

GETTING STARTED WITH KNOWLES SISONIC™ SIMULATION MODELS



Knowles' SiSonic™ datasheets contain the performance specifications for raw microphones in a free field environment. However, the acoustic performance of consumer electronic applications using MEMS microphones is always impacted by the product's Industrial Design or ID.

The acoustic path leading from the microphone to the outside world, commonly referred to as porting, represents an acoustic load unique to every ID. This load on the acoustic path impacts the sound present at the microphone input. This, in turn, impacts the microphone output which is a combination of the specific microphone's performance and the acoustic load of the ID.

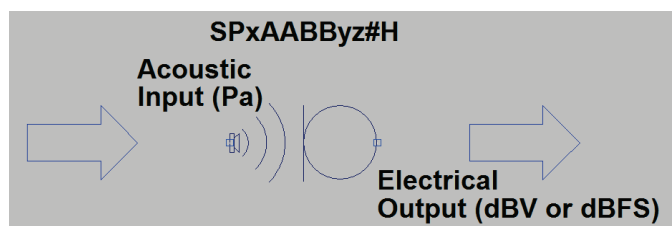
Designers can use models provided by Knowles to simulate the full acoustic performance, including gaskets, meshes, and Knowles SiSonic™ MEMS microphones—all of the components comprising the acoustic path. This document describes the Knowles simulation models, how to use them, and how they can be customized to estimate the response of the microphone in actual applications. While not specifically addressed in detail by this application note, the impacts of tubes, cavities, meshes, and manufacturing processes are among the many design considerations that can be simulated.

SIMULATION MODEL INTENTIONS

Knowles' SiSonic™ Microphone models are intended to assist in the initial product design when integrating MEMS microphones. Simulations help provide an understanding of the acoustic performance of the application under development. They help analyze the impact of the resonances and attenuation from porting. It is assumed that the porting design is robust and does not leak. Mechanisms for robust mechanical sealing are outside the scope of this document. Models are valid in the audio frequency range.

Knowles simulation models abstract the acoustic path as electrical components and are otherwise known as Equivalent Circuit Models. The input to the model is a voltage representing the sound entering the microphone. For simplicity, volts correspond to Pascals; 0dBV = 0dBPa = 94dB SPL. This is the reference used in many parameters found in SiSonic™ datasheets. Sweeping the frequency of the input will produce the frequency response of the microphone at its output.

Figure 1: Basic Equivalent Circuit Model Premise



For analog microphones, this output is electrically the same as the microphone output. It can be observed as the frequency response or fed into circuitry downstream in the audio processing.

Digital microphones encode their output in a 1-bit PDM (Pulse Density Modulated) stream. The simulation output represents the microphone output if it were decoded using an infinite bandwidth decimation filter. The output is then converted to an electrical signal for simulation purposes only. This conversion scales the output so 1 volt represents the digital output relative to full scale. For example, an output of -26dBV from the simulation would represent an output of -26dBFS from the microphone.

Note that the output for PDM microphones is assumed to have been generated by the default clock frequency of the microphone, which in normal mode is usually 2.4MHz. For SiSonic™ digital microphones with multi-mode function, the clock frequency can be lowered for lower power consumption. Some audio performance will change as clock rate increases or decreases. Only normal mode is simulated. Low-power and sleep modes are not implemented in the simulation environment.

Clock rate is an additional input parameter for I2S microphone simulation models. Here the input clock rate impacts the frequency response since decimation is performed inside the microphone and the Over Sampling Rate (OSR) is fixed at 64x.



REQUIREMENTS

The models run using the fast and powerful LTspice simulator with schematic capture and waveform viewer. This software does not require special license to perform general circuit simulation. It can be downloaded and installed from the URL below:

<http://www.linear.com/designtools/software/#LTspice>

LTspice operates on both Windows and Macintosh platforms. Further details on which OS versions are supported can be found on the www.linear.com website.

