Helmholtz Resonator with Flow: Interaction of Flow and Acoustics
Introduction

Helmholtz resonators are used in exhaust systems as they can attenuate a specific narrow frequency band. The presence of a flow in the system alters the acoustic properties of the resonator and hence the transmission loss of the sub-system. In this model example a Helmholtz resonator is located as a side branch to a main duct. The transmission loss through the main duct is investigated when a mean flow is introduced.

The mean background flow in the system is calculated with the SST turbulence model for Mach numbers $Ma = 0.05$ and $Ma = 0.1$. The acoustic problem is then solved using the Linearized Navier-Stokes, Frequency Domain (LNS) physics interface. The mean flow velocity, pressure, and turbulent viscosity is coupled to the LNS model. The model uses a mapping study to interpolate the flow solution from the CFD mesh onto the acoustics mesh.

The geometry dimensions of the system are taken from those presented in Ref. 1. The results can be compacted to the measurements found in the journal paper.

Note: This model requires both the Acoustics Module and the CFD Module.

Model Definition

The geometry and dimensions are the same as the ones presented in Ref. 1. The geometry consists of a man cylindrical duct with a diameter of 4.859 cm and a Helmholtz resonator attached to the side. The resonator has a neck of length 8.05 cm and diameter 4.044 cm. The volume of the resonator is $4501 \, \text{cm}^3$. The geometry is depicted in Figure 1. The symmetry of the system has been utilized.

The system is filled with air. A mean stationary background flow of average velocity $U_{in} = c_0 \cdot Ma$ enters at the left (downstream in the z-axis orientation), where $c_0$ is the speed of sound. Three flow configurations are studied with $Ma = 0$ (no flow), $Ma = 0.05$, and $Ma = 0.1$.

A plane acoustic wave enters the system downstream. The wave is added as a Background Acoustics Field feature in a small domain backed with the perfectly matched layer (PML). In this way reflected waves are free to leave the computational domain both downstream and upstream. A PML is also added at the upstream outlet. The background plane wave variables are defined according to
The plane wave expressions are imported into variables in the model. Since the acoustics are solved using the linearized Navier-Stokes equations, perturbations in pressure, velocity, and temperature need to be defined. These are also solved for.

\[
p_b = 1 \text{ Pa} \cdot e^{-i k_0 z} \\
k_0 = \frac{\omega}{c_0 + U_{in}}
\]

\[
\mathbf{u}_b = \left(0, 0, -\frac{1}{i \omega \rho_0} \frac{\partial p_b}{\partial z}\right)
\]

\[
T_b = \frac{\alpha_p T_0}{\rho_0 C_p} p_b
\]

The plane wave expressions are imported into variables in the model. Since the acoustics are solved using the linearized Navier-Stokes equations, perturbations in pressure, velocity, and temperature need to be defined. These are also solved for.

\[
\begin{align*}
T_L &= 20 \log_{10} \left( \frac{1 \text{ Pa}}{p_{out}} \right)
\end{align*}
\]

where \(p_{out}\) is the average pressure at the outlet and the incident field is defined with an amplitude of 1 Pa.

Two effects are important to capture correctly when analyzing this kind of system. It is the attenuation experience by the acoustic field as it interacts with turbulence (see Ref. 2 and Ref. 3) and the convective effects of the flow and the interaction with the flow gradients. The two phenomena have opposite effects on the resonance frequency of the system. The additional attenuation due to turbulence will shift the resonance towards a lower
frequency while the convective flow effects will shift it to a higher frequency. If the two are not correctly captured the results will be wrong. The transmission loss calculation is a good indicator that this is done correctly, this is seen in Figure 8 below. In order to capture the turbulence attenuation in detail, the SST turbulence model is used for the CFD as it introduces much less numerical diffusion and gives a better prediction of the eddy viscosity. The detailed interaction between flow and acoustics including the extra attenuation can only be modeled using the linearized Navier-Stokes equations as available in the Acoustics Module.

**Results and Discussion**

The magnitude of the mean background flow velocity is depicted in Figure 2 and the turbulent viscosity is depicted in Figure 3. The value of the turbulent viscosity should be compared to the dynamic viscosity of air \( \mu = 2 \times 10^{-5} \text{ Pa} \cdot \text{s} \). At the highest Mach number (depicted in the figure) the resulting experienced viscosity will be a factor 200 larger than the no-flow situation.

![Figure 2: Background mean flow velocity magnitude for Ma = 0.1.](image)
When a mapping study is used, it is good practice to visualize the mapped solution and compare it to the original solution on the CFD mesh. In Figure 4 the axial flow velocity and the turbulent viscosity evaluated on the CFD mesh is compared to the mapped values on the acoustics mesh.

Figure 3: Turbulent viscosity for Ma = 0.1.

Figure 4: Mapped value of the axial velocity (left) and turbulent (viscosity) compared to the value as evaluated on the CFD mesh.
The total acoustic pressure in the system is depicted at 200 Hz for the three cases, no flow in Figure 5, Ma = 0.05 in Figure 6, and Ma = 0.1 in Figure 7.

Figure 5: Acoustic pressure in the no flow situation (Ma = 0).

Figure 6: Acoustic pressure for Ma = 0.05.
Figure 7: Acoustic pressure for $Ma = 0.1$. 
Finally, the transmission loss of the system is depicted in Figure 8. The graph can be compared to measurements performed on the same system presented in Ref. 1. The results show good agreement.

![Figure 8: Transmission loss for the three flow configurations.](image)

**Notes About the COMSOL Implementation**

**Mapping Between CFD and Acoustics**

When the acoustics and the CFD models are not solved on the same computational mesh, careful mapping of the CFD solution from the CFD mesh onto the acoustics mesh should be done. This step is crucial in order not to introduce non-physical numerical noise into the acoustics solution. This can happen as the terms containing gradients of the background mean flow variables can become very noisy.

