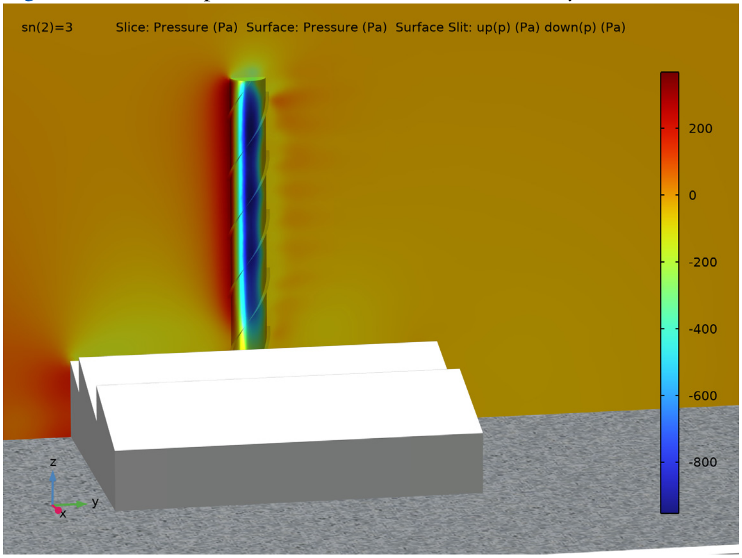

Figure 6: Pressure distribution on the surface of the chimney and in a vertical cut plane with strakes.

In a model with a refined mesh that resolves the boundary layers down to around 40 viscous units, one can see in Figure 7 and Figure 8 that the vortex structures downstream the chimney are refined and much more complex, creating local recirculation zones that you cannot see for a mesh-refined model of the chimney without strakes Figure 9.

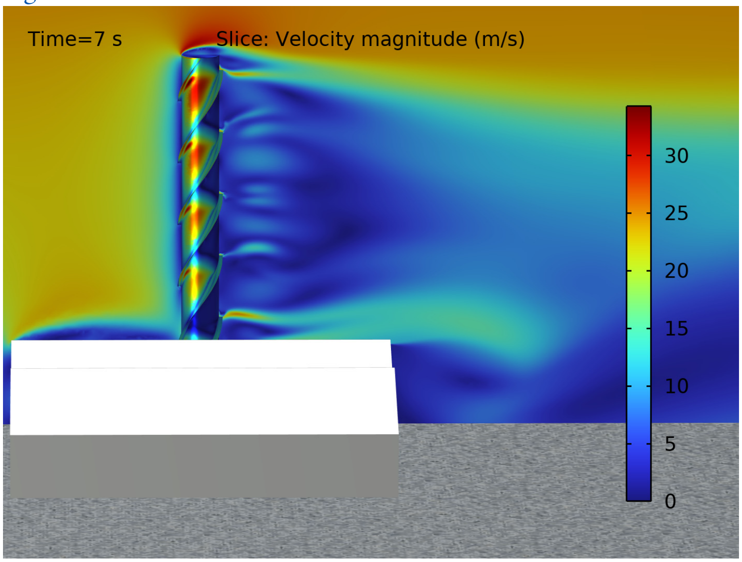

Figure 7: Snapshot at t = 7 s of the velocity magnitude for a model with strakes and a refined mesh.

