

# Focused Ultrasound Induced Heating in Tissue Phantom

This model is licensed under the COMSOL Software License Agreement 6.0. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

# Introduction

When an ultrasound beam passes through a volume of tissue, some of the energy of the primary acoustic field is absorbed locally by the tissue and turned into heat. This results in a temperature increase whose magnitude is a function of the physical properties of the medium (acoustic absorption coefficient, density, and specific heat), the properties of the ultrasound device (beam geometry), and the frequency and time-averaged acoustic intensity of the acoustic field; see Ref. 1 and Ref. 2. The actual temperature rise that can be obtained also depend on the conduction and convection properties of the tissue involved, for example, the blood perfusion rate of the tissue.



Figure 1: Focused ultrasound makes selective and targeted heating possible: heating of tissues lying within the focal volume can be achieved with minimal damage to nearby healthy tissue and other structures lying elsewhere in the path of the beam.

Note: This application requires the Acoustics Module and the Heat Transfer Module.

Therapeutic applications of ultrasound typically involve focused beams, which allow the ultrasound energy to be directed into a small area within the tissue region that needs the treatment. Moreover, heating of tissues lying within the focal volume can be achieved with minimal damage to nearby healthy tissue and other structures lying elsewhere in the path of the beam, as shown in Figure 1.

Depending on the dosage parameters — that is, field intensity and exposure time — the clinical applications of focused ultrasound (FU) are generally grouped into two categories, namely "ultrasound hyperthermia" and "focused ultrasound surgery" (FUS); see Ref. 3. Generally, during hyperthermia applications, tissues are exposed to ultrasound for long

periods (from 10 to 60 minutes) at lower intensity levels, such that the irradiated tissue temperature is elevated and maintained at 41 °C to 45 °C during the therapy. The biological change thus induced can be reversed. In contrast, focused ultrasound surgery utilizes intense, relatively short bursts (0.1 s to 30 s) to induce irreversible changes in the focal tissue volume. In this type of applications, nonlinear acoustic effects and acoustic cavitation usually play essential roles; the tissue temperature in the focal zone can reach 70 °C to 90 °C within a few seconds. This technique is also known as ultrasound ablation.

The thermal effect of focused ultrasound also brings concerns about possible harmful effects of diagnostic ultrasound, especially in obstetrical examinations when the fetus is exposed to ultrasound. The U.S. Food and Drug Administration has set up regulations on the maximum thermal index, a dosage parameter reflecting the combined effect of temperature and exposure time that is an estimate of risk from heat (Ref. 4). Diagnostic ultrasound systems now come with displays with the Thermal Index (TI) and the Mechanical Index (MI), the other estimate of risk from the nonthermal effects of ultrasound in order to meet the U.S. government's regulations. Safety in the use of ultrasound has also been addressed extensively in the academic field; see, for example, Ref. 5, Ref. 6, and Ref. 7.

The current model is inspired by the experiments to measure focused ultrasound induced heating in a tissue phantom from Ref. 8. The model uses the same geometry and material properties as in Ref. 8. The model exemplifies how to use COMSOL Multiphysics to model tissue heating induced by focused ultrasound. The simulation results are compared to the experimental data in the reference.

# Model Definition

Figure 2 shows the geometry simulated in this model. Both the tissue phantom and the acoustic transducer are immersed in water. The transducer is bowl shaped with a focal length of 62.64 mm, an aperture of 35 mm in radius, and a hole of 10 mm in radius in the center. The tissue phantom has the shape of a cylinder with 53.6 mm in radius and 80.5 mm in length. The tissue phantom and the transducer are arranged coaxially so the model can be defined as being 2D axisymmetric.

The transducer is driven at the frequency of 1 MHz. It is turned on for 1 second and then turned off to let the tissue phantom cool down completely in water. The model thus solves

for the heating of the tissue phantom for 1 second and then simulates the cooling process after the acoustic source is turned off.



Figure 2: The geometry of the acoustic transducer and tissue phantom. The transducer is bowl shaped with a hole in the center. Both the tissue phantom and the transducer are immersed in water. The axisymmetric geometry allows for a 2D axisymmetric simulation.