In this model two different mesh are used. In order to map the CFD solution a separate mapping study is used. The study solves an additional equation that maps and smooths the background flow variables. The mapping equations are set up using the **Weak Form PDE** interface from the Mathematics branch. In this model the mean background flow pressure $p_0$, velocity field $u_0$, and turbulent viscosity $\mu_T$ variables are mapped onto corresponding
variables on the acoustics mesh $p_{0,aco}$, $u_{0,aco}$, and $\mu_{T,aco}$. The mapping and smoothing is achieved by solving

$$\begin{align*}
p_{0,aco} - p_0 &= \delta h^2 \nabla \cdot (\nabla p_{0,aco}) \\
u_{i,0,aco} - u_{i,0} &= \delta h^2 \nabla \cdot (\nabla u_{i,0,aco}) \\
\mu_{T,aco} - \mu_T &= \delta h^2 \nabla \cdot (\nabla \mu_{T,aco})
\end{align*}$$

where the term on the right hand side adds smoothing using isotropic diffusion. The amount of diffusion is controlled by the parameter $\delta$ (a constant that can be tuned) and the mesh size squared $h^2$. The term in some sense corresponds to source term stabilization as known form CFD.

In the implementation in the model the value of the CFD variable is fetched from the CFD mesh using the `withsol()` operator. This is an extrusion coupling operator that can refer directly to a solution object and parameter value. It is important to use such an operator such that the CFD solution is mapped and interpolated correctly to the integration (Gauss) points on the acoustics mesh.

In some cases it can be necessary to extend the mapping equation with boundary conditions for the mapped variables. If, for example, the flow details near walls is important then add a no-slip condition on $u_{0,aco}$. This is not necessary in this model. But we do add a symmetry condition on the mapped velocity field at the symmetry plane using a constraint $u_{0,aco} \cdot n = 0$.

References


Application Library path: Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow

Modeling Instructions

The following model consists of three parts. First the background mean flow is solved using the CFD module, then the CFD solution is mapped to the acoustics mesh using a user-defined mapping study, and then the acoustics is solved with the mapped flow as input.

The first step is to set up the CFD model and solve the flow for two Mach numbers. Running this part of the model can take 8 h.

From the File menu, choose New.

NEW
In the New window, click Model Wizard.

MODEL WIZARD
1 In the Model Wizard window, click 2D Axisymmetric.
3 Click Add.
4 Click Study.
5 In the Select Study tree, select Preset Studies>Stationary with Initialization.
6 Click Done.

GLOBAL DEFINITIONS
In the Model Builder window, collapse the Global Definitions node.

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

Start by setting up a 2D axisymmetric model that is used to solve a fully developed turbulent inlet profile that is used in the 3D model. This procedure is the same as in, for example, the Ultrasonic Flow Meter with Generic Time-of-Flight Configuration model found under the Ultrasound applications.

**GEOMETRY 1**

**Rectangle 1 (r1)**
1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type \( \text{Dmain}/2 \).
4. In the **Height** text field, type \( \text{L2Daxi} \).
5. Click **Build Selected**.

**DEFINITIONS**

**General Extrusion 1 (genext1)**
1. On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, type \( \text{genext}_\text{2Daxi} \) in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 3 only.
5. Locate the **Destination Map** section. In the **r-expression** text field, type \( \sqrt{x^2+y^2} \).
6. Clear the **z-expression** text field.
7. Locate the **Source** section. Select the **Use source map** check box.
8. Clear the **z?-expression** text field.

**MATERIALS**

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

**ADD MATERIAL**
1. Go to the **Add Material** window.
2. In the tree, select **Built-In>Air**.
3. Click **Add to Component 1**.
4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

**TURBULENT FLOW, SST (SPF)**

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, SST (spf).**
2 In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.
3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3).**
4 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section.
   In the **Velocity field** text field, type $u_2$.
5 In the **Velocity field components** table, enter the following settings:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$u_2$</td>
<td></td>
</tr>
<tr>
<td>$v_2$</td>
<td></td>
</tr>
<tr>
<td>$w_2$</td>
<td></td>
</tr>
</tbody>
</table>

6 In the **Pressure** text field, type $p_2$.
7 In the **Turbulent kinetic energy** text field, type $k_2$.
8 In the **Specific dissipation rate** text field, type $\omega_2$.
9 In the **Reciprocal wall distance** text field, type $G_2$.

**Inlet 1**

1 On 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 $U_{in}$.

**Outlet 1**

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2 Select Boundary 3 only.

**MESH 1**

**Distribution 1**

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2 Right-click **Mapped 1** and choose **Distribution**.
3 Select Boundary 3 only.
4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
5 From the **Distribution properties** list, choose **Predefined distribution type**.
6 In the **Number of elements** text field, type 30.
7 In the **Element ratio** text field, type 15.
8 Select the **Reverse direction** check box.

**Distribution 2**
1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 From the **Distribution properties** list, choose **Predefined distribution type**.
5 In the **Number of elements** text field, type 30.
6 In the **Element ratio** text field, type 15.

**Distribution 3**
1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 4 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type 200.
   Now, set up the 3D model that solves for the flow inside the Helmholtz resonator system.
5 Click **Build All**.

**ROOT**
1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

   The first step is to build the geometry of the Helmholtz resonator system. To save time you can **Insert Sequence** from file by referring to the *helmholtz_renoator_with_flow.mph* file in the applications folder of the COMSOL installation.

**GEOMETRY 2**
In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

**Cylinder 1 (cyl1)**
1 On 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 Dmain/2.
4 In the **Height** text field, type \( \text{Lin} + \text{Lout} + 2 \times \text{Lpml} \).

5 Locate the **Position** section. In the **z** text field, type \( -\text{Lin} - \text{Lpml} \).

6 Click to expand the **Layers** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer name</th>
<th>Thickness (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 1</td>
<td>Lpml</td>
</tr>
</tbody>
</table>

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

*Cylinder 2 (cyl2)*

1 On 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 \( \text{Dneck}/2 \).

4 In the **Height** text field, type \( 1.2 \times \text{Lneck} \).

5 Locate the **Position** section. In the **x** text field, type \( \text{Dmain} / 2 - 0.2 \times \text{Lneck} \).

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

*Cylinder 3 (cyl3)*

1 On 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 \( \text{Dreson}/2 \).

4 In the **Height** text field, type \( \text{Lreson} \).

5 Locate the **Position** section. In the **x** text field, type \( \text{Dmain} / 2 + \text{Lneck} \).

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

*Cylinder 4 (cyl4)*

1 On 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 \( \text{Dmain}/2 \).

4 In the **Height** text field, type \( \text{Lsource} \).

5 Locate the **Position** section. In the **z** text field, type \( -\text{Lin} \).

*Cylinder 5 (cyl5)*

1 On 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 Dmain/2.
4 In the Height text field, type 0.2.
5 Locate the Position section. In the z text field, type -0.1.
6 Click Build Selected.

