

# Probe Tube Microphone

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

## Introduction

It is often not possible to insert a normal microphone directly into the sound field being measured. The microphone may be too big to fit inside the measured system, such as for in-the-ear measurements for hearing aid fitting. The size of the microphone may also be too large compared to the wavelength, so that it disturbs the acoustic field. In these cases, a probe tube may be attached to the microphone case (see the sketch in Figure 1) in order to distance the microphone from the measurement point. In this model, the microphone sensitivity changes due to the addition of this small probe is investigated, see Ref. 1 (Chapter 4, pp. 160–162) for further details.

This is a time-dependent model of a generic probe tube microphone setup consisting of an external acoustic domain, an elastic probe tube, and the cavity in front of the microphone diaphragm; see the sketch in Figure 1. The probe tube, modeled using the Pipe Acoustics, Transient interface, is connected to two separate 3D pressure acoustics domains, leading to a fully coupled acoustics simulation.



Figure 1: Sketch of the probe tube microphone setup, consisting of a probe tube of length L and outer diameter  $D_0$ . The tube is connected to a cavity (with the indicated dimensions) in front of the microphone diaphragm. An external sound field hits the probe tube with wave vector  $\mathbf{k}$ .

**Note:** This application requires the Acoustics Module.

## Model Definition

The probe tube microphone geometry and the computational domain considered in this model are shown schematically in Figure 1. From right to left, the model consists of the cavity in front of the microphone, which takes the form of a cylinder of radius R and height H connected to a cone of bottom radius R and top radius  $D_0$ .  $D_0$  is also the outer diameter of the probe tube of length L. The probe is made of an elastic material with inner diameter  $2d_r = 2$  mm and wall thickness  $d_w = 0.3$  mm. The tube material is set to have a Young's modulus of 0.1 GPa and a Poisson's ratio of 0.4.

Table 1	l lists the	model	parameters,
---------	-------------	-------	-------------

VARIABLE	VALUE	DESCRIPTION
f	500 Hz	Incident wave frequency
T	1/f	Incident wave period
ω	$2\pi f$	Incident wave angular frequency
$c_0$	343 m/s	Speed of sound at 20 <sup>0</sup> C and 1 atm
$\lambda_{\min}$	$c_0/f$	Wavelength of the incident wave
k	$2\pi/\lambda_{\min}$	Wave number of the incident wave
L	20 mm	Probe tube length
R	5 mm	Microphone casing radius
Н	5 mm	Cavity height
$d_{\mathrm{r}}$	1 mm	Pipe inner radius
$d_{ m w}$	0.3 mm	Pipe wall thickness
$D_0$	$2d_{\rm r} + 2d_{\rm w} = 2.6 {\rm mm}$	Probe tube outer diameter
$kD_0$	$kD_0 = 0.02$	Wave number times tube diameter
$d_{ m visc}$	0.1 mm	Viscous boundary layer thickness at 500 Hz

TABLE I: MODEL PARAMETERS.

A sinusoidal wave moving in the positive x direction is incident on the probe tube microphone. The wave with an amplitude of 1 Pa is given by

$$p_{\rm in} = 1 \,\mathrm{Pa} \,\cdot \sin(\omega t - kx) \tag{1}$$

where  $\omega$  is the angular frequency, and k is the wave number (these are given in Table 1). Such a plane monochromatic wave exists as a built-in option for the **Incident Pressure Field** feature in the interface. In the transient analysis, a ramp is automatically added to smoothly increase the background pressure field amplitude over the first period  $T = 1/f_0$ . This is done to ensure numerical stability and can be deactivated under the **Incident Pressure Field** with the **Advanced Physics Options** turned on.

Modeled using the Pipe Acoustics, Transient interface, the long probe tube is treated as a 1D structure. This assumption is valid as long as the interaction between the probe tube and the incoming sound field can be neglected. This, in turn, requires that  $kD_0 \ll 1$ , which is true for the current choice of parameters because  $kD_0 = 0.02$ ; see Table 1.

Another important assumption is that there are no significant thermal and viscous boundary losses inside the probe tube. Also, this assumption holds for the current setup, where the incident field is a monochromatic wave. At a driving frequency of 500 Hz, the viscous boundary layer thickness inside the probe tube is of the order 0.1 mm, which is 10 times smaller than the internal radius  $d_r$ . The thickness of the boundary layer is proportional to  $1/\sqrt{f}$ .

Because the diaphragm is not a fully rigid structure, assume it to have a resistive loss given by the impedance  $Z = 100 \cdot 10^6$  Ns/m<sup>5</sup>. This value is of the order of magnitude observed in common condenser microphones.