The model uses the Pressure Acoustics, Frequency Domain interface to model the stationary acoustic field in the water and the tissue domain, to obtain the acoustic intensity distribution in the tissue phantom. The absorbed acoustic energy is calculated and used as the heat source for the Bioheat Transfer interface model. Because the acoustic focal region (like the heated area) is much smaller than the size of the tissue phantom, the thermal simulation is performed only in the tissue domain.

The wave equation solved is the homogeneous Helmholtz equation in 2D axisymmetric cylindrical coordinates:

$$\frac{\partial}{\partial r} \left[ -\frac{r}{\rho_{c}} \left( \frac{\partial p}{\partial r} \right) \right] + r \frac{\partial}{\partial z} \left[ -\frac{1}{\rho_{c}} \left( \frac{\partial p}{\partial z} \right) \right] - \left[ \left( \frac{\omega}{c_{c}} \right)^{2} \right] \frac{rp}{\rho_{c}} = 0$$
(1)

Here *r* and *z* are the radial and axial coordinates, *p* is the acoustic pressure, and  $\omega$  is the angular frequency. The density,  $\rho_c$ , and the speed of sound,  $c_c$ , are complex-valued to account for the material's damping properties.

Using Equation 1 involves the assumption that the acoustic wave propagation is linear and also that the amplitude of shear waves in the tissue domain are much smaller than that of the pressure waves. Nonlinear effects and shear waves are therefore neglected.

Given the acoustic pressure field, the acoustic intensity field is readily derived. The heat source Q for thermal simulation, given in the plane-wave limit, is then calculated as

$$Q = 2\alpha_{ABS}I = 2\alpha_{ABS}\left|\operatorname{Re}\left(\frac{1}{2}p\mathbf{v}\right)\right|$$
(2)

where  $\alpha_{ABS}$  is the acoustic absorption coefficient, I is the acoustic intensity magnitude, p is the acoustic pressure, and  $\mathbf{v}$  is the acoustic particle velocity vector. In COMSOL, the intensity is a derived variable where the magnitude can be accessed as acpr.I\_mag. Likewise, the dissipated power density (for plane waves) is defined as acpr.Q\_pw. The heat source Q is thus readily calculated once the acoustic field is solved.

**Note:** For further details about the intensity variables choose **Help>Documentation** and then search for the string *Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)*. Then select the section *Postprocessing Variables*. Two parts exist here; one describing the intensity variables and one describing power dissipation variables.

Inserting the volumetric acoustic heat source into the Pennes' Bioheat Transfer equation to model heat transfer within biological tissue gives

$$\rho C_{\rm p} \frac{\partial T}{\partial t} = \nabla \cdot (k \nabla T) - \rho_{\rm b} C_{\rm b} w_{\rm b} (T - T_{\rm b}) + Q + Q_{\rm met}$$

where *T* is the temperature,  $\rho$  is the density,  $C_p$  is the specific heat, **k** is the thermal conductivity,  $\rho_b$  is the density of blood,  $C_b$  is the specific heat of blood,  $w_b$  is the blood perfusion rate,  $T_b$  is the temperature of the blood, *Q* is the heat source (the absorbed ultrasound energy calculated from Equation 2), and  $Q_{met}$  is the metabolic heat source.

In this model, assume that the tissue properties do not change when the temperature rises. Blood perfusion is also neglected.

Figure 3 shows the model geometry and material domains defined in the model. Four cylindrical perfectly matched layers (PMLs) (r1-r4) and one spherical PML (c1) are used



to absorb the outgoing waves. The pressure acoustics simulation is performed in all domains while the heat transfer model is only applied in the tissue phantom domain.

Figure 3: Model geometry. Water domains are shown in blue and tissue phantom domains are shown in purple. Four cylindrical PMLs (r1-r4) and one spherical PML (c1) are used to absorb the outgoing waves.

To accurately resolve the sharp pressure gradient in the focal region, the model uses a fine mesh with size  $\lambda/8$  (where  $\lambda$  is the wavelength) within that region (both for the heat and the acoustics problem). A coarser mesh with size  $\lambda/5$  in water is used for the other acoustic domains. Both the acoustic pressure and the temperature uses the default quadratic (2nd order) elements as discretization.

Table 1 shows the material properties used in the model simulation. The properties used for the tissue phantom are the measured data described in Ref. 8. For comparison, the table also lists properties for human tissue published in Ref. 9.