Partition Domains 1 (pard1)
1 On the Geometry toolbar, click Booleans and Partitions and choose Partition Domains.
2 On the object cyl2, select Domain 1 only.
3 In the Settings window for Partition Domains, locate the Partition Domains section.
4 From the Partition with list, choose Extended faces.
5 On the object cyl1, select Boundaries 12 and 15 only.
6 From the Repair tolerance list, choose Relative.
7 Click Build All Objects.
8 Click the Wireframe Rendering button on the Graphics toolbar.

Delete Entities 1 (del1)
1 In the Model Builder window, right-click Geometry 2 and choose Delete Entities.
2 In the Settings window for Delete Entities, locate the Entities or Objects to Delete section.
3 From the Geometric entity level list, choose Domain.
4 On the object pard1, select Domain 1 only.
5 Click Build Selected.

Union 1 (uni1)
1 On the Geometry toolbar, click Booleans and Partitions and choose Union.
2 Click in the Graphics window and then press Ctrl+A to select all objects.

Work Plane 1 (wp1)
1 On 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 zx-plane.

Partition Objects 1 (par1)
1 On the Geometry toolbar, click Booleans and Partitions and choose Partition Objects.
2 Select the object uni1 only.
3 In the Settings window for Partition Objects, locate the Partition Objects section.
4 From the **Partition with** list, choose **Work plane**.

5 Click **Build Selected**.

**Delete Entities 2 (del2)**

1 Right-click **Geometry 2** and choose **Delete Entities**.

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

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

4 On the object **par1**, select Domains 2, 4, 6, 8, 10, 12, 14, and 16 only.

5 Click **Build All Objects**.

Rotate the geometry in the **Graphics** window. The geometry should look like the figure above.

**ADD PHYSICS**

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Recently Used>Turbulent Flow, SST (spf)**.

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

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.
**TURBULENT FLOW, SST 2 (SPF2)**

1. In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.
2. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

**DEFINITIONS**

*Explicit 1*

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, locate the **Input Entities** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 2, 5, 8, 11, 14, 17, 29, and 34 only.
5. In the **Label** text field, type **Symmetry**.

*Explicit 2*

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, locate the **Input Entities** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 1, 4, 7, 10, 13, 16, 20–25, 27, 28, 30–32, and 35 only.
5. In the **Label** text field, type **Walls**.

**MATERIALS**

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

**ADD MATERIAL**

1. Go to the **Add Material** window.
2. In the tree, select **Built-In>Air**.
3. Click **Add to Component** in the window toolbar.
4. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

**TURBULENT FLOW, SST 2 (SPF2)**

*Symmetry 1*

1. On 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**.
Outlet 1
1 On the Physics toolbar, click Boundaries and choose Outlet.
2 Select Boundary 19 only.

Inlet 1
1 On the Physics toolbar, click Boundaries and choose Inlet.
2 Select Boundary 3 only.
3 In the Settings window for Inlet, locate the Velocity section.
4 Click the Velocity field button.
5 Specify the $u_0$ vector as

<table>
<thead>
<tr>
<th>Expression</th>
<th>Field</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{comp1.genext}_2\text{Daxi}(u_2)\cos(\text{atan2}(y,x))$</td>
<td>$x$</td>
</tr>
<tr>
<td>$\text{comp1.genext}_2\text{Daxi}(u_2)\sin(\text{atan2}(y,x))$</td>
<td>$y$</td>
</tr>
<tr>
<td>$\text{comp1.genext}_2\text{Daxi}(w_2)$</td>
<td>$z$</td>
</tr>
</tbody>
</table>

6 Locate the Turbulence Conditions section. Click the Specify turbulence variables button.
7 In the $k_0$ text field, type $\text{comp1.genext}_2\text{Daxi}(k_2)$.
8 In the $\omega_0$ text field, type $\text{comp1.genext}_2\text{Daxi}(\omega_2)$.

Generate the mesh used for the CFD simulation. In this model we set up a user-defined mesh consisting of hexahedral elements in the main part of the duct. Such a mesh will give the smallest amount of numerical diffusion. The turbulent viscosity will yield a good estimate of the eddy viscosity used in the acoustics model.

Mesh 2
1 In the Model Builder window, under Component 2 (comp2) click Mesh 2.
2 In the Settings window for Mesh, type Mesh - CFD in the Label text field.
3 Right-click Component 2 (comp2)>Mesh - CFD and choose More Operations>Free Quad.

Mesh - CFD

Free Quad 1
Select Boundaries 3 and 19 only.

Size 1
1 Right-click Component 2 (comp2)>Mesh - CFD>Free Quad 1 and choose Size.
2 In the Settings window for Size, locate the Element Size section.
3 From the Calibrate for list, choose Fluid dynamics.
4 From the Predefined list, choose Extremely fine.

**Size**
1 In the Model Builder window, under Component 2 (comp2)>Mesh - CFD click Size.
2 In the Settings window for Size, locate the Element Size section.
3 From the Calibrate for list, choose Fluid dynamics.
4 Click the Custom button.
5 Locate the Element Size Parameters section. In the Maximum element size text field, type 0.005.
6 In the Minimum element size text field, type 0.001.
7 In the Maximum element growth rate text field, type 1.1.
8 In the Curvature factor text field, type 0.4.
9 In the Resolution of narrow regions text field, type 1.

**Swept 1**
1 In the Model Builder window, right-click Mesh - CFD and choose Swept.
2 In the Settings window for Swept, locate the Domain Selection section.
3 From the Geometric entity level list, choose Domain.
4 Select Domains 1 and 2 only.