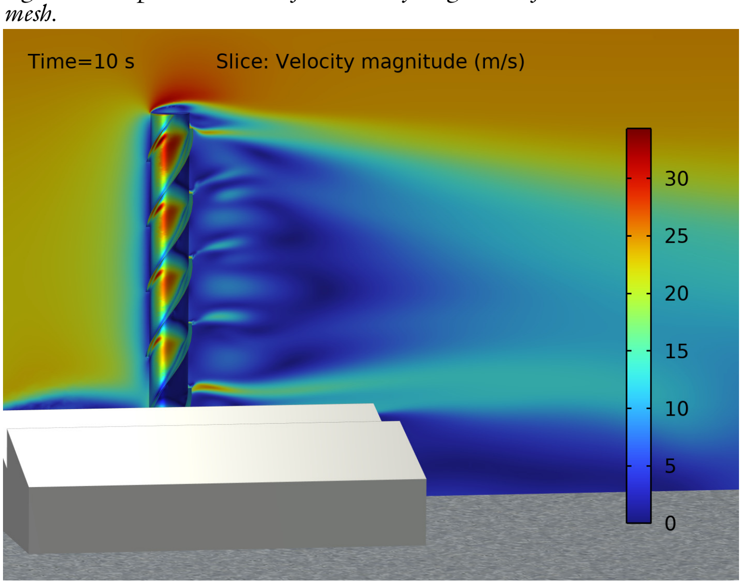

Figure 8: Snapshot at t = 10 s of the velocity magnitude for a model with strakes and a refined mesh. Notice the change in some of the recirculation patterns compared to previous image.

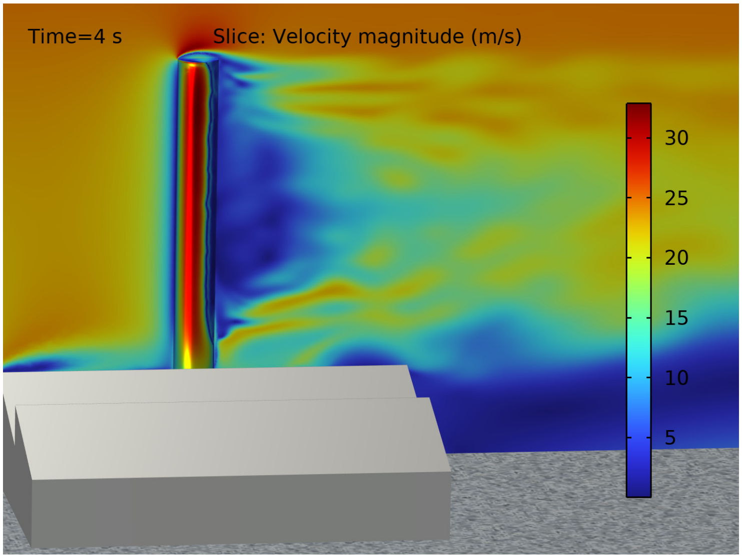

Figure 9: Snapshot at t = 4 s of the velocity magnitude for a model without strakes but with a refined mesh. Notice that the separation line on the downstream surface of the chimney is rather uniform along the height.

In Figure 10, a snapshot of the streamlines, colored by the turbulent kinetic energy, is shown for the time-dependent study with strakes.

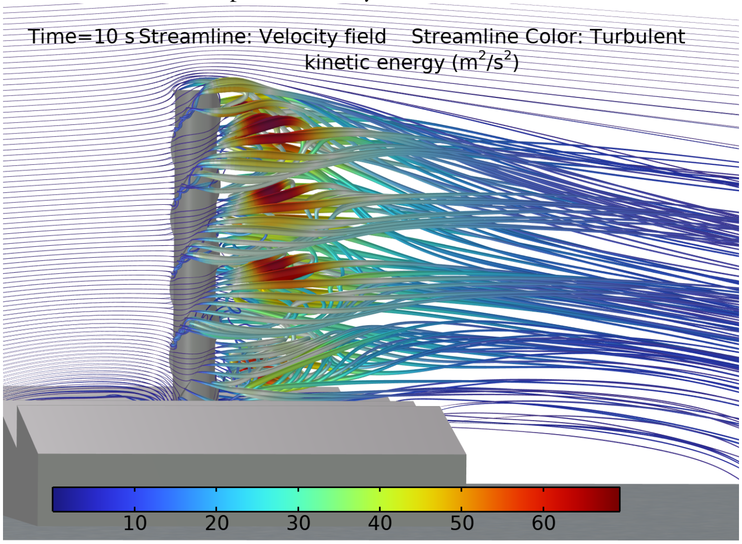

Figure 10: Streamline plot colored by the turbulent kinetic energy at t = 10 s for the model with strakes and a refined mesh. Note the increase in turbulent energy, as well as the complex periodicity in the flow downstream of the chimney.