Property	Density (kg/m <sup>3</sup> )	Speed of sound (m/s)	Attenuatio n (Np/m/ MHz)	Specific heat (J/(kg·K))	Thermal conductivity (W/(m·K))
Water (at 293.7 K)	1000	1483	0.025	N/A	N/A
Tissue phantom	1044	1568	8.55	3710	0.59
Human tissue	1000- 1100	1450-1640	4.03–17.27	3600–3890	0.45–0.56

TABLE I: MATERIAL PROPERTIES USED IN THE MODEL.

# Results and Discussion

Figure 4 depicts the acoustic pressure. The ultrasonic beam travels through the water layer and into the tissue phantom. The beam converges into a focal area where the acoustic pressure amplitude reaches as high as 1.11 MPa at the focal point. This result agrees well with the results presented in Ref. 8. Note the diffraction-like pattern near the edges (especially at the top of the computational domain); this is not a physical phenomenon but an effect of the perfectly matched layer.



Figure 4: The acoustic pressure field in the water and tissue domains.

A depiction of the acoustic intensity magnitude is given in Figure 5. This plot shows more clearly how the acoustic energy is focused and distributed in the area of interest. Most of the heating happens in the oval-shaped focal area which is about 8 mm long and 1.3 mm wide. Figure 6 shows the acoustic pressure amplitude profile along the *z*-axis (r = 0). By zooming in around the peak pressure amplitude in Figure 6, the exact location of the acoustic focus is seen to be on-axis and 35 mm away from the tissue phantom and water interface (at z = 59.6 mm). Figure 7 shows the acoustic pressure amplitude profile along the radial direction in the focal plane (z = 59.6 mm).

Figure 8 shows the result of the temperature rise in the tissue phantom for a focal pressure amplitude of 1.11 MPa after 1 second of insonation. At t = 1 s, the maximum temperature rise is about one degree. The oval-shaped heated spot is about the same size as that of the acoustic focal area, which is more easily seen in the contour plot of the temperature as shown in Figure 9.



Figure 5: Surface plot of the acoustic intensity field, showing the acoustic energy to be concentrated to the focal region.

Figure 10 plots the heating and cooling curves at acoustic focus and 0.5 mm off acoustic focus in the focal plane. This is again for a focal pressure amplitude of 1.11 MPa. At these locations, the tissue heats up when insonated and then cools down through natural conduction. The result at the acoustic focus agrees well with the results presented in Ref. 8.

This example shows how to model tissue heating induced by focused ultrasound when the acoustic pressure at focus is well below acoustic cavitation threshold. Because the model does not take nonlinearity into account it can be solved in the frequency domain. Linear acoustic is valid in the limit where  $p << \rho c^2$ . The maximal pressure in the focal region is much smaller than the product  $\rho c^2 = 2.6 \ 10^9$  Pa and thus the linear assumption of the frequency domain is valid. At pressures comparable to and above the limit, acoustic energy is pumped out from the fundamental frequency to higher harmonics (Ref. 10). This nonlinear behavior can be modeled in the time domain using the *Nonlinear Acoustics (Westervelt)* domain feature of the *Pressure Acoustics, Transient* interface. In tissue, the absorption coefficient is close to  $f^{1.1}$  (for water, the power law for absorption is  $f^2$ ). Higher harmonics result in a narrower focal region and more energy dissipation. Therefore

at higher sound pressure levels, the current model would tend to underestimate the total energy deposition associated with the absorption of the ultrasonic wave and consequently also the maximum temperature rise at the focal region.



Figure 6: Acoustic pressure amplitude profile along the symmetry axis (r = 0).



Figure 7: Acoustic pressure amplitude profile along the radial direction in the focal plane.

10 | FOCUSED ULTRASOUND INDUCED HEATING IN TISSUE PHANTOM



Figure 8: Surface plot of the temperature rise in the tissue phantom after 1 second insonation for a focal pressure amplitude of 1.11 MPa.



Figure 9: Contour plot of the temperature rise in the tissue phantom after 1 second of insonation for a focal pressure amplitude of 1.11 MPa. The oval-shaped heated spot is about the same size as that of the acoustic focal area.



