Created in COMSOL Multiphysics 6.2

Impedance Tube Parameter Estimation with Data Generation

Introduction

Impedance tubes are used for characterizing poroacoustic materials, usually with a focus on a frequency-dependent surface impedance. Alternatively, it is possible to assume a poroacoustic model and estimate the associated (frequency-independent) material parameters. This model uses a Johnson–Champoux–Allard material to generate synthetic data for two different meshes. This ensures that the data noise is dominated by the explicitly added noise rather than that of the mesh. The material parameters are taken from Ref. 1, and they are recovered based on the real and imaginary parts of the pressure at two probes (microphones). The generation of synthetic data and recovery of material parameters is a good sanity test, which can indicate the level of achievable accuracy given a certain experimental measurement error.

Model Definition

The model geometry (Figure 1) consists of two regions: the tube filled with air above a domain for the porous material (the test sample).

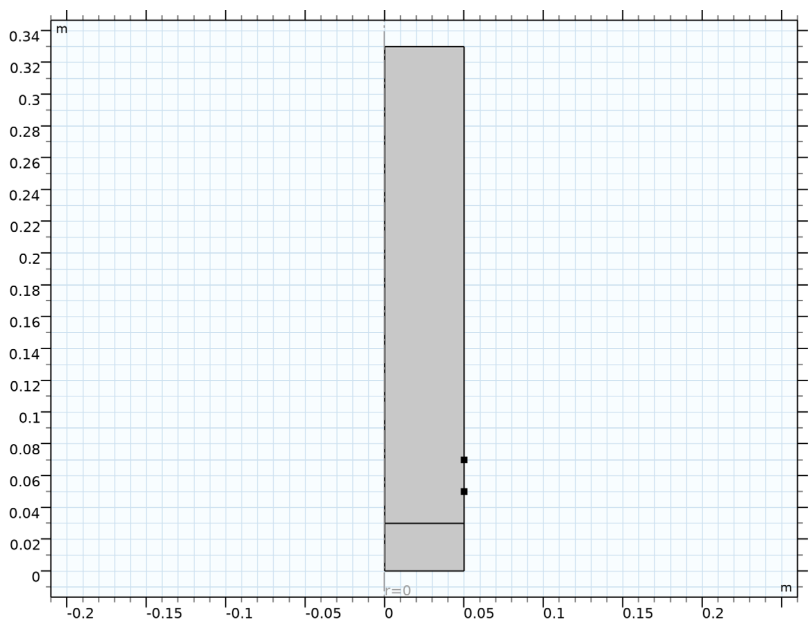

Figure 1: The model geometry.

The pressure in the top of the tube is described using a linear elastic pressure acoustics model, while the bottom is governed by the Johnson–Champoux–Allard model (using the Poroacoustics domain feature). The tube is exited using Normal Acceleration on the top boundary. The model makes use of axisymmetry and the azimuthal mode number is fixed at 0 as illustrated in Figure 2.

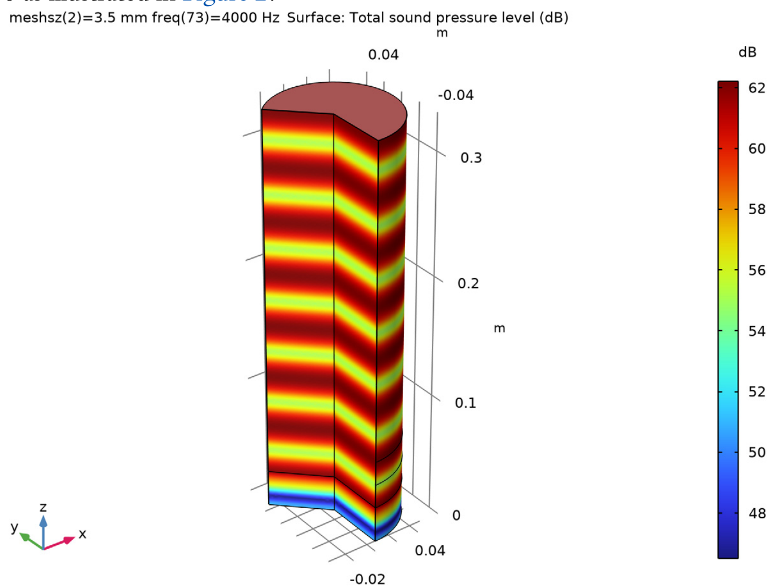

Figure 2: The sound pressure level is plotted for a frequency of 4 KHz.