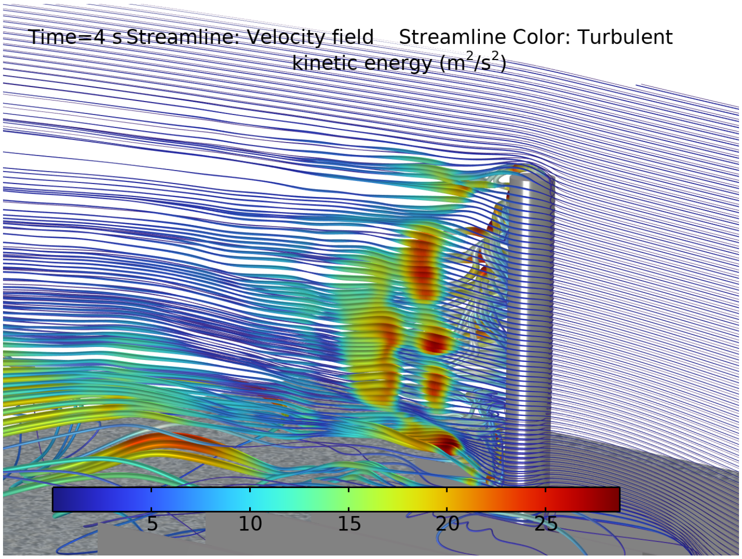

Figure 11: Streamline plot colored by the turbulent kinetic energy at t = 10 s for the model without strakes, but with a refined mesh. An oscillatory behavior can be seen on the streamlines.

Notes About the COMSOL Implementation

The instructions below are for a stationary simulation with a coarse mesh. In order to obtain a stationary solution, isotropic diffusion had to be added. This stabilizes the solution, but smears out the results to some degree. To resolve the time dependent shedding of vortexes, a finer mesh must be applied, both in the boundary layer on the chimney and in its downstream wake. The isotropic diffusion also needs to be removed. If the required mesh size were to be implemented in a full 3D model, it would result in nearly 80 million degrees of freedom.

Application Library path: `CFD_Module/Single-Phase_Flow/chimney`

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** > **Turbulent Flow** > **Turbulent Flow, SST (spf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Stationary with Initialization**.
6 Click **Done**.

First, define some parameters for the geometry.

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
fw	10[m]	10 m	Factory module width
fh	8[m]	8 m	Factory module height
fwh	3[m]	3 m	Factory window height
fn	5	5	Number of modules
fd	30[m]	30 m	Factory depth
c_d	3[m]	3 m	Chimney diameter
c_h	25[m]	25 m	Chimney height
swf	0.2	0.2	Strake width factor
sp	5*c_d	15 m	Strake pitch
sn	3	3	Number of strakes
d_w	fw*fn+fw*4	90 m	Domain width
d_l	100[m]	100 m	Domain length
d_b	20[m]	20 m	Space in front of building

Analytic 1 (an1)

1 In the **Home** toolbar, click **Functions** and choose **Global** > **Analytic**.

2 In the **Settings** window for **Analytic**, type `U_in` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `20*(x/15)^0.27`.

4 Locate the **Units** section. In the **Function** text field, type `m/s`.

5 In the table, enter the following settings:

Argument	Unit
x	m

6 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	50	1	0	m

7 Click **Plot**.

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) > 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
0	fh
fw	fh-fwh
fw	0

4 Click **Build Selected**.

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

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.
2 Select the object **pol1** only.
3 In the **Settings** window for **Array**, locate the **Size** section.
4 In the **xw size** text field, type `5`.
5 Locate the **Displacement** section. In the **xw** text field, type `fw`.
6 Click **Build Selected**.

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (m)
`fd`

4 Select the **Reverse direction** checkbox.
5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
6 In the **New Cumulative Selection** dialog, type `Factory building` in the **Name** text field.
7 Click **OK**.
8 In the **Settings** window for **Extrude**, click **Build All Objects**.

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 In the **z-coordinate** text field, type `fh-fwh`.

Work Plane 2 (wp2) > Plane Geometry

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

Work Plane 2 (wp2) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `c_d/2`.
4 Locate the **Position** section. In the **xw** text field, type `fw*fn/2`.