This document demonstrates basic operations, and the tool is fairly simple and intuitive to use. However, if needed, additional helpful documentation can be found on the tool itself can be found at

<http://cds.linear.com/docs/en/software-and-simulation/LTspiceGettingStartedGuide.pdf>.

SIMULATION MODEL CONTENTS

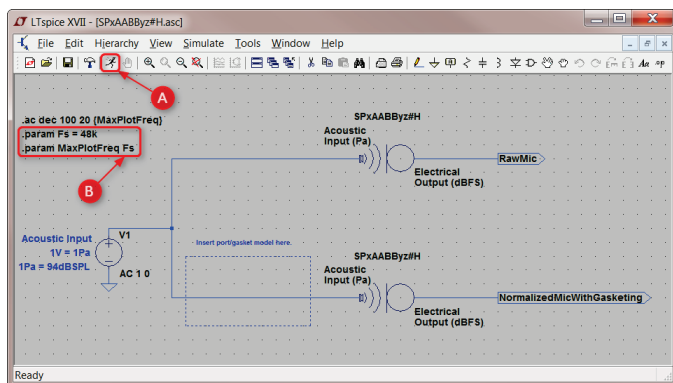
The simulation models are contained in a zipfile (SPxAABByz#H.zip) comprised of a single directory named "SPxAABByz#H Simulation Model" where "SPxAABByz#H" represents the specific microphone model you are simulating. Each supported Knowles microphone will have its own simulation files. Unzip this file then open the subdirectory just created. You will see the following five files:

1. SPxAABByz#H.asc—an example schematic including an acoustic source and microphone
2. SPxAABByz#H.asy—a symbol of the microphone which references the simulation model library
3. SPxAABByz#H.lib—an encrypted microphone model library
4. SPxAABByz#H.plt—parameters used for plotting the frequency response within LTspice
5. AN20 Getting Started With Knowles SiSonic™ Simulation Models.PDF—this document

RUNNING A SIMULATION

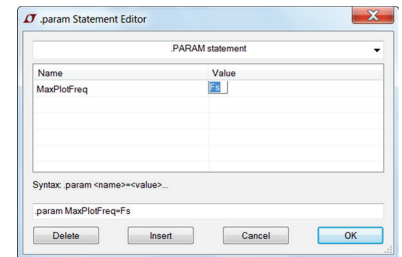
The included schematic contains a microphone, acoustic source for simulation, placeholder for gasketing components, and a second microphone for comparison against the raw microphone.

Figure 2: Baseline Simulation Schematic



To get started, launch LTspice by double-clicking the schematic file (SPxAABByz#H.asc). You will see a circuit similar to what you see in Figure 2. To run a simulation, select "Run" under the simulation menu or click the "run simulation" icon in the tool bar (A in the schematic shown).

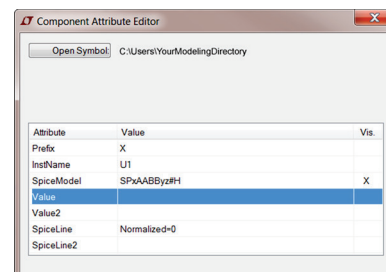
Figure 3: Setting Maximum Plot Frequency



By default, the sampling rate (Fs) is set to 48kHz and the simulation plot will be generated from 20Hz to 48kHz; the maximum plot frequency (MaxPlotFreq) is configured as the sampling frequency (Fs), but this is not a requirement. These parameters are user configurable. To edit the parameter for maximum frequency to plot, right-click the ".param MaxPlotFreq Fs" spice directives then right-click the Value and set it to the desired value as shown below. Setting the sampling rate is done in a similar manner. Sampling rate is usually related to system bandwidth (Fs/2), but it is often useful to understand the microphone characteristics at frequencies above Fs/2.

As stated earlier, models are valid at audio frequencies, but it is helpful to understand expected performance into ultrasonic frequencies. Model assumptions change at ultrasonic frequencies, and different models would be required for more accurate absolute results above ~24kHz.