Figure 3 shows the relative difference between using 18 and 36 elements per wavelength. Since the difference is smaller than 0.1 %, we add 0.5 % (normally distributed) noise to the data generated with 18 elements per wavelength.

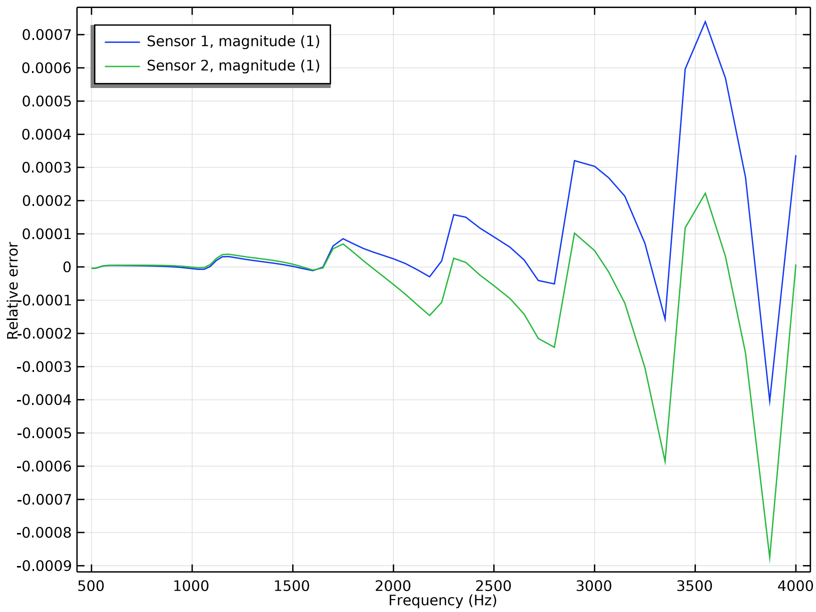

Figure 3: The relative difference between using 18 and 36 elements per (minimum) wavelength.

The relative errors on the estimated parameters are 0.0008%, 0.6%, 3.5%, 1.9%, and 0.2% for the flow resistivity, thermal characteristic length, viscous characteristic length, tortuosity factor, and porosity, respectively. These values will be larger if more noise is added to the generated data, but if less noise is added the accuracy of the estimation might be limited by the mesh resolution. This effect might not be observable unless different meshes are used for generating the data and recovering the parameters.

The generated pressure data with 1% noise is compared with the fitted values in Figure 4.