## Results and Discussion

An important parameter of a probe tube extension to a microphone is the relation between the pressure at the tip and the pressure at the diaphragm (as measured by the microphone). This transfer function is necessary to calibrate the measurement system. The pressure at the tip of the probe and the pressure at the diaphragm are shown in Figure 2. After an initial transient, the solution becomes periodic after about 4 ms. Hereafter, the system has a gain of about 1.4 and a phase shift equal to  $\phi = \Delta t \cdot \omega = 0.1 \text{ ms} \cdot 2\pi f = 9^\circ$ . The gain and phase shift depend on the frequency content of the applied signal — in this case a pure harmonic tone of 500 Hz.

A full transfer function could be obtained, for example, by using a Gaussian pulse as the incident signal and then performing a Fourier transform of the signal. This requires resolving the frequency content with an appropriate mesh that would become very dense. Moreover, the model assumptions might not hold for the full range of frequencies.

Figure 3 depicts the pressure distribution in the *xz*-plane at the end of the time interval shown in Figure 2.



Figure 2: Pressure as function of time at the probes tip (green line) and at the microphone diaphragm (blue line).

Time=0.01 s Slice: Pressure (Pa) Slice: Pressure (Pa) Line: Pressure (Pa)



Figure 3: Pressure distribution in the xz-plane at time t = 10 ms.

#### COUPLING BETWEEN ID AND 3D DOMAINS

The Pipe Acoustics domain (probe domain) is connected to the first Pressure Acoustics domain (exterior domain) through a pressure condition at the pipe end. This manual coupling ensures continuity of the pressure between the pipe domain and the incident pressure field.

The Pipe Acoustics domain is connected to the second Pressure Acoustics domain (diaphragm cavity domain) through the Acoustics-Pipe Acoustics multiphysics coupling. This coupling guarantees continuity of pressure and velocity between both domains through the equations

$$p_{\rm pipe} = \frac{1}{A} \int p_{\rm t} dS \tag{2}$$

$$-\mathbf{n} \cdot \left(-\frac{1}{\rho} (\nabla p_{t} - \mathbf{q}_{d})\right) = \frac{\partial u_{\text{pipe}}}{\partial t}$$
(3)

where  $p_{pipe}$  is the pipe pressure at the connection point, *A* is the total area of the connected acoustic boundary,  $p_t$  is the total pressure in the acoustic domain, and *S* is the connected acoustic boundary. Furthermore, **n** is the vector normal to the boundary,  $\mathbf{q}_d$  is a dipole source, and  $u_{pipe}$  is the pipe velocity at the connection point.

**Note:** Further examples of coupling between 1D pipes and 3D domains are found in the tutorial model Acoustics of a Pipe System with 3D Bend and Junction, also available in the Application Gallery: www.comsol.com/model/acoustics-of-a-pipe-system-with-3d-bend-and-junction-40831.

#### SOLVER SETUP

The only adjustment needed to be done to the solver is to change it from using a segregated approach to using a fully coupled approach. The default logic in the study is to use segregated groups for physics that do not have a built-in multiphysics coupling. Here, the coupling is set up manually in one of the ends of the pipe. The settings for the transient solver are handled automatically by setting the **Maximum frequency to resolve** in the **Transient Solver Settings** section (as done in the instructions below.)

## Reference

1. D.T. Blackstock, Fundamentals of Physical Acoustics, John Wiley & Sons, 2000.

## **Application Library path:** Acoustics\_Module/Tutorials,\_Pipe\_Acoustics/

probe\_tube\_microphone

## Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click 🙆 Model Wizard.

#### MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Acoustics>Pipe Acoustics>Pipe Acoustics, Transient (patd).
- 3 Click Add.
- 4 In the Select Physics tree, select Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd).
- 5 Click Add.
- 6 In the Select Physics tree, select Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd).
- 7 Click Add.
- 8 Click 🔿 Study.
- 9 In the Select Study tree, select General Studies>Time Dependent.
- IO Click 🗹 Done.

#### GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click **b** Load from File.

4 Browse to the model's Application Libraries folder and double-click the file probe tube microphone parameters.txt.

#### GEOMETRY I

Next, define the geometry of the model. Either build the geometry by following the instructions at the end of the document or import the geometry from a sequence, and then jump to the Materials section. To import the geometry sequence, click Insert Sequence in the Geometry toolbar. Browse to the model's Application Libraries folder and open the file probe tube microphone geom sequence.mph.

- I In the Geometry toolbar, click Insert Sequence and choose Insert Sequence.
- 2 Browse to the model's Application Libraries folder and double-click the file probe\_tube\_microphone\_geom\_sequence.mph.
- 3 In the Geometry toolbar, click 🟢 Build All.