Figure 4: Normalizing Mic Output to 1kHz

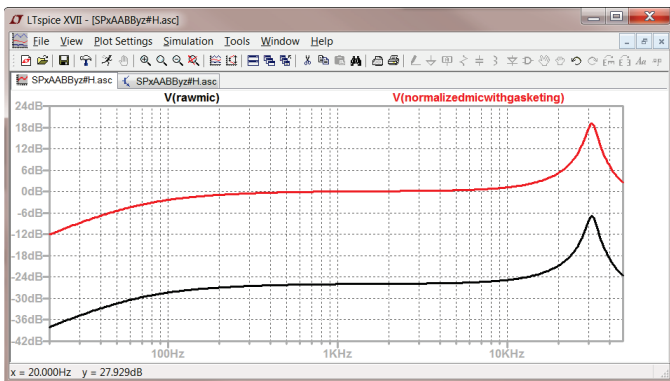


The "raw mic" output is not normalized. This implies that if a 1kHz, 94dB SPL input is simulated, the output of the raw mic will match the microphone's sensitivity as specified in its datasheet. Note that the datasheet normalizes the free field frequency response to 1kHz. As it is generally easier to

compare when output is normalized to 1kHz, the Knowles library includes the ability to change the simulated output to be normalized to 1kHz or non-normalized (raw microphone) data. In LTspice, this is controlled by updating attributes in the microphone model. Simply right-click on the microphone and edit the "Normalized" variable—the first SpiceLine as shown in Figure 4. Setting Normalized to 0 (False) disables normalization; setting Normalized to 1 (True) enables normalization of the output to 1kHz.

After you've run a simulation, you will see the response of the raw mic and the normalized mic with gasketing. Double clicking the simulation result tab will make the results fill the entire LTspice window, similar to the screenshot below.

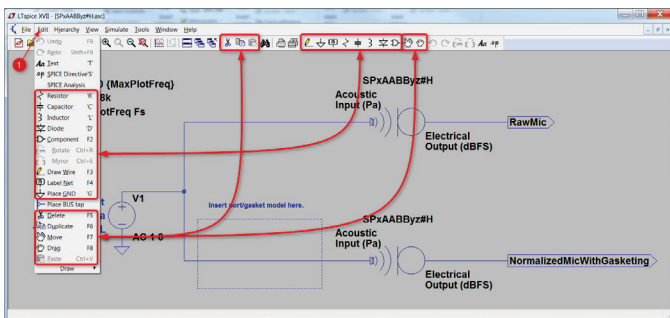




The default .plt file included in the zip file sets up the display format of the simulation's results. If you wish to add a waveform, this is easily handled by clicking on the circuit node you wish to observe using the red probe cursor which you will see after running a simulation. Alternatively, if you click on the simulation window then select the "Plot Settings" menu and click on "Visible Traces" you can select the waveforms you'd like to plot by clicking on the trace (ctrl-clicking any additional waveforms desired).

ADDING ACOUSTIC LOAD

Now we will show how to include an acoustic load representing gasketing in simulations. This will familiarize users with the LTspice tool set and how to use it to model acoustic applications. We will show an example modifying the path in Figure 2 then comparing its response to that of the raw mic. The example will also go through "bracketing" parameters through the use of the ".STEP PARAM" LTspice directive. This should enable you to functionally perform simulations. The actual mathematical modeling used in simulations is beyond the scope of this document; however, an example from Leo Beranek's *Acoustics*¹ is included as his works are great references. Prototyping is always recommended to verify and validate any simulation modeling.



First, let's review basic LTspice operators to add and modify elements in the simulation. The above figure highlights the basic commands and operators, showing that there are several ways to access them in the LTspice User Interface (UI). The paradigm for editing within the LTspice UI is to select the action (move, mirror, rotate, drag, or delete) then select the objects to act on. For example, to delete an element, any of the following will change the UI to "delete mode" indicated by a scissors cursor:

1. Scissors on the schematic toolbar
2. F5
3. Edit Menu→Delete
4. Alt-e + d

Pressing the escape key will leave the mode you are in, putting you back in the default mode of the LTspice UI; by default, no specific action is chosen, but clicking and moving the mouse moves you around the schematic, rolling the mouse wheel forward/backwards zooms in/out, etc.