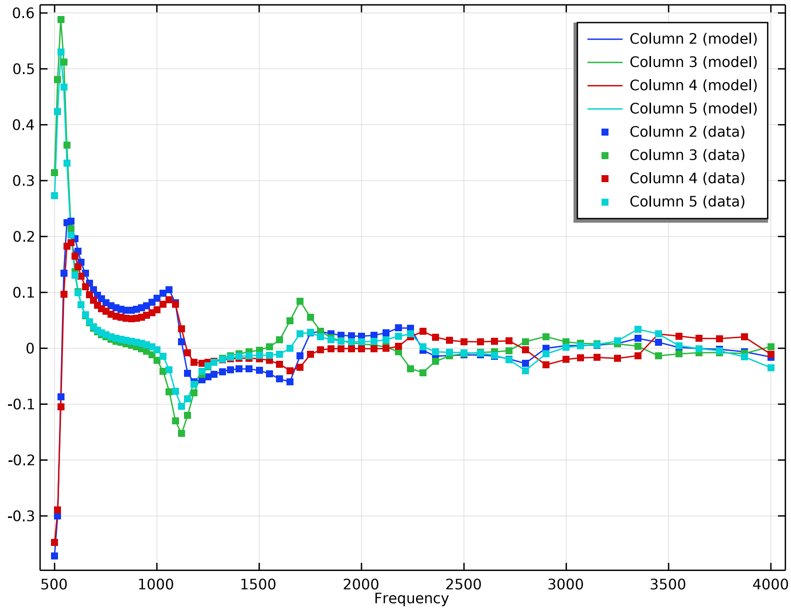

Figure 4: The real and imaginary parts of the pressure compared with the generated data for sensor 4.

Notes About the COMSOL Implementation

This model combines the Optimization and Pressure Acoustics interfaces. Splitting of complex variables into real and imaginary parts is enabled in the Compile Equations step in the optimization study. The generated data is imported using a Global Least-Squares Objective, and subnodes are added in the order corresponding to the column in the file. The Levenberg–Marquardt optimizer is used to solve the optimization problem with the frequencies automatically taken from the file. Global Evaluation groups as well as Global plots make use of the `withsol` operator to combine data from different solutions.

Reference

1. Multilayered Porous Material: Poroelastic Waves with Thermal and Viscous Losses (Biot-Allard Model), www.comsol.com/model/multilayered-porous-material-poroelastic-waves-with-thermal-and-viscous-losses-b-34531

Application Library path: `Optimization_Module/Parameter_Estimation/impedance_tube_parameter_estimation_data`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Add four parameter sets: One each for the geometry, the material, the initial guess material parameters, and miscellaneous parameters. Alternatively, you can load the parameters from the file `impedance_tube_parameter_estimation_data_parameters.txt` in the model's Application Libraries folder.

Geometrical Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometrical Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
L1	30[cm]	0.3 m	Length of tube
D1	10[cm]	0.1 m	Diameter of tube

Name	Expression	Value	Description
L2	3[cm]	0.03 m	Length of porous region
L3	2[cm]	0.02 m	Microphone spacing

Material Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Material Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
Rf0	87[kN*s/m^4]	87000 N·s/(m·m³)	Flow resistivity
Lth0	119[um]	1.19E-4 m	Thermal characteristic length
Lv0	37[um]	3.7E-5 m	Viscous characteristic length
tau0	2.52	2.52	Tortuosity factor
epsilon0	0.97	0.97	Porosity

Material Parameters (Initial Guess)

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Material Parameters (Initial Guess)` in the **Label** text field.

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

Name	Expression	Value	Description
Rf1	100[kN*s/m^4]	1E5 N·s/(m·m³)	Flow resistivity
Lth1	100[um]	1E-4 m	Thermal characteristic length
Lv1	100[um]	1E-4 m	Viscous characteristic length
tau1	1	1	Tortuosity factor
epsilon1	0.5	0.5	Porosity

Parameters 4

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
error	5e-3	0.005	Relative measurement error
fmin	500[Hz]	500 Hz	Minimum frequency
fmax	4[kHz]	4000 Hz	Maximum frequency
meshsz	340[m/s]/fmax/12	0.0070833 m	Mesh size

GEOMETRY 1

Use the parameters to define the geometry.

Rectangle 1 (r1)

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `D1/2`.
4. In the **Height** text field, type `L1`.
5. Locate the **Position** section. In the **z** text field, type `L2`.

Rectangle 2 (r2)

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `D1/2`.
4. In the **Height** text field, type `L2`.

Point 1 (pt1)

1. In the **Geometry** toolbar, click **Point**.
2. In the **Settings** window for **Point**, locate the **Point** section.
3. In the **r** text field, type `D1/2`.
4. In the **z** text field, type `L2+L3`.
5. Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Point 2 (pt2)

1. In the **Geometry** toolbar, click **Point**.
2. In the **Settings** window for **Point**, locate the **Point** section.
3. In the **r** text field, type `D1/2`.

4 In the **z** text field, type `L2+2*L3`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

7 In the **Graphics** window toolbar, click next to **Select Objects**, then choose **Select Points**.

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

Define a **Random** function to simulate noise. Then add probes for the two pressure sensors.