Figure 10: Heating and cooling curves at acoustic focus and 0.5 mm off-axis in the focal plane for 1 second of insonation with a focal pressure amplitude of 1.11 MPa.

The effect of energy dissipation serves to counteract the nonlinear effect and thus mitigate the waveform distortion (Ref. 10). Therefore, the assumption of linear progressive wave motion should remain good as long as the nonlinearity is relatively small, especially for hyperthermia applications. The simulation results show that this model provides a good estimate of both acoustic field and temperature rise for focal pressure up to 1.11 MPa. It not only simulates the heating and cooling behavior at the focal region but also gives other useful information, such as the shape of the heated area and the side-lobe heating effect at those surrounding locations with pressure maxima outside the focal region (which is the main lobe). The results also show that the side lobes in the intensity field are mitigated in the temperature response due to the smoothing effect of conduction, as shown in Figure 11. In general, this model suggests that the temperature change is roughly proportional to the acoustic intensity.



Figure 11: he temperature and intensity profiles show that the side lobes in the intensity field are mitigated in the temperature response due to the smoothing effect of the thermal conduction.

# Notes About the COMSOL Implementation

This model uses two studies. Study 1 solves for Pressure Acoustics in the Frequency Domain to obtain the acoustic pressure field. In Study 2, the heat source calculated from the solution of Study 1 is then used in a time-dependent analysis to model heating and cooling of the tissue phantom.

# References

1. K.J. Parker, "The Thermal Pulse Decay Technique for Measuring Ultrasonic Absorption Coefficients," *J. Acoust. Soc. Am.*, vol. 74, no. 5, pp. 1356–1361, 1983.

2. R.L. Clarke and G. ter Haar, "Temperature Rise Recorded during Lesion Formation by High Intensity Focused Ultrasound," *Ultrasound Med. Biol.*, vol. 23, no. 2, pp. 299–306, 1997.

3. N.T. Sanghvi, K. Hynynen, and F.L. Lizzi, "New Developments in Therapeutic Ultrasound," *IEEE Eng. Med. Biol. Mag.*, vol. 15, no. 6, pp. 83–92, 1996.

4. "Exposure Criteria for Medical Diagnostic Ultrasound: II. Criteria Based on All Known Mechanisms (Report No 140)," *Diagnostic Ultrasound Safety: A Summary of the Technical Report*, issued by the National Council on Radiation Protection and Measurements (NCRP), 2002.

5. P.N.T. Wells, "Biological Effects," *Biomedical Ultrasonics*, chap. 9, Academic Press, New York, 1977.

6. G. ter Haar, "Biological Effects of Ultrasound in Clinical Applications," Ultrasound: Its Chemical, Physical, and Biological Effects, K.S. Suslick, ed., VCH Publishers, New York, 1988.

7. W.L. Nyborg, "Biological Effects of Ultrasound: Development of Safety Guidelines. Part II: General Review," *Ultrasound Med. Biol.*, vol. 27, no. 3, pp. 301–333, 2001.

8. J. Huang, R.G. Holt, R.O. Cleveland, and R.A. Roy, "Experimental Validation of a Tractable Numerical Model for Focused Ultrasound Heating in Flow-through Tissue Phantoms," *J. Acoust. Soc. Am.*, vol. 116, no. 4, pt. 1, pp. 2451–2458, 2004.

9. F.A. Duck, Physical Properties of Tissue, Academic Press, 1990.

10. M.F. Hamilton and D.T. Blackstock eds., *Nonlinear Acoustics*, Academic Press, 1998.

**Application Library path:** Acoustics\_Module/Ultrasound/ ultrasound\_induced\_heating

# 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 🖚 2D Axisymmetric.

- 2 In the Select Physics tree, select Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr).
- 3 Click Add.
- 4 In the Select Physics tree, select Heat Transfer>Bioheat Transfer (ht).
- 5 Click Add.
- 6 Click  $\bigcirc$  Study.
- 7 In the Select Study tree, select Preset Studies for Some Physics Interfaces> Frequency Domain.

8 Click **M** Done.

# GLOBAL DEFINITIONS

Define the parameters used in the model.

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.