5 In the **yw** text field, type `fd/2`.

6 Click **Build Selected**.

7 Click **Build Selected**.

Extrude 2 (ext2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
c_h

4 Click **Build Selected**.

Add an if-condition for the number of strakes. This part of the sequence is just active if the number of strakes are larger than 0

If 1 (if1)

1 In the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `sn>0`.

Work Plane 3 (wp3)

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 **Face parallel**.

4 On the object **ext2**, select Boundary 4 only.

Work Plane 3 (wp3) > Plane Geometry

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

Work Plane 3 (wp3) > 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)
c_d/2	0
c_d/2*(1+swf)	0

4 Click **Build Selected**.

Work Plane 3 (wp3) > Rotate 1 (rot1)

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

2 Select the object **pol1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `range(0,360/sn,359)`.

5 Click **Build Selected**.

Extrude 3 (ext3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 3 (wp3)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
sp/8/sn
sp/8/sn*2
sp/8/sn*3
sp/8/sn*4
sp/8/sn*5
sp/8/sn*6
sp/8/sn*7
sp/8/sn*8

4 Select the **Reverse direction** checkbox.

5 Click to expand the **Scales** section. Click to expand the **Displacements** section. Click to expand the **Twist Angles** section. In the table, enter the following settings:

Twist angles (deg)
360/sn/8
360/sn/8*2

Twist angles (deg)
360/sn/8*3
360/sn/8*4
360/sn/8*5
360/sn/8*6
360/sn/8*7
360/sn/8*8

6 Click **Build Selected**.

Array 1 (arr1)

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

2 Select the object **ext3** only.

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

4 In the **z size** text field, type `ceil(c_h/sp*sn)`.

5 Locate the **Displacement** section. In the **z** text field, type `-sp/sn`.

6 Click **Build Selected**.

End If 1 (endif1)

1 In the **Model Builder** window, click **End If 1 (endif1)**.

2 In the **Settings** window for **End If**, click **Build Selected**.

Block 1 (blk1)

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

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

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

4 In the **Depth** text field, type `d_l`.

5 In the **Height** text field, type `d_h`.

6 Click to select the **Height** text field. Right-click and choose **Create Parameter**.

7 In the **Create Parameter** dialog, type `d_h` in the **Name** text field.

8 In the **Expression** text field, type `c_h*1.5`.

9 Click **OK**.

10 In the **Settings** window for **Block**, locate the **Position** section.

11 In the **x** text field, type `-(d_w-fw*fn)/2`.

12 In the **y** text field, type `-d_b`.

Adding some domains for mesh control may be useful for further development of the model, albeit they are not used in this tutorial.

Work Plane 4 (wp4)

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 **Face parallel**.

4 On the object **ext2**, select Boundary 4 only.

Work Plane 4 (wp4) > Plane Geometry

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

Work Plane 4 (wp4) > Circle 1 (c1)

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

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

3 In the **Radius** text field, type `c_d/2+1`.

Work Plane 4 (wp4) > 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)
-c_d/2-1	0
-c_d*2	d_l-d_b-fd/2
c_d*2	d_l-d_b-fd/2
c_d/2+1	0

4 Click **Build Selected**.

Work Plane 4 (wp4) > Union 1 (uni1)

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

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

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

4 Clear the **Keep interior boundaries** checkbox.

5 Click **Build Selected**.

Work Plane 4 (wp4) > Circle 2 (c2)

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

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

3 In the **Radius** text field, type `c_d/2*(1+swf+0.1)`.

Work Plane 4 (wp4) > Difference 1 (dif1)

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

2 Select the object **uni1** 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 **c2** only.

6 Click **Build Selected**.

Extrude 4 (ext4)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 4 (wp4)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 Select the **Reverse direction** checkbox.

4 In the table, enter the following settings:

Distances (m)
`c_h*1.1`

5 Click **Build Selected**.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Work Plane 5 (wp5)

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**.