GLOBAL DEFINITIONS

Random 1 (rn1)

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

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

3 From the **Distribution** list, choose **Normal**.

DEFINITIONS

Microphone 1 (real)

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `Microphone 1 (real)` in the **Label** text field.

3 In the **Variable name** text field, type `mic1_real`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Point 1**.

5 Locate the **Expression** section. In the **Expression** text field, type `real(acpr.p_t)`.

Microphone 1 (imag)

1 Right-click **Microphone 1 (real)** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `Microphone 1 (imag)` in the **Label** text field.

3 In the **Variable name** text field, type `mic1_imag`.

4 Locate the **Expression** section. In the **Expression** text field, type `imag(acpr.p_t)`.

Microphone 2 (real)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Microphone 1 (real) (mic1_real)** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `Microphone 2 (real)` in the **Label** text field.

3 In the **Variable name** text field, type `mic2_real`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Point 2**.

Microphone 2 (imag)

1 Right-click **Microphone 2 (real)** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `Microphone 2 (imag)` in the **Label** text field.

3 In the **Variable name** text field, type `mic2_imag`.

4 Locate the **Expression** section. In the **Expression** text field, type `imag(acpr.p_t)`.

ADD MATERIAL

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

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

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

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

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

MATERIALS

Material 2 (mat2)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Add four parameter sets: One each for the geometry, the material, the initial guess material parameters, and miscellaneous parameters. Alternatively, you can load the parameters from the file `impedance_tube_parameter_estimation_data_parameters.txt` in the model's Application Libraries folder.

Poroacoustics 1

1 In the **Model Builder** window, expand the **Material 2 (mat2)** node.

2 Right-click **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** and choose **Poroacoustics**.

3 Select Domain 1 only.

4 In the **Settings** window for **Poroacoustics**, locate the **Poroacoustics Model** section.

5 From the **Poroacoustics model** list, choose **Johnson-Champoux-Allard (JCA)**.

6 Locate the **Porous Matrix Properties** section. From the **Porous elastic material** list, choose **Material 2 (mat2)**.

Normal Acceleration 1

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

2 Select Boundary 5 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `1`.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 2 (mat2)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	`epsilon0`	1	Basic
Viscous characteristic length	Lv	`Lv0`	m	Poroacoustics model
Thermal characteristic length	Lth	`Lth0`	m	Poroacoustics model
Tortuosity factor	tau	`tau0`	1	Poroacoustics model
Flow resistivity	Rf	`Rf0`	Pa·s/m²	Poroacoustics model

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz`.

5 In the **Minimum element size** text field, type `meshsz/2`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

GENERATE EXPERIMENTAL DATA

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

2 In the **Settings** window for **Study**, type `Generate Experimental Data` in the **Label** text field.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
meshsz (Mesh size)	7 3.5	mm

5 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, click to expand the **Results While Solving** section.

3 From the **Probes** list, choose **None**.

4 Locate the **Study Settings** section. Click **Range**.

5 In the **Range** dialog box, choose **ISO preferred frequencies** from the **Entry method** list.

6 In the **Start frequency** text field, type `fmin`.

7 In the **Stop frequency** text field, type `fmax`.

8 From the **Interval** list, choose **1/24 octave**.

9 Click **Replace**.

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

RESULTS

Sound Pressure Level, 3D (acpr)

1. In the **Model Builder** window, under **Results** click **Sound Pressure Level, 3D (acpr)**.
2. In the **Sound Pressure Level, 3D (acpr)** toolbar, click **Plot**.
3. Click the **Zoom Extents** button in the **Graphics** toolbar.

Acoustic Pressure (acpr), Acoustic Pressure, 3D (acpr), Sound Pressure Level (acpr), Sound Pressure Level, 3D (acpr)

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr)**, **Sound Pressure Level (acpr)**, **Acoustic Pressure, 3D (acpr)**, and **Sound Pressure Level, 3D (acpr)**.
2. Right-click and choose **Group**.