Use the wireframe rendering to more easily see and access faces and lines inside the geometry.

- 4 Click the 🖂 Wireframe Rendering button in the Graphics toolbar.
- **5** Click the **F Zoom Extents** button in the **Graphics** toolbar. The geometry should look like the figure below.



y 🕴 🕳 ×

#### ADD MATERIAL

- I 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 tree, select Built-in>Air.
- 6 Click Add to Component in the window toolbar.
- 7 In the Home toolbar, click 👫 Add Material to close the Add Material window.

#### MATERIALS

Air I (mat2)

- I In the Settings window for Material, locate the Geometric Entity Selection section.
- 2 From the Geometric entity level list, choose Edge.
- 3 Select Edge 25 only.

#### DEFINITIONS

Average 1 (aveop1)

- I In the Definitions toolbar, click 🖉 Nonlocal Couplings and choose Average.
- 2 In the Settings window for Average, type aveop\_mic in the Operator name text field.
- **3** Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- **4** Select Boundary 12 only.

Integration 1 (intop1)

- I In the Definitions toolbar, click 🖉 Nonlocal Couplings and choose Integration.
- 2 In the Settings window for Integration, type intop\_tip in the Operator name text field.
- 3 Locate the Source Selection section. From the Geometric entity level list, choose Point.
- **4** Select Point 14 only.

#### PIPE ACOUSTICS, TRANSIENT (PATD)

Proceed to set up the physics of the problem. Select the domains where the different physics are applied and add the appropriate boundary conditions to couple these.

I In the Model Builder window, under Component I (comp1) click Pipe Acoustics, Transient (patd).

- 2 In the Settings window for Pipe Acoustics, Transient, locate the Edge Selection section.
- 3 Click Clear Selection.
- 4 Select Edge 25 only.

#### Pipe Properties 1

- I In the Model Builder window, under Component I (comp1)>Pipe Acoustics, Transient (patd) click Pipe Properties 1.
- 2 In the Settings window for Pipe Properties, locate the Pipe Shape section.
- 3 From the list, choose Circular.
- **4** In the  $d_i$  text field, type 2\*dr.
- 5 Locate the Pipe Model section. From the Pipe model list, choose Anchored at one end.
- 6 From the *E* list, choose User defined. In the associated text field, type 0.1[GPa].
- 7 From the v list, choose User defined. In the associated text field, type 0.4.
- **8** In the  $\Delta w$  text field, type dw.
- 9 In the Model Builder window, click Pipe Acoustics, Transient (patd).
- **10** In the **Settings** window for **Pipe Acoustics, Transient**, locate the **Transient Solver Settings** section.
- II In the  $f_{\text{max}}$  sol text field, type f.

#### Pressure 1

- I In the Physics toolbar, click 📄 Points and choose Pressure.
- **2** Select Point 14 only.
- 3 In the Settings window for Pressure, locate the Pressure section.
- **4** In the  $p_{in}$  text field, type p2.

This manual coupling applies the pressure of the first Pressure Acoustics Domain into the Pipe Acoustics Domain.

#### PRESSURE ACOUSTICS, TRANSIENT (ACTD)

- I In the Model Builder window, under Component I (compl) click Pressure Acoustics, Transient (actd).
- 2 In the Settings window for Pressure Acoustics, Transient, locate the Domain Selection section.
- 3 Click Telear Selection.
- **4** Select Domain 1 only.
- 5 Locate the Transient Solver and Mesh Settings section. In the  $f_{\text{max}}$  text field, type f.

#### Cylindrical Wave Radiation I

- I In the Physics toolbar, click 🔚 Boundaries and choose Cylindrical Wave Radiation.
- 2 Select Boundaries 1, 2, 13, and 18 only.
- **3** In the Settings window for Cylindrical Wave Radiation, locate the Cylindrical Wave Radiation section.
- **4** Specify the **r**<sub>axis</sub> vector as

0	x
0	у
1	z

Add an incident plane wave as defined in Equation 1 using the built-in Plane wave (monochromatic) option. The wave is automatically multiplied with a smoothing ramp function over the first period. This option can be seen by enabling the Advanced Physics **Options** view.