<b>3</b> In the table, enter the following settings:	
--	--

Name	Expression	Value	Description
d0	3.8[nm]	3.8E-9 m	Displacement amplitude of transducer
z_tissue	24.6[mm]	0.0246 m	Starting position of tissue phantom
то	293.7[K]	293.7 K	Initial temperature value
alpha_water	0.025[1/m]	0.025 l/m	Absorption coefficient of water
alpha_tissue	8.55[1/m]	8.55 l/m	Absorption coefficient of tissue phantom
fO	1[MHz]	IE6 Hz	Source frequency

Define a step function to turn off the acoustic source after 1 second of insonation. This is used for the transient thermal simulation. To improve convergence, define a smoothing transition zone that gently decreases the amplitude of the source to zero.

# Step | (step |)

- I In the Home toolbar, click f(X) Functions and choose Global>Step.
- 2 In the Settings window for Step, locate the Parameters section.
- **3** In the **From** text field, type 1.

- **4** In the **To** text field, type 0.
- **5** Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 0.005.

# GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Now, proceed to create the geometry. A sketch is depicted in Figure 2 and Figure 3. First, create the bowl-shaped transducer used as the acoustic source.

Circle I (c1)

- I In the **Geometry** toolbar, click  $\bigcirc$  **Circle**.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 62.64.
- 4 Locate the **Position** section. In the **z** text field, type 62.64.
- 5 Click 🔚 Build Selected.

Rectangle 1 (r1)

- I 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 35.
- 4 In the **Height** text field, type 10.69.
- 5 Click 틤 Build Selected.

Intersection 1 (int1)

- I In the Geometry toolbar, click 📁 Booleans and Partitions and choose Intersection.
- 2 Click in the Graphics window and then press Ctrl+A to select both objects.
- 3 In the Settings window for Intersection, click 틤 Build Selected.

Create the tissue phantom domain and add layers for the perfectly matched layer (PML).

Rectangle 2 (r2)

- I 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 48.6.
- 4 In the **Height** text field, type **75.5**.

- **5** Locate the **Position** section. In the **z** text field, type **z\_tissue**.
- 6 Click to expand the Layers section. In the table, enter the following settings:

Layer name	Thickness (mm)		
Layer 1	5		

- 7 Select the Layers to the right check box.
- 8 Select the Layers on top check box.
- 9 Clear the Layers on bottom check box.
- 10 Click 📄 Build Selected.

Create the water domains and, also here, add layers for the PML.

# Rectangle 3 (r3)

- I 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 48.6.
- **4** In the **Height** text field, type z\_tissue-10.69.
- 5 Locate the Position section. In the z text field, type 10.69.
- 6 Locate the Layers section. In the table, enter the following settings:

Layer name	Thickness (mm)		
Layer 1	5		

- 7 Select the Layers to the right check box.
- 8 Clear the Layers on bottom check box.
- 9 Click 틤 Build Selected.

# Circle 2 (c2)

- I In the **Geometry** toolbar, click  $\bigcirc$  **Circle**.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 15.
- 4 In the Sector angle text field, type 90.
- 5 Locate the Position section. In the z text field, type 0.80336.
- 6 Locate the Rotation Angle section. In the Rotation text field, type -90.

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

Layer name	Thickness (mm)	
Layer 1	5	

8 Click 틤 Build Selected.

Union I (uniI)

- I In the Geometry toolbar, click Pooleans and Partitions and choose Union.
- 2 Select the objects c2 and intl only.
- 3 In the Settings window for Union, click 틤 Build Selected.

Delete Entities I (dell)

- I In the Model Builder window, right-click Geometry I and choose Delete Entities.
- 2 On the object unil, select Boundary 10 only.



3 In the Settings window for Delete Entities, click 📳 Build Selected.

Finally, create an oval-shaped focal region where a finer mesh can be applied to resolve high gradient in the pressure field.

Ellipse I (el)

- I In the **Geometry** toolbar, click  $\bigcirc$  **Ellipse**.
- 2 In the Settings window for Ellipse, locate the Size and Shape section.
- 3 In the a-semiaxis text field, type 7.5.

- 4 In the **b-semiaxis** text field, type 1.5.
- 5 In the Sector angle text field, type 180.
- 6 Locate the Position section. In the z text field, type z\_tissue+35.
- 7 Locate the Rotation Angle section. In the Rotation text field, type 270.
- 8 Click 틤 Build Selected.