Experimental Data

In the **Settings** window for **Group**, type `Experimental Data` in the **Label** text field.

Experimental Data

1. In the **Results** toolbar, click **Evaluation Group**.
2. In the **Settings** window for **Evaluation Group**, type `Experimental Data` in the **Label** text field.

Generate the experimental data using the probes and the random error function. You can import the table contents from `impedance_tube_parameter_estimation_data_pressure.txt`.

Global Evaluation 1

1. Right-click **Experimental Data** and choose **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`mic1_real*(1+error*rn1(freq+0))`	`Pa`	`Sensor 1, real`
`mic1_imag*(1+error*rn1(freq+1))`	`Pa`	`Sensor 1, imaginary`
`mic2_real*(1+error*rn1(2*freq+0))`	`Pa`	`Sensor 2, real`
`mic2_imag*(1+error*rn1(2*freq+1))`	`Pa`	`Sensor 2, imaginary`

4. In the **Experimental Data** toolbar, click **Evaluate**.

The error due to the mesh should be smaller than the added error. To check if this is the case, investigate the effect of mesh refinement for the measured pressure.

Relative Errors (mesh)

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Generate Experimental Data/Parametric Solutions 1 (sol2)**.
4 From the **Parameter selection (meshsz)** list, choose **First**.
5 In the **Label** text field, type `Relative Errors (mesh)`.

Compute the relative errors of the measured pressures. Use the magnitude to avoid spikes when the real and imaginary parts go through zero.

Global Evaluation 1

1 Right-click **Relative Errors (mesh)** and choose **Global Evaluation**.
2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3 Click **Clear Table**.
4 Click **Load from File**.
5 Browse to the model's Application Libraries folder and double-click the file `impedance_tube_parameter_estimation_data_errors_mesh.txt`.
6 In the **Relative Errors (mesh)** toolbar, click **Evaluate**.

Errors

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Errors` in the **Label** text field.

Table Graph 1

1 Right-click **Errors** and choose **Table Graph**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Source** list, choose **Evaluation group**.
4 From the **Evaluation group** list, choose **Relative Errors (mesh)**.
5 From the **x-axis data** list, choose **freq (Hz)**.
6 From the **Plot columns** list, choose **Manual**.
7 In the **Columns** list, choose **Sensor 1, magnitude (1)** and **Sensor 2, magnitude (1)**.
8 Click to expand the **Legends** section. Select the **Show legends** check box.

Errors

1 In the **Model Builder** window, click **Errors**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `Frequency (Hz)`.

4 Select the **y-axis label** check box. In the associated text field, type `Relative error`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Errors** toolbar, click **Plot**.

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

The error due to the mesh is an order of magnitude smaller than the added error, so the mesh is sufficiently fine. It is thus safe to export the table.

Table 1

1 In the **Results** toolbar, click **Data** and choose **Table**.

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

3 From the **Source** list, choose **Evaluation group**.

4 Locate the **Output** section. In the **Filename** text field, type `impedance_tube_parameter_estimation_data.csv`.

5 Locate the **Layout** section. Clear the **Include header** check box.

6 Click **Export**.

ROOT

Add a second study for the parameter estimation.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