Incident Pressure Field 1

- I In the Physics toolbar, click 📃 Attributes and choose Incident Pressure Field.
- 2 In the Settings window for Incident Pressure Field, locate the Incident Pressure Field section.
- **3** In the  $p_0$  text field, type 1.
- 4 From the *c* list, choose From material.
- 5 From the Material list, choose Air (matl).
- 6 Specify the **e**<sub>k</sub> vector as

1 x 0 y 0 z

7 In the  $f_0$  text field, type f.

#### PRESSURE ACOUSTICS, TRANSIENT 2 (ACTD2)

- I In the Model Builder window, under Component I (compl) click Pressure Acoustics, Transient 2 (actd2).
- **2** In the Settings window for Pressure Acoustics, Transient, locate the Domain Selection section.
- 3 Click Clear Selection.

**4** Select Domains 2 and 3 only.

5 Locate the Transient Solver and Mesh Settings section. In the  $f_{\text{max}}$  text field, type f.

#### Impedance I

- I In the Physics toolbar, click 📄 Boundaries and choose Impedance.
- 2 Select Boundary 7 only.
- 3 In the Settings window for Impedance, locate the Impedance section.
- 4 In the  $Z_n$  text field, type 100e6[N\*s/m^5]\*R^2\*pi.

#### MULTIPHYSICS

Acoustic-Pipe Acoustic Connection 1 (apc1)

- I In the Physics toolbar, click An Multiphysics Couplings and choose Global>Acoustic-Pipe Acoustic Connection.
- **2** In the Settings window for Acoustic-Pipe Acoustic Connection, locate the Coupled Interfaces section.
- 3 From the Acoustics list, choose Pressure Acoustics, Transient 2 (actd2).

Selecting this second **Pressure Acoustics** domain is important, as otherwise the probe will have both ends connected to the same domain.

#### MESH I

Proceed and generate the mesh based on the **Physics-controlled mesh** suggestion. The frequency controlling the maximum element size is per default taken **From study**, that is, from the **Maximum frequency to resolve**. In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. In this model, we use 10 elements per wavelength; the default **Automatic** is to have 5.

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Pressure Acoustics, Transient (actd) section.
- **3** From the Number of mesh elements per wavelength list, choose User defined.
- **4** In the text field, type 10.
- 5 Locate the Pressure Acoustics, Transient 2 (actd2) section. From the Number of mesh elements per wavelength list, choose User defined.
- 6 In the text field, type 10.
- 7 Locate the Physics-Controlled Mesh section. From the Element size list, choose Coarse.
- 8 Locate the Sequence Type section. From the list, choose User-controlled mesh.

Size 1

- I In the Model Builder window, right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Edge**.
- **4** Select Edge 25 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section.
- 7 Select the Maximum element size check box. In the associated text field, type L/10.

Size 2

- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Boundary**.
- **4** Select Boundary 12 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section.
- 7 Select the Maximum element size check box. In the associated text field, type dr/2.
- 8 Click 📗 Build All.

#### STUDY I

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- **3** In the **Output times** text field, type range(0, T/25, 5\*T).

The data will be recorded for 5 periods (5\*T) with a sampling of 25 points per period (increase the number of points to get better resolution in the stored solution).

#### Solution 1 (soll)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Time-Dependent Solver I node.

## 4 Right-click Study I>Solver Configurations>Solution I (sol1)>Time-Dependent Solver I and choose Fully Coupled.

In this model, the coupling between the first Pressure Acoustics domain and the Pipe Acoustics Domain is set up manually. The default behavior, in the study, is for COMSOL to generate solver suggestions for the individual constituting physics (notice the tags) and sets up a segregated solver suggestion. This model is best solved using the fully coupled approach.

Now, solve the model and proceed to look at the results. First, look at the default plots and then create two custom plot groups. The model only takes a few seconds to solve.

- 5 In the Model Builder window, under Study I>Solver Configurations>Solution I (solI)> Time-Dependent Solver I click Fully Coupled I.
- 6 In the Settings window for Fully Coupled, click **=** Compute.

## RESULTS

## Acoustic Pressure (patd)

Time=0.01 s

#### Line: Pressure (Pa)



### Acoustic Pressure (actd)



Acoustic Pressure, Isosurfaces (actd)



Isosurface: Total acoustic pressure (Pa)



### Acoustic Pressure (actd2)



#### Surface: Total acoustic pressure (Pa)



Acoustic Pressure, Isosurfaces (actd2)



Isosurface: Total acoustic pressure (Pa)



Follow the steps below to reproduce the plot in Figure 3.

#### 3D Plot Group 7

In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.

Slice 1

- I Right-click **3D Plot Group 7** and choose **Slice**.
- 2 In the Settings window for Slice, locate the Expression section.
- **3** In the **Expression** text field, type p2.
- 4 Locate the Plane Data section. From the Plane list, choose ZX-planes.
- 5 In the Planes text field, type 1.
- 6 Locate the Coloring and Style section. Click Change Color Table.
- 7 In the Color Table dialog box, select Wave>Wave in the tree.
- 8 Click OK.
- 9 In the Settings window for Slice, locate the Coloring and Style section.
- **IO** From the Scale list, choose Linear symmetric.