Now we will add new signal paths and compare their performance to the already existing ones in the example. To do this, you can copy one of the existing paths or create a new one. For demonstration purposes, we will both create a new path and then replicate it so you will know how to create new paths or copy-paste-edit existing ones.

To your schematic, add R, L, and C components representing an "Intermediate-Sized Tube" as developed in Beranek's *Acoustics*, section 5.9¹. You will need to adjust l (tube length) and r (tube radius) to match the tube you are modeling. As the resistance is a function of frequency, the "laplace" operator is used in a voltage-controlled current source, where ohm's law tells us the Laplace Transform of the current, $I(s) = V(s)/R(s)$ with s equal to $j\omega$. Per Beranek:

$$R = \frac{\rho_0}{\pi r^2} \sqrt{2\omega\mu_0} \left[\left(\frac{l}{r} + 1 \right) \right] \text{ (MKS } \Omega \text{ or } N \cdot S/m)$$

$$L = \frac{\rho_0(l + 0.85r)}{\pi r^2} \text{ (kg/m}^4\text{)}$$

$$C = \frac{\text{volume}}{3\rho_0 c^2} \text{ (m}^4\text{S}^2/\text{kg)}$$

where:

$$l = \text{tube length (m)}$$

$$r = \text{tube radius (m)}$$

$$\omega = \text{frequency of input (radians)}$$

$$c = \text{speed of sound in air (m/s)}$$

$$\rho_0 = \text{Density of Air (kg/m}^3\text{)}$$

$$\mu_0 = \text{Viscosity Coefficient of Air (N} \cdot \text{S/m}^2\text{)}$$

The steps to be performed to add the tube gasket are:

1. F2 to add a "Component" then add a voltage-controlled current source (G2) representing equivalent resistance. Note that you may need to rotate (ctrl-R) and/or mirror (ctrl-E) the component before placement to get the desired orientation
2. F3 to add wire connections
3. "L" to add an equivalent inductor
4. "C" to add an equivalent capacitor and "G" to add a ground
5. Right-click the microphone in the "raw mic" path and change the "Normalized" parameter to 1 (TRUE)



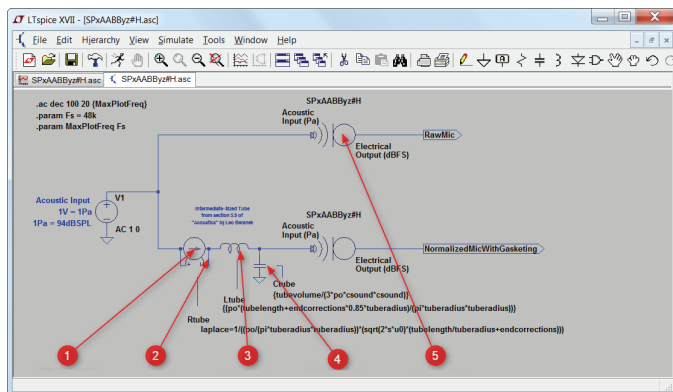
GETTING STARTED WITH KNOWLES SISONIC™ SIMULATION MODELS

Enter any “PARAMS” that will be used to make the circuit easier to understand and edit. Params (Parameters) and other LTspice simulation directives can be entered by typing “S” or through Edit Menu→Spice Directive. Similarly, text describing the parameters (or any other detail worthy of textual description) can be added by typing “T” or Edit Menu→Text.