**Distribution 1**
1 Right-click Component 2 (comp2)>Mesh - CFD>Swept 1 and choose Distribution.
2 Select Domain 1 only.
3 In the Settings window for Distribution, locate the Distribution section.
4 In the Number of elements text field, type 8.

**Distribution 2**
1 Right-click Swept 1 and choose Distribution.
2 Select Domain 2 only.

**Swept 2**
1 In the Model Builder window, right-click Mesh - CFD and choose Swept.
2 In the Settings window for Swept, locate the Domain Selection section.
3 From the Geometric entity level list, choose Domain.
4 Select Domain 3 only.
**Distribution 1**
2. In the `Settings` window for `Distribution`, locate the `Distribution` section.
3. From the `Distribution properties` list, choose `Predefined distribution type`.
4. In the `Number of elements` text field, type 50.
5. In the `Element ratio` text field, type 4.
6. Select the `Reverse direction` check box.

**Swept 3**
1. In the `Model Builder` window, right-click `Mesh - CFD` and choose `Swept`.
2. In the `Settings` window for `Swept`, locate the `Domain Selection` section.
3. From the `Geometric entity level` list, choose `Domain`.
4. Select Domain 6 only.

**Distribution 1**
2. In the `Settings` window for `Distribution`, locate the `Distribution` section.
3. In the `Number of elements` text field, type 8.

**Swept 4**
1. In the `Model Builder` window, right-click `Mesh - CFD` and choose `Swept`.
2. In the `Settings` window for `Swept`, locate the `Domain Selection` section.
3. From the `Geometric entity level` list, choose `Domain`.
4. Select Domain 5 only.

**Distribution 1**
2. In the `Settings` window for `Distribution`, locate the `Distribution` section.
3. From the `Distribution properties` list, choose `Predefined distribution type`.
4. In the `Number of elements` text field, type 50.
5. In the `Element ratio` text field, type 4.
6. Select the `Reverse direction` check box.

**Corner Refinement 1**
1. In the `Model Builder` window, right-click `Mesh - CFD` and choose `Corner Refinement`.
2. In the `Settings` window for `Corner Refinement`, locate the `Boundary Selection` section.
3 From the Selection list, choose Walls.

Size 1
1 Right-click Mesh - CFD and choose Free Tetrahedral.
2 In the Model Builder window, under Component 2 (comp2)>Mesh - CFD right-click Free Tetrahedral 1 and choose Size.
3 In the Settings window for Size, locate the Geometric Entity Selection section.
4 From the Geometric entity level list, choose Boundary.
5 Select Boundary 26 only.
6 Locate the Element Size section. Click the Custom button.
7 Locate the Element Size Parameters section. Select the Maximum element size check box.
8 In the associated text field, type 0.0015.

Boundary Layer Properties
1 In the Model Builder window, right-click Mesh - CFD and choose Boundary Layers.
2 In the Settings window for Boundary Layer Properties, locate the Boundary Selection section.
3 From the Selection list, choose Walls.
4 Locate the Boundary Layer Properties section. In the Thickness adjustment factor field, type 1.4.
5 Click **Build All**.

The CFD mesh should look like the figure above.

6 Click **Build Selected**.

**STUDY 1**

1 In the **Settings** window for **Study**, type **Study 1 - CFD** in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

**STUDY 1 - CFD**

**Step 2: Stationary**

1 In the **Model Builder** window, under **Study 1 - CFD** click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics interface</th>
<th>Solve for</th>
<th>Discretization</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST 2 (spf2)</td>
<td></td>
<td>physics</td>
</tr>
</tbody>
</table>

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
5 From the Method list, choose Solution.
6 From the Study list, choose Study 1 - CFD, Wall Distance Initialization.

**Step 3: Stationary 2**
1 On the Study toolbar, click Study Steps and choose Stationary>Stationary.
2 In the Settings window for Stationary, locate the Physics and Variables Selection section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics interface</th>
<th>Solve for</th>
<th>Discretization</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST (spf)</td>
<td></td>
<td>physics</td>
</tr>
</tbody>
</table>

4 Click to expand the Values of dependent variables section. Locate the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
5 From the Method list, choose Solution.
6 From the Study list, choose Study 1 - CFD, Stationary.

**Parametric Sweep**
1 On 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:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
<th>Parameter unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>0.05 0.1</td>
<td></td>
</tr>
</tbody>
</table>

**Solution 1 (sol1)**
1 On the Study toolbar, click Show Default Solver.

After generating the solver sequence you can inspect the setup and make changes to the defaults if necessary.

Note that solving the model can take 5 h per value of the Mach number parameter (depending on your hardware).
2 Click Compute.

**RESULTS**
In the Model Builder window, expand the Results node.
Cut Line 3D

1. On the Results toolbar, click Cut Line 3D.
2. In the Settings window for Cut Line 3D, locate the Data section.
3. From the Data set list, choose Study 1 - CFD/Parametric Solutions 1 (8) (sol4).
4. Locate the Line Data section. In row Point 1, set x to 0.
5. In row Point 2, set x to 0.
6. In row Point 1, set y to 0.
7. In row Point 2, set y to 0.
8. In row Point 1, set z to -1.
9. In row Point 2, set z to 1.

3D Plot Group 1

1. On the Results toolbar, click 3D Plot Group.
2. In the Settings window for 3D Plot Group, type CFD: Velocity in the Label text field.
3. Locate the Data section. From the Data set list, choose Study 1 - CFD/Parametric Solutions 1 (8) (sol4).

Surface 1

1. Right-click CFD: Velocity and choose Surface.
2. On the CFD: Velocity toolbar, click Plot.
   The plot should reproduce Figure 2.

3D Plot Group 2

1. On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
2. In the Settings window for 3D Plot Group, type CFD: Turbulent Viscosity in the Label text field.
3. Locate the Data section. From the Data set list, choose Study 1 - CFD/Parametric Solutions 1 (8) (sol4).