Slice 2

- I In the Model Builder window, right-click 3D Plot Group 7 and choose Slice.
- 2 In the Settings window for Slice, locate the Expression section.
- **3** In the **Expression** text field, type p**3**.
- 4 Locate the Plane Data section. From the Plane list, choose ZX-planes.
- 5 In the Planes text field, type 1.
- 6 Click to expand the Inherit Style section. From the Plot list, choose Slice 1.

Line 1

- I Right-click **3D Plot Group 7** and choose Line.
- 2 In the Settings window for Line, locate the Coloring and Style section.
- **3** From the Line type list, choose Tube.
- 4 In the **Tube radius expression** text field, type 0.5\*patd.dh.
- 5 Select the Radius scale factor check box.
- 6 Click to expand the Inherit Style section. From the Plot list, choose Slice 1.
- 7 In the **3D Plot Group 7** toolbar, click **9** Plot.

Cross Sections

I In the Model Builder window, under Results click 3D Plot Group 7.

2 In the Settings window for 3D Plot Group, type Cross Sections in the Label text field.

The figure should look like the one depicted in Figure 3.

ID Plot Group 8

In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.

Global I

- I Right-click ID Plot Group 8 and choose Global.
- 2 In the Settings window for Global, locate the y-Axis Data section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
aveop_mic(p3)	Ра	Diaphragm Pressure
<pre>intop_tip(p)</pre>	Ра	Probe Tip Pressure

**Pressure Profiles** 

- I In the Model Builder window, click ID Plot Group 8.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.
- 3 Select the x-axis label check box.
- 4 Select the y-axis label check box. In the associated text field, type Pressure (Pa).
- 5 In the ID Plot Group 8 toolbar, click 💽 Plot.
- 6 In the Label text field, type Pressure Profiles.

The figure should look like the one in Figure 2.

## Appendix: Geometry Sequence Instructions

If you want to create the geometry yourself, follow these steps.

From the File menu, choose New.

#### NEW

In the New window, click 🙆 Model Wizard.

#### MODEL WIZARD

I In the Model Wizard window, click 间 3D.

2 Click **M** Done.

#### **GLOBAL DEFINITIONS**

#### Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

**3** In the table, enter the following settings:

Name	Expression	Value	Description
L	20[mm]	0.02 m	Probe tube length
R	5[mm]	0.005 m	Microphone casing radius
Н	5[mm]	0.005 m	Probe tube volume height
dr	1 [ mm ]	0.001 m	Pipe inner radius
dw	0.3[mm]	3E-4 m	Pipe wall thickness

#### GEOMETRY I

Cone I (cone I)

- I In the **Geometry** toolbar, click **Cone**.
- 2 In the Settings window for Cone, locate the Size and Shape section.
- 3 In the Bottom radius text field, type R.
- 4 In the **Height** text field, type H.
- 5 In the **Top radius** text field, type dr+dw.

#### Cylinder I (cyl1)

- I In the Geometry toolbar, click 💭 Cylinder.
- 2 In the Settings window for Cylinder, locate the Size and Shape section.
- 3 In the Radius text field, type R.
- 4 In the **Height** text field, type H.
- 5 Locate the Position section. In the z text field, type -H.

#### Polygon I (poll)

- I In the Geometry toolbar, click  $\bigoplus$  More Primitives and choose Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- **3** From the **Data source** list, choose **Vectors**.
- **4** In the **x** text field, type 0 0.
- **5** In the **y** text field, type 0 0.

6 In the z text field, type H H+L.

#### Cylinder 2 (cyl2)

- I In the **Geometry** toolbar, click 💭 **Cylinder**.
- 2 In the Settings window for Cylinder, locate the Size and Shape section.
- 3 In the Radius text field, type 10[mm].
- 4 In the Height text field, type 35[mm].
- 5 Locate the Position section. In the z text field, type -H.

Work Plane I (wp1)

- I In the Geometry toolbar, click 📥 Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 From the Plane type list, choose Face parallel.
- 4 On the object **cone1**, select Boundary 4 only.

Work Plane 1 (wp1)>Plane Geometry In the Model Builder window, click Plane Geometry.

Work Plane 1 (wp1)>Circle 1 (c1)

- I In the Work Plane toolbar, click (•) Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type dr.
- 4 Click to expand the Layers section.

Form Union (fin)

In the Home toolbar, click 🟢 Build All.