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

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

PARAMETER ESTIMATION

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

2 In the **Settings** window for **Study**, type `Parameter Estimation` in the **Label** text field.

Parameter Estimation

1 In the **Study** toolbar, click **Optimization** and choose **Parameter Estimation**.

2 In the **Settings** window for **Parameter Estimation**, locate the **Experimental Data** section.

3 In the **Filename** text field, type `impedance_tube_parameter_estimation_data.csv`.

4 Click **Import**.

5 Locate the **Column Settings** section. In the table, click to select the cell at row number 1 and column number 2.

6 In the table, click to select the cell at row number 2 and column number 3.

7 From the drop-down list, choose **Frequency**.

8 In the **Model expression** text field, type `comp1.mic1_real`.

9 In the **Variable name** text field, type `pressure1`.

10 In the **Unit** text field, type `Pa`.

11 In the table, click to select the cell at row number 3 and column number 3.

12 In the **Model expression** text field, type `comp1.mic1_imag`.

13 In the **Variable name** text field, type `pressure2`.

14 In the **Unit** text field, type `Pa`.

15 In the table, click to select the cell at row number 4 and column number 3.

16 In the **Model expression** text field, type `comp1.mic2_real`.

17 In the **Variable name** text field, type `pressure3`.

18 In the **Unit** text field, type `Pa`.

19 In the table, click to select the cell at row number 5 and column number 3.

20 In the **Model expression** text field, type `comp1.mic2_imag`.

21 In the **Variable name** text field, type `pressure4`.

22 In the **Unit** text field, type `Pa`.

23 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **Levenberg-Marquardt**.

24 Find the **Solver settings** subsection. From the **Least-squares time/parameter method** list, choose **Use only least-squares data points**.

25 Click to expand the **Output While Solving** section. From the **Probes** list, choose **None**.

26 Locate the **Column Settings** section. In the table, click to select the cell at row number 2 and column number 2.

27 Locate the **Estimated Parameters** section. Click **Add** five times.

28 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
epsilon0 (Porosity)	`epsilon1`	1		
Lth0 (Thermal characteristic length)	`Lth1`	`100[um]`		
Lv0 (Viscous characteristic length)	`Lv1`	`100[um]`		
Rf0 (Flow resistivity)	`Rf1`	`100[kN*s/m^4]`		
tau0 (Tortuosity factor)	`tau1`	1		

Solution 5 (sol5)

1 In the **Study** toolbar, click **Show Default Solver** to initialize the solver so that the splitting of complex variables can be enabled.

2 In the **Model Builder** window, expand the **Solution 5 (sol5)** node, then click **Compile Equations: Frequency Domain**.

3 In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.

4 Select the **Split complex variables in real and imaginary parts** check box.

5 In the **Model Builder** window, under **Parameter Estimation>Solver Configurations>Solution 5 (sol5)** click **Optimization Solver 1**.

6 In the **Settings** window for **Optimization Solver**, locate the **Optimization Solver** section.

7 From the **Gradient method** list, choose **Adjoint** to make the sensitivity analysis a bit faster.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Parameter Estimation** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 1

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

RESULTS

Acoustic Pressure (acpr) 1, Acoustic Pressure, 3D (acpr) 1, Parameter estimation, Sound Pressure Level (acpr) 1, Sound Pressure Level, 3D (acpr) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Acoustic Pressure (acpr) 1**, **Sound Pressure Level (acpr) 1**, **Acoustic Pressure, 3D (acpr) 1**, **Sound Pressure Level, 3D (acpr) 1**, and **Parameter estimation**.

2 Right-click and choose **Group**.

Parameter Estimation

In the **Settings** window for **Group**, type `Parameter Estimation` in the **Label** text field.

Add two 1D plot groups for comparing the noisy exported pressures with the fitted pressures.

Parameter estimation

1 In the **Model Builder** window, click **Parameter estimation**.

2 In the **Parameter estimation** toolbar, click **Plot**.

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

Relative Errors

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Relative Errors` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation/ Solution 5 (sol5)**.

4 From the **Parameter selection (freq)** list, choose **First**.

Compute the relative errors of the fitted material parameters. You can import the table contents from `impedance_tube_parameter_estimation_errors_parameters.txt`.

Global Evaluation 1

1 Right-click **Relative Errors** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(Rf0-withsol('sol1',Rf0))/withsol('sol1',Rf0)`	1	
`abs(Lth0-withsol('sol1',Lth0))/withsol('sol1',Lth0)`	1	
`abs(Lv0-withsol('sol1',Lv0))/withsol('sol1',Lv0)`	1	
`abs(tau0-withsol('sol1',tau0))/withsol('sol1',tau0)`	1	
`abs(epsilon0-withsol('sol1',epsilon0))/withsol('sol1',epsilon0)`	1	

4 In the **Relative Errors** toolbar, click **Evaluate**.