Surface 1

1. Right-click CFD: Turbulent Viscosity and choose Surface.
2. In the Settings window for Surface, locate the Expression section.
3. In the Expression text field, type spf2.muT.
4. On the CFD: Turbulent Viscosity toolbar, click Plot.
   The plot should reproduce Figure 3.
1D Plot Group 3
1 On the Home toolbar, click Add Plot Group and choose 1D Plot Group.
2 In the Settings window for 1D Plot Group, type CFD: Turbulent Viscosity Axial in the Label text field.
3 Locate the Data section. From the Data set list, choose Study 1 - CFD/Parametric Solutions 1 (7) (sol4).

Line Graph 1
1 Right-click CFD: Turbulent Viscosity Axial and choose Line Graph.
2 Select Boundary 1 only.
3 In the Settings window for Line Graph, locate the y-Axis Data section.
4 In the Expression text field, type spf.muT.
5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
6 In the Expression text field, type z-Lin-Lpml-100*Dmain.

Line Graph 2
1 In the Model Builder window, under Results right-click CFD: Turbulent Viscosity Axial and choose Line Graph.
2 In the Settings window for Line Graph, locate the Data section.
3 From the Data set list, choose Cut Line 3D 1.
4 Locate the y-Axis Data section. In the Expression text field, type spf2.muT.
5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
6 In the Expression text field, type z.
On the CFD: Turbulent Viscosity Axial toolbar, click **Plot**.

The plot shows the turbulent viscosity along the central axis of the model including the 2D axi inlet section. Use the plot to inspect the convergence of the turbulent viscosity.

**Mapping Model**

An important part of solving aeroacoustic problems is to correctly map the CFD solution onto the acoustics domain. In the following set up a user defined equation that maps the solution from the CFD mesh onto the acoustics mesh. In this step a small amount of smoothing is used. The procedure is described in detail in the main part of the model documentation.

**COMPONENT 2 (COMP2)**

In the Model Builder window, click **Component 2 (comp2)**.

**ADD PHYSICS**

1. On the Home toolbar, click **Add Physics** to open the Add Physics window.
2. Go to the Add Physics window.
3. In the tree, select Mathematics>PDE Interfaces>Weak Form PDE (w).
4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Studies</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Study 1 - CFD</td>
<td></td>
</tr>
</tbody>
</table>

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Field name** text field, type `U0`.

6 In the **Number of dependent variables** text field, type 5.

7 In the **Dependent variables** table, enter the following settings:

<table>
<thead>
<tr>
<th>P0</th>
</tr>
</thead>
<tbody>
<tr>
<td>U0</td>
</tr>
<tr>
<td>V0</td>
</tr>
<tr>
<td>W0</td>
</tr>
<tr>
<td>MU_eff0</td>
</tr>
</tbody>
</table>

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

9 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

**WEAK FORM PDE (W)**

1 In the **Settings** window for **Weak Form PDE**, type **Weak Form PDE: Mapping** in the **Label** text field.

2 In the **Model Builder** window’s toolbar, click the **Show** button and select **Discretization** in the menu.

3 Click to expand the **Discretization** section. From the **Element order** list, choose **Linear**.

**WEAK FORM PDE: MAPPING (W)**

1 In the **Model Builder** window, under **Component 2 (comp2)>Weak Form PDE: Mapping (w)**, click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text-field array, type `(P0-withsol('sol4',spf2.pA,setval(Ma, Ma)))*test(P0)+delta*h^2*(P0x*test(P0x)+P0y*test(P0y)+P0z*test(P0z))` on the first row.

4 In the weak text-field array, type `(U0-withsol('sol4',u,setval(Ma, Ma)))*test(U0)+delta*h^2*(U0x*test(U0x)+U0y*test(U0y)+U0z*test(U0z))` on the second row.
5 In the weak text-field array, type \((V_0 - \text{withsol}'sol4', v, \text{setval}(Ma, Ma)))*
test(V0)+\text{delta}\cdot h^2\cdot(V0x\cdot test(V0x)+V0y\cdot test(V0y)+V0z\cdot test(V0z))\) on the
third row.

6 In the weak text-field array, type \((W_0 - \text{withsol}'sol4', w, \text{setval}(Ma, Ma)))*
test(W0)+\text{delta}\cdot h^2\cdot(W0x\cdot test(W0x)+W0y\cdot test(W0y)+W0z\cdot test(W0z))\) on the
fourth row.

7 In the weak text-field array, type \((\mu_{\text{eff}0} - \text{withsol}'sol4', \text{spf2.mu_eff,}
\text{setval}(Ma, Ma)))*test(\mu\_\text{eff}0)+\text{delta}\cdot h^2\cdot(\mu\_\text{eff}0x\cdot test(\mu\_\text{eff}0x)+\mu\_\text{eff}0y\cdot test(\mu\_\text{eff}0y)+\mu\_\text{eff}0z\cdot test(\mu\_\text{eff}0z))\) on the fifth row.

Make sure to enforce symmetry in the mapped velocity field by setting \(V0 = 0\) on the
symmetry plane. Do this by using a constraint.

\textit{Constraint 1}

1 On the \texttt{Physics} toolbar, click \texttt{Boundaries} and choose \texttt{Constraint}.

2 In the \texttt{Settings} window for \texttt{Constraint}, locate the \texttt{Boundary Selection} section.

3 From the \texttt{Selection} list, choose \texttt{Symmetry}.

4 Locate the \texttt{Constraint} section. In the \texttt{R} text-field array, type \(V0\) on the first row.

Create the acoustics mesh that the CFD solution is mapped onto. The mesh can be
relatively coarse. At the walls we do not resolve the acoustic boundary layer since slip
(and adiabatic) conditions are used here. A single boundary layer is added to resolve the
background flow profile near the wall.

\textbf{COMPONENT 2 (COMP2)}

\textit{Mesh 3}

On the \texttt{Mesh} toolbar, click \texttt{Add Mesh}.