4 In the **y-coordinate** text field, type `d_l/2`.

Partition Objects 1 (par1)

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

2 Select the object **ext4** only.

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

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

Difference 1 (dif1)

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

2 Select the objects **blk1** and **par1** 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 Click the **Select Box** button in the **Graphics** toolbar.

6 Select the objects **arr1(1,1,1)**, **arr1(1,1,2)**, **arr1(1,1,3)**, **arr1(1,1,4)**, **arr1(1,1,5)**, **ext1**, and **ext2** only.

7 Click **Build Selected**.

Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domains 2 and 3 only.

Extrude 2 (ext2)

1 In the **Model Builder** window, click **Extrude 2 (ext2)**.

2 In the **Settings** window for **Extrude**, locate the **Selections of Resulting Entities** section.

3 Find the **Cumulative selection** subsection. Click **New**.

4 In the **New Cumulative Selection** dialog, type `Chimney` in the **Name** text field.

5 Click **OK**.

Extrude 3 (ext3)

1 In the **Model Builder** window, click **Extrude 3 (ext3)**.

2 In the **Settings** window for **Extrude**, locate the **Selections of Resulting Entities** section.

3 Find the **Cumulative selection** subsection. Click **New**.

4 In the **New Cumulative Selection** dialog, type `Strakes` in the **Name** text field.

5 Click **OK**.

Chimney and Strakes

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

2 In the **Settings** window for **Union Selection**, type `Chimney and Strakes` 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, in the **Selections to add** list, choose **Chimney** and **Strakes**.

6 Click **OK**.

Explicit Selection 1 (sel1)

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

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **mcd1**, select Boundary 3 only.

Factory and ground

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

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

3 From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to add** list, choose **Factory building** and **Explicit Selection 1**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Factory and ground` in the **Label** text field.

8 Click **Build Selected**.

Add a union selection for all boundaries that should have boundary layers. This is necessary when we use a parametric sweep later on to run the simulation with and without strakes.

Boundary layer boundaries

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

2 In the **Settings** window for **Union Selection**, type `Boundary layer boundaries` 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, in the **Selections to add** list, choose **Chimney**, **Strakes**, and **Factory and ground**.

6 Click **OK**.

ADD MATERIAL

1 In the **Materials** 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 the **Add to Component** button in the window toolbar.

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

TURBULENT FLOW, SST (SPF)

Inlet 1

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

2 Select Boundary 2 only.

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

4 In the U_0 text field, type `U_in(z)`.

Open Boundary 1

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

2 Select Boundaries 1, 4, 5, and 167 only.

3 In the **Settings** window for **Open Boundary**, locate the **Turbulence Conditions** section.

4 In the U_{ref} text field, type `20[m/s]`.

Interior Wall 1

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

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

3 From the **Selection** list, choose **Strakes**.

To stabilize the solution, add inconsistent stabilization (isotropic diffusion) contributions to the navier stokes and turbulence equations. This is needed in such cases where no stationary solution can be found. This smooth out the solution, and makes it possible to find a stationary solution even though it would not converge without isotropic diffusion.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog, select **Physics** > **Stabilization** in the tree.

6 In the tree, select the checkbox for the node **Physics** > **Stabilization**.

7 Click **OK**.

8 In the **Model Builder** window, click **Turbulent Flow, SST (spf)**.

9 In the **Settings** window for **Turbulent Flow, SST**, click to expand the **Inconsistent Stabilization** section.

10 Find the **Navier-Stokes equations** subsection. Select the **Isotropic diffusion** checkbox.

11 Find the **Turbulence equations** subsection. Select the **Isotropic diffusion** checkbox.

12 Find the **Navier-Stokes equations** subsection. In the δ_{id} text field, type `0.15`.

13 Find the **Turbulence equations** subsection. In the δ_{id} text field, type `0.15`.

DEFINITIONS

Integration 1 (intop1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings** > **Integration**.