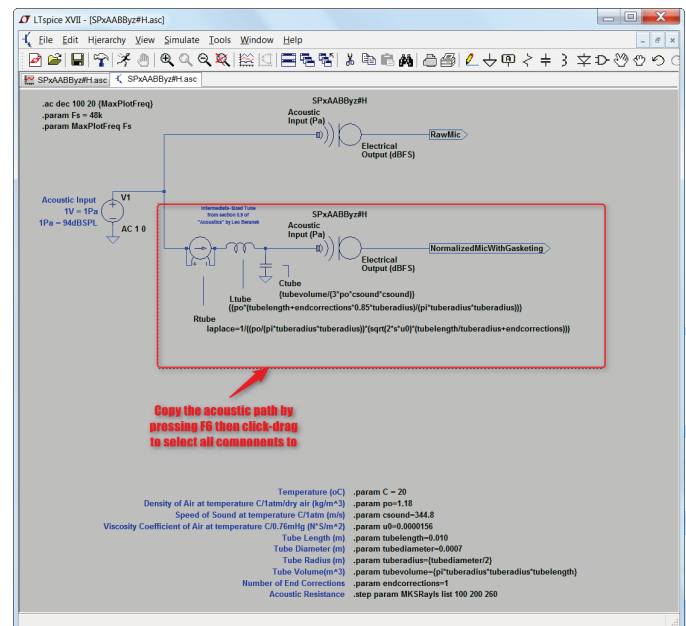
Text Description	LTspice Directive
Density of Air at temperature C/1 atm/dry air (kg/m^3)	.param po=1.18
Speed of Sound at temperature C/1 atm (m/s)	.param csound=344.8
Viscosity Coefficient of Air at temperature C/0.76mHg (N*S/m^2)	.param u0=0.0000156
Tube Length (m)	.param tubelength=0.010
Tube Diameter (m)	.param tubediameter=0.0007
Tube Radius (m)	.param tuberadius={tubediameter/2}
Tube Volume (m^3)	.param tubevolume=(pi*tuberadius*tuberadius*tubelength)
Number of Error Corrections	.param endcorrections=1
Acoustic Resistance	.step param MKSRayls list 100 200 260

The tube length and diameter in this specific example have been chosen as 10mm (0.010m) and 0.7mm (0.0007m), respectively. Below are the parameters added for this example.

When complete, you should now have a circuit similar to the below schematic.



Now copy the “normalized mic with gasket” path by clicking F6 to enter “copy” mode then select the whole path. You can add any missing wires or press escape and repeat the component selection step.

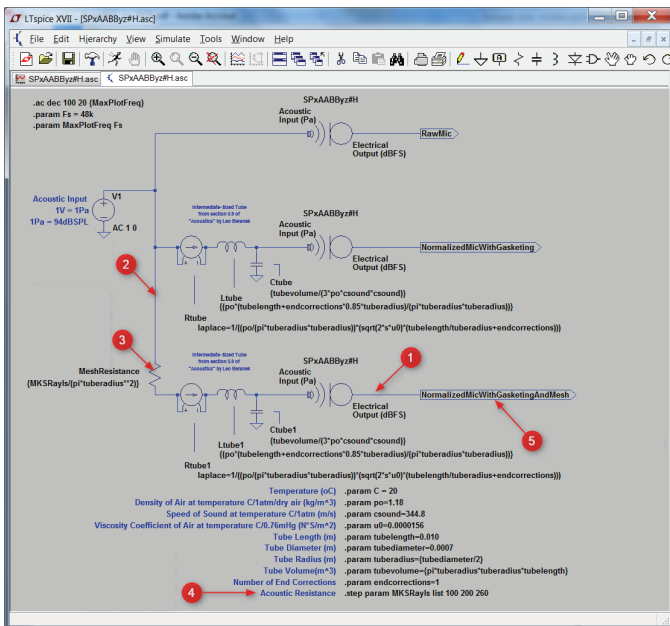


Following the steps show in the figure below, you will add a path representing the gasket along with an acoustic mesh (3) intended to attenuate the resonance induced by the gasketing. Assuming a uniform resistance, the acoustic resistance is the rating of the mesh (in MKSRayls) divided by the active area of the mesh (equal to the tube opening in this example).