\textbf{MESH 3}

1 In the \texttt{Settings} window for \texttt{Mesh}, type \texttt{Mesh - Acoustics} in the \texttt{Label} text field.

2 Right-click \texttt{Component 2 (comp2)>Meshes>Mesh - Acoustics} and choose \texttt{Free Tetrahedral}.

\textbf{MESH - ACoustics}

\textit{Size}

1 In the \texttt{Settings} window for \texttt{Size}, locate the \texttt{Element Size} section.

2 Click the \texttt{Custom} button.

3 Locate the \texttt{Element Size Parameters} section. In the \texttt{Maximum element size} text field, type
\(\text{Domain}/5\).
4 In the **Minimum element size** text field, type Dmain/15.

**Free Tetrahedral 1**
1 In the **Model Builder** window, under Component 2 (comp2)>Meshes>Mesh - Acoustics click Free Tetrahedral 1.
2 In the **Settings** window for Free Tetrahedral, locate the **Domain Selection** section.
3 From the Geometric entity level list, choose Domain.
4 Select Domains 2–5, 7, and 8 only.

**Distribution 1**
1 In the **Model Builder** window, right-click Mesh - Acoustics and choose Swept.
2 Right-click Swept 1 and choose Distribution.
3 In the **Settings** window for Distribution, locate the Distribution section.
4 In the **Number of elements** text field, type 10.
5 Click Build Selected.

**Boundary Layers 1**
1 In the **Model Builder** window, right-click Mesh - Acoustics and choose Boundary Layers.
2 In the **Settings** window for Boundary Layers, click to expand the Transition section.
3 Clear the Smooth transition to interior mesh check box.

**Boundary Layer Properties**
1 In the **Model Builder** window, under Component 2 (comp2)>Meshes>Mesh - Acoustics> Boundary Layers 1 click Boundary Layer Properties.
2 In the **Settings** window for Boundary Layer Properties, locate the Boundary Selection section.
3 From the Selection list, choose Walls.
4 Locate the Boundary Layer Properties section. In the **Number of boundary layers** text field, type 1.
5 From the **Thickness of first layer** list, choose Manual.
6 In the **Thickness** text field, type Dmain/16.
7 Click **Build All**.

---

The acoustics mesh should look like the figure above.

**ROOT**

On the **Home** toolbar, click **Windows** and choose **Add Study**.

**ADD STUDY**

1. Go to the **Add Study** window.

2. Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST (spf)</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, SST 2 (spf2)</td>
<td></td>
</tr>
</tbody>
</table>

3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4. Click **Add Study** in the window toolbar.

5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
**STUDY 2**

*Step 1: Stationary*
1. In the Model Builder window, click **Study 2**.
2. In the **Settings** window for **Study**, type **Study 2 - Mapping** in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

*Parametric Sweep*
On the **Study** toolbar, click **Parametric Sweep**.

**STUDY 2 - MAPPING**

*Parametric Sweep*
1. In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
2. Click **Add**.
3. Click to select row number 1 in the table.
4. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
<th>Parameter unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>0.05 0.1</td>
<td></td>
</tr>
</tbody>
</table>

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

Create two plots that compare the original CFD solution (evaluated on the CFD mesh) with the mapped solution (evaluated on the acoustics mesh).

**RESULTS**

*1D Plot Group 4*
1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type **Mapping: Comparison w** in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Study 1 - CFD/Parametric Solutions 1 (8) (sol4)**.

*Line Graph 1*
1. Right-click **Mapping: Comparison w** and choose **Line Graph**.
2. In the **Settings** window for **Line Graph**, locate the **Selection** section.
3. Select the **Active** toggle button.
4. Select Edge 12 only.
5 Locate the y-Axis Data section. In the Expression text field, type \( w \).

**Line Graph 2**
1 In the Model Builder window, under Results right-click Mapping: Comparison w and choose Line Graph.
2 In the Settings window for Line Graph, locate the Data section.
3 From the Data set list, choose Study 2 - Mapping/Solution 7 (10) (sol7).
4 Select Edge 12 only.
5 Locate the y-Axis Data section. In the Expression text field, type \( W_0 \).
6 On the Mapping: Comparison w toolbar, click Plot.

The plot should reproduce Figure 4 (left).

**Mapping: Comparison w 1**
1 Right-click Mapping: Comparison w and choose Duplicate.
2 In the Settings window for 1D Plot Group, type Mapping: Comparison mu_eff in the Label text field.

**Line Graph 1**
1 In the Model Builder window, expand the Mapping: Comparison w 1 node, then click Results>Mapping: Comparison mu_eff>Line Graph 1.
2 In the Settings window for Line Graph, locate the y-Axis Data section.
3 In the Expression text field, type spf2.mu_eff.

**Line Graph 2**
1 In the Model Builder window, under Results>Mapping: Comparison mu_eff click Line Graph 2.
2 In the Settings window for Line Graph, locate the y-Axis Data section.
3 In the Expression text field, type MU_eff0.
4 On the Mapping: Comparison mu_eff toolbar, click Plot.

The plot should reproduce Figure 4 (right).

**Acoustics Model**

Finally, set up the acoustics model solving the Linearized Navier-Stokes equations for the no-flow situation and for the background mean flow with \( Ma = 0.05 \) and \( Ma = 0.1 \).
ADD PHYSICS
1. On the Home toolbar, click Add Physics to open the Add Physics window.
2. Go to the Add Physics window.
3. In the tree, select Acoustics>Aeroacoustics>Linearized Navier-Stokes, Frequency Domain (lnsf).
4. Find the Physics interfaces in study subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Studies</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Study 1 - CFD</td>
<td></td>
</tr>
<tr>
<td>Study 2 - Mapping</td>
<td></td>
</tr>
</tbody>
</table>

5. Click Add to Component in the window toolbar.
6. On the Home toolbar, click Add Physics to close the Add Physics window.

DEFINITIONS
In the Model Builder window, expand the Definitions node.

Variables 1
1. Right-click Component 2 (comp2)>Definitions and choose Variables.
2. In the Settings window for Variables, type Variables: Plane Wave in the Label text field.
3. Locate the Variables section. Click Load from File.
4. Browse to the model’s Application Libraries folder and double-click the file helmholtz_resonator_with_flow_variables.txt.

These variables define a plane wave moving in the positive z-direction including the effect of a uniform flow. This wave is used as the source of the model.