3 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

5 From the **Selection** list, choose **Strakes**.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **Chimney**.

Variables 1

1 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
Fstr_y	intop1(spf.T_trac_uy)+ intop1(spf.T_trac_dy)	N	
Fstr_x	intop1(spf.T_trac_dx)+ intop1(spf.T_trac_ux)	N	
Fch_y	intop2(spf.T_tracy)	N	
Fch_x	intop2(spf.T_tracx)	N	
F_y	Fstr_y+Fch_y	N	
F_x	Fstr_x+Fch_x	N	

Modify the mesh sequence so that we have at least 2 elements in the width of the strakes. Also, add the predefined "Boundary layer boundaries"-selection to make sure boundary layers are created on the strakes during the parametric sweep.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** right-click **Size 1** and choose **Build Selected**.

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

3 From the **Selection** list, choose **Boundary layer boundaries**.

Size 2

1 In the **Model Builder** window, click **Size 2**.

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

3 From the **Selection** list, choose **Strakes**.

4 Click to expand the **Element Size Parameters** section. In the **Maximum element size** text field, type `.15`.

5 In the **Minimum element size** text field, type `0.1`.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Mesh 1** > **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

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

3 From the **Selection** list, choose **Boundary layer boundaries**.

STUDY 1

Parametric Sweep

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

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

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

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sn (Number of strakes)	0 3	

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

RESULTS

Velocity (spf)

In the **Model Builder** window, expand the **Velocity (spf)** node.

Vertical velocity slice

1 In the **Model Builder** window, expand the **Velocity (spf)** node.

2 Right-click **Results** and choose **3D Plot Group**.

3 In the **Settings** window for **3D Plot Group**, type `Vertical velocity slice` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

Slice 1

1 Right-click **Vertical velocity slice** and choose **Slice**.

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

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

4 In the **Vertical velocity slice** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, right-click **Vertical velocity slice** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Slice 1**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Chimney and Strakes**.

4 In the **Vertical velocity slice** toolbar, click **Plot**.

5 From the **Selection** list, choose **Chimney**.

Vertical velocity slice

In the **Model Builder** window, under **Results** click **Vertical velocity slice**.

Surface Slit 1

1 In the **Vertical velocity slice** toolbar, click **More Plots** and choose **Surface Slit**.

2 In the **Settings** window for **Surface Slit**, locate the **Expression on the Upside** section.

3 In the **Expression** text field, type `up(spf.U)`.

4 Locate the **Expression on the Downside** section. In the **Expression** text field, type `down(spf.U)`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Selection 1

1 Right-click **Surface Slit 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Strakes**.

Surface 2

1 In the **Model Builder** window, right-click **Vertical velocity slice** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Title** section.

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

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Factory building**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

Surface 3

Right-click **Surface 2** and choose **Duplicate**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Selection 1**.

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

3 Click **Clear Selection**.

4 Select Boundary 3 only.

Material Appearance 1

1 In the **Model Builder** window, click **Material Appearance 1**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Material type** list, choose **Rock**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

5 Locate the **Color** section. Select the **Use the plot's color** checkbox.

Vertical velocity slice

1 In the **Model Builder** window, under **Results** click **Vertical velocity slice**.

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

3 Clear the **Plot dataset edges** checkbox.

4 In the **Vertical velocity slice** toolbar, click **Plot**.

Horizontal velocity slice

1 Right-click **Vertical velocity slice** and choose **Duplicate**.

2 In the **Model Builder** window, click **Vertical velocity slice 1**.

3 In the **Settings** window for **3D Plot Group**, type `Horizontal velocity slice` in the **Label** text field.

Slice 1

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

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

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

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

5 In the **Horizontal velocity slice** toolbar, click **Plot**.

Pressure slice

1 In the **Model Builder** window, right-click **Vertical velocity slice** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Pressure slice` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Pressure slice** node, then click **Slice 1**.

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

3 In the **Expression** text field, type `p`.

Surface 1

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

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

3 In the **Expression** text field, type `p`.

Surface Slit 1

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

2 In the **Settings** window for **Surface Slit**, locate the **Expression on the Upside** section.

3 In the **Expression** text field, type `up(p)`.

4 Locate the **Expression on the Downside** section. In the **Expression** text field, type `down(p)`.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Expression** text field, type `1`.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 In the **Expression** text field, type `1`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

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

4 In the **Pressure slice** toolbar, click **Plot**.