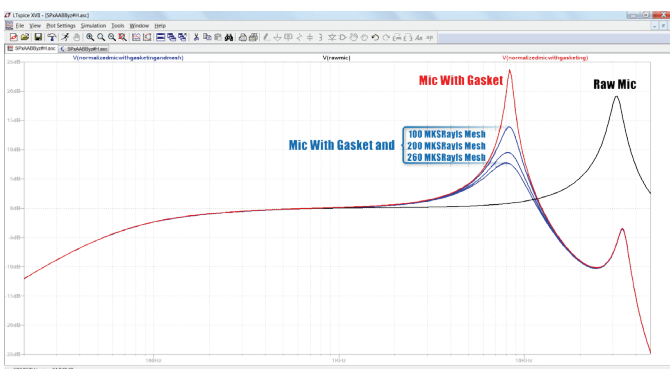
1. Paste the copied acoustic path
2. Add wires (F3) to connect the new acoustic path to the simulation source



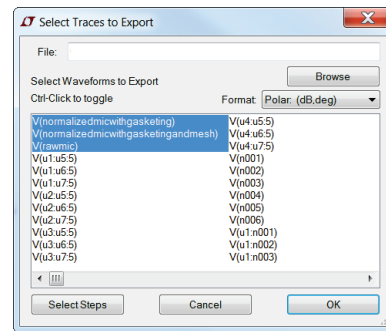
3. Add a resistor representing the equivalent acoustic resistance ("R")
4. Using a .step param allows simulation of several values at the same time
5. Rename the new path's output to avoid simulation errors



The mesh's acoustic resistance is declared in an LTspice "STEP PARAM" directive. For this example, 100, 200, and 260 MKSRays meshes are simulated. "STEPS" can be enumerated or generated lists. The simulation output is shown below, plotting the 1) raw mic, 2) mic with gasket, and 3) mic with gasket and mesh.



The data from simulations can also be exported to a text file. This is done from the menu File->Export data as text (alt-f then e) if the simulation plot window is active. You will see a pop-up window similar to the one below. Here you can select the data you wish to export.



The format of the exported data will have frequency followed by polar data for each selected waveform. The polar coordinates (dB and degrees) will be in parenthesis and separated by a tab. The first column containing frequency data will be a single number in scientific notation. For a simulation that started at

20 Hz, the first frequency entry would be:

```
2.0000000000000000e+001
```

The polar data in the next column will be in parentheses as below:

```
(-1.20387323438029e+001dB, 8.32186253337431e+001°)
```

The LTspice-exported text file can be imported into programs such as Excel. For Excel, text can be imported, but the text would need to be converted before plotting or manipulation. The frequency column can be directly interpreted as a number by selecting the column and changing the format to "scientific." To convert the polar data text to a number, you will need to extract the magnitude data. This can be achieved by typing the formula `"=VALUE(MID(CELL,2,FIND("dB",CELL)-2))"` where `CELL` is the imported cell you wish to convert.

These are the steps and tools available to better understand the expected performance of acoustic applications before actually building anything. After finding a design with the desired acoustic response for your application (or close to it), prototyping and validation testing are always recommended.



Information contained herein is subject to change without notice. It may be used by a party at their own discretion and risk. We do not guarantee any results or assume any liability in connection with its use. This publication is not to be taken as a license to operate under any existing patents.

REVISION HISTORY

Revision	Description	Date
1	Initial Release	12/28/17
2	Change image with parameters to a table, add clarification of spice directives vs text	2/8/18

¹ Beranek, Leo L. *Acoustics*. New York, N.Y.: Published by the American Institute of Physics for the Acoustical Society of America, 1986. Print.

² Beranek, Leo L. "5.9. Intermediate-sized Tube—Mixed Mass-Resistance Element [$a(\text{meters}) > 0.01/\sqrt{f}$ and $a < 10/f$]." *Acoustics*. New York, N.Y.: Published by the American Institute of Physics for the Acoustical Society of America, 1986. 137-8. Print.