Integration 1 (intop1)
1. On the Definitions toolbar, click Component 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. Select Boundary 6 only.
5. In the Operator name text field, type intop_in.

Integration 2 (intop2)
1. On the Definitions toolbar, click Component 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 Select Boundary 18 only.
5 In the Operator name text field, type intop_out.

**Perfectly Matched Layer 1 (pml1)**
1 On the Definitions toolbar, click Perfectly Matched Layer.
2 Select Domains 1 and 6 only.

**LINEARIZED NAVIER-STOKES, FREQUENCY DOMAIN (LNSF)**

Linearized Navier-Stokes Model 1
1 In the Model Builder window, under Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf) click Linearized Navier-Stokes Model 1.
2 In the Settings window for Linearized Navier-Stokes Model, locate the Model Input section.
3 In the \( p_0 \) text field, type \( P_0 \,[Pa] \).
4 Specify the \( u_0 \) vector as

\[
\begin{align*}
U_0 \,[m/s] & \quad x \\
V_0 \,[m/s] & \quad y \\
W_0 \,[m/s] & \quad z
\end{align*}
\]

5 Locate the Fluid Properties section. From the \( \mu \) list, choose User defined. In the associated text field, type \( \mu_{eff0} \).

Use the From speed of sound option for both the thermal expansion and the compressibility.
6 Locate the Thermal Expansion and Compressibility section. From the \( \alpha_p \) list, choose From speed of sound.
7 From the \( \beta_T \) list, choose From speed of sound.

**MATERIALS**

Air (mat2)
1 In the Model Builder window, expand the Component 2 (comp2)>Materials node, then click Air (mat2).
2 In the Settings window for Material, locate the Material Contents section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bulk viscosity</td>
<td>muB</td>
<td>0</td>
<td>Pa·s</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**LINEARIZED NAVIER-STOKES, FREQUENCY DOMAIN (LNSF)**

**Wall 1**
1 In the Model Builder window, under Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf) click Wall 1.
2 In the Settings window for Wall, locate the Mechanical section.
3 From the Mechanical condition list, choose Slip.
4 Locate the Thermal section. From the Thermal condition list, choose Adiabatic.

**Symmetry 1**
1 On 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.

**Background Acoustic Fields 1**
1 On the Physics toolbar, click Domains and choose Background Acoustic Fields.
2 Select Domain 2 only.
3 In the Settings window for Background Acoustic Fields, locate the Background Acoustic Fields section.
4 In the $p_b$ text field, type $p_b$.
5 Specify the $u_b$ vector as

<table>
<thead>
<tr>
<th>$u_b$</th>
<th>$x$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_b$</td>
<td>$y$</td>
</tr>
<tr>
<td>$w_b$</td>
<td>$z$</td>
</tr>
</tbody>
</table>

6 In the $T_b$ text field, type $T_b$.

**Linearized Navier-Stokes Model 2**
1 On the Physics toolbar, click Domains and choose Linearized Navier-Stokes Model.
2 In the Settings window for Linearized Navier-Stokes Model, type Linearized Navier-Stokes Model: No Flow in the Label text field.
3 Locate the Domain Selection section. From the Selection list, choose All domains.
4 Locate the Thermal Expansion and Compressibility section. From the $\alpha_p$ list, choose From speed of sound.

5 From the $\beta_T$ list, choose From speed of sound.

**ADD STUDY**

1 On the Home toolbar, click Add Study to open the Add Study window.

2 Go to the Add Study window.

3 Find the Physics interfaces in study subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST (spf)</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, SST 2 (spf2)</td>
<td></td>
</tr>
<tr>
<td>Weak Form PDE: Mapping (w)</td>
<td></td>
</tr>
</tbody>
</table>

4 Find the Studies subsection. In the Select Study tree, select Preset Studies> Frequency Domain.

5 Click Add Study in the window toolbar.

6 On the Home toolbar, click Add Study to close the Add Study window.

**STUDY 3**

*Step 1: Frequency Domain*

1 In the Settings window for Frequency Domain, locate the Study Settings section.

2 In the Frequencies text field, type range(50,10,200).

3 In the Model Builder window, click Study 3.

4 In the Settings window for Study, type Study 3 - Acoustics, Ma = 0 in the Label text field.

5 Locate the Study Settings section. Clear the Generate default plots check box.

6 On the Home toolbar, click Compute.

Remember to change the value of the Mach number parameter Ma to be 0.05 for the next simulation (it’s initial value is 0 as used for the previous study).

**GLOBAL DEFINITIONS**

*Parameters*

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

2 In the Settings window for Parameters, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>0.05</td>
<td>0.05</td>
<td>Flow Mach number</td>
</tr>
</tbody>
</table>

**ADD STUDY**

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST (spf)</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, SST 2 (spf2)</td>
<td></td>
</tr>
<tr>
<td>Weak Form PDE: Mapping (w)</td>
<td></td>
</tr>
</tbody>
</table>

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies** > **Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

**STUDY 4**

**Step 1: Frequency Domain**

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(50,10,200)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf)>Linearized Navier-Stokes Model: No Flow**.

5 Click **Disable**.

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 2 - Mapping, Stationary**.

9 From the **Parameter value (Ma)** list, choose 0.05.
In the Model Builder window, click Study 4.

In the Settings window for Study, type Study 4 - Acoustics, Ma = 0.05 in the Label text field.

Locate the Study Settings section. Clear the Generate default plots check box.

On the Home toolbar, click Compute.

Remember to change the value of the Mach number parameter Ma to be 0.1 for the next simulation (it’s value is 0.05 as used for the previous study).

GLOBAL DEFINITIONS

Parameters
1 In the Model Builder window, under Global Definitions click Parameters.
2 In the Settings window for Parameters, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>0.1</td>
<td>0.1</td>
<td>Flow Mach number</td>
</tr>
</tbody>
</table>

ADD STUDY
1 On the Home toolbar, click Add Study to open the Add Study window.
2 Go to the Add Study window.
3 Find the Physics interfaces in study subsection. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Physics</th>
<th>Solve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Flow, SST (spf)</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, SST 2 (spf2)</td>
<td></td>
</tr>
<tr>
<td>Weak Form PDE: Mapping (w)</td>
<td></td>
</tr>
</tbody>
</table>

4 Find the Studies subsection. In the Select Study tree, select Preset Studies>
Frequency Domain.
5 Click Add Study in the window toolbar.
6 On the Home toolbar, click Add Study to close the Add Study window.

STUDY 5

Step 1: Frequency Domain
1 In the Settings window for Frequency Domain, locate the Study Settings section.
2 In the **Frequencies** text field, type `range(50, 10, 200)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select Component 2 (comp2) > Linearized Navier-Stokes, Frequency Domain (lnsf) > Linearized Navier-Stokes Model: No Flow.

5 Click **Disable**.

6 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 2 - Mapping, Stationary**.

9 From the **Parameter value (Ma)** list, choose **0.1**.

   Under the **Mesh Selection** section you can check that the acoustics mesh is used.

10 Click to expand the **Mesh selection** section. In the **Model Builder** window, click **Study 5**.

11 In the **Settings** window for **Study**, type **Study 5 - Acoustics, Ma = 0.1** in the **Label** text field.

12 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

13 On the **Home** toolbar, click **Compute**.

**RESULTS**

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type **Acoustics: Pressure Ma = 0** in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics, Ma = 0/ Solution 8 (12) (sol8)**.

**Surface 1**

1 Right-click **Acoustics: Pressure Ma = 0** and choose **Surface**.

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

3 In the **Expression** text field, type `lnsf.p_t`.

**Selection 1**

1 Right-click **Results>Acoustics: Pressure Ma = 0>Surface 1** and choose **Selection**.

2 Select Boundaries 4, 5, 7–15, 21–24, and 26–35 only.
On the Acoustics: Pressure Ma = 0 toolbar, click Plot.

The plot should reproduce Figure 5.

Acoustics: Pressure Ma = 0.1
1 In the Model Builder window, under Results right-click Acoustics: Pressure Ma = 0 and choose Duplicate.
2 In the Settings window for 3D Plot Group, type Acoustics: Pressure Ma = 0.05 in the Label text field.
3 Locate the Data section. From the Data set list, choose Study 4 - Acoustics, Ma = 0.05/Solution 9 (14) (sol9).
4 On the Acoustics: Pressure Ma = 0.05 toolbar, click Plot.

The plot should reproduce Figure 6.

Acoustics: Pressure Ma = 0.05.1
1 Right-click Results>Acoustics: Pressure Ma = 0.05 and choose Duplicate.
2 In the Settings window for 3D Plot Group, type Acoustics: Pressure Ma = 0.1 in the Label text field.
3 Locate the Data section. From the Data set list, choose Study 5 - Acoustics, Ma = 0.1/Solution 10 (16) (sol10).
4 On the Acoustics: Pressure Ma = 0.1 toolbar, click Plot.

The plot should reproduce Figure 7.

1D Plot Group 9
1 On the Home toolbar, click Add Plot Group and choose 1D Plot Group.
2 In the Settings window for 1D Plot Group, type Transmission Loss in the Label text field.
3 Locate the Data section. From the Data set list, choose None.
4 Locate the Plot Settings section. Select the x-axis label check box.
5 In the associated text field, type f (Hz).
6 Select the y-axis label check box.
7 In the associated text field, type TL (dB).
8 Click to expand the Title section. From the Title type list, choose Manual.
9 In the Title text area, type Transmission Loss.
10 Locate the Axis section. Select the Manual axis limits check box.
11 In the y minimum text field, type 0.
In the **y maximum** text field, type 40.
In the **x minimum** text field, type 50.
In the **x maximum** text field, type 200.

**Global 1**
1 Right-click **Transmission Loss** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Data set** list, choose **Study 3 - Acoustics, Ma = 0/Solution 8 (12) (sol8)**.
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>20*log10(abs(intop_in(pb)/intop_out(lnsf.p_t)))</td>
<td></td>
<td>Ma = 0</td>
</tr>
</tbody>
</table>

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.
6 From the **Positioning** list, choose **In data points**.

**Global 2**
1 Right-click **Results>Transmission Loss>Global 1** and choose **Duplicate**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Data set** list, choose **Study 4 - Acoustics, Ma = 0.05/Solution 9 (14) (sol9)**.
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>20*log10(abs(intop_in(pb)/intop_out(lnsf.p_t)))</td>
<td></td>
<td>Ma = 0.05</td>
</tr>
</tbody>
</table>

**Global 3**
1 Right-click **Results>Transmission Loss>Global 2** and choose **Duplicate**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Data set** list, choose **Study 5 - Acoustics, Ma = 0.1/Solution 10 (16) (sol10)**.
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Expression</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>20*log10(abs(intop_in(pb)/intop_out(lnsf.p_t)))</td>
<td></td>
<td>Ma = 0.1</td>
</tr>
</tbody>
</table>
On the Transmission Loss toolbar, click Plot.
The plot should reproduce Figure 8.
The last thing to do is to disable the PMLs in the two non-acoustic studies. This is to make sure that, if the two studies are solved again then the results will be correct. The CFD interface is not supported inside domains with PMLs.

**STUDY 1 - CFD**

*Step 3: Stationary 2*

1. In the Model Builder window, under Study 1 - CFD click Step 3: Stationary 2.
2. In the Settings window for Stationary, locate the Physics and Variables Selection section.
3. Select the Modify model configuration for study step check box.
4. In the Physics and variables selection tree, select Component 2 (comp2)>Definitions>Perfectly Matched Layer 1 (pml1).
5. Click Disable.

**STUDY 2 - MAPPING**

*Step 1: Stationary*

1. In the Model Builder window, expand the Study 2 - Mapping node, then click Step 1: Stationary.
2. In the Settings window for Stationary, locate the Physics and Variables Selection section.
3. Select the Modify model configuration for study step check box.
4. In the Physics and variables selection tree, select Component 2 (comp2)>Definitions>Perfectly Matched Layer 1 (pml1).
5. Click Disable.