Form Union (fin)

- I In the Geometry toolbar, click 🟢 Build All.
- **2** Click the |+| **Zoom Extents** button in the **Graphics** toolbar.

The geometry should look like the one depicted in the figure below.



# DEFINITIONS

Define the Perfectly Matched Layer domains for the Pressure Acoustics simulation. The rational stretching type is more efficient than the polynomial stretching type in these open problems, whereas the polynomial is preferred in waveguide-like problems.

Perfectly Matched Layer I (pml1)

- I In the Model Builder window, expand the Component I (compl)>Definitions node.
- 2 Right-click Definitions and choose Perfectly Matched Layer.

**3** Select Domains 7–10 only.

It might be easier to select the domains by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



4 In the Settings window for Perfectly Matched Layer, locate the Geometry section.

5 From the Type list, choose Cylindrical.

6 Locate the Scaling section. From the Coordinate stretching type list, choose Rational.

# Perfectly Matched Layer 2 (pml2)

- I In the Definitions toolbar, click W Perfectly Matched Layer.
- **2** Select Domain 1 only.
- 3 In the Settings window for Perfectly Matched Layer, locate the Geometry section.
- 4 In the Center coordinate text field, type 0.8034[mm].
- 5 Locate the Scaling section. From the Coordinate stretching type list, choose Rational.

Now, set up the physics for Pressure Acoustics, Frequency Domain.

# PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

 In the Model Builder window, under Component I (compl) click Pressure Acoustics, Frequency Domain (acpr).

- 2 In the Settings window for Pressure Acoustics, Frequency Domain, locate the Sound Pressure Level Settings section.
- **3** From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.
- 4 Locate the Typical Wave Speed for Perfectly Matched Layers section. In the c<sub>ref</sub> text field, type 1483[m/s].

Now, specify the physical properties for the water domains.

Pressure Acoustics 1

- In the Model Builder window, under Component I (comp1)>Pressure Acoustics, Frequency Domain (acpr) click Pressure Acoustics I.
- **2** In the Settings window for Pressure Acoustics, locate the Pressure Acoustics Model section.
- **3** From the Fluid model list, choose User-defined attenuation.
- **4** In the  $\alpha$  text field, type alpha\_water.

Add a second Pressure Acoustics Material Model node for the tissue domains. You will define the tissue material below.

Pressure Acoustics 2

- I In the Physics toolbar, click 🔵 Domains and choose Pressure Acoustics.
- **2** Select Domains 5–7, 9, and 10 only.



- **3** In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.
- 4 From the Fluid model list, choose User-defined attenuation.
- **5** In the  $\alpha$  text field, type alpha\_tissue.

Define the normal displacement amplitude d0 at the surface of the ultrasound transducer.

Normal Displacement I

- I In the Physics toolbar, click Boundaries and choose Normal Displacement.
- 2 Select Boundary 32 only.
- 3 In the Settings window for Normal Displacement, locate the Normal Displacement section.
- **4** In the  $d_n$  text field, type d0.

Now, specify the physics of the Bioheat Transfer interface.

# **BIOHEAT TRANSFER (HT)**

- I In the Model Builder window, under Component I (compl) click Bioheat Transfer (ht).
- 2 In the Settings window for Bioheat Transfer, locate the Domain Selection section.
- 3 Click Clear Selection.
- 4 Select Domains 5 and 6 only.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Bioheat Transfer (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the T text field, type T0.

Add the absorbed acoustic energy as the domain heat source. The variable acpr.Q\_pw represents the dissipated power density for plane waves.

Heat Source 1

- I In the Physics toolbar, click 🔵 Domains and choose Heat Source.
- **2** Select Domains 5 and 6 only.
- 3 In the Settings window for Heat Source, locate the Heat Source section.
- **4** In the  $Q_0$  text field, type acpr.Q\_pw\*step1(t[1/s]-1).

Apply a constant temperature boundary condition on the computational boundaries of the thermal simulation.

#### Temperature 1

- I In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
- 2 Select Boundaries 9, 14, and 20 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the  $T_0$  text field, type T0.

The next step is to set up the materials used in the model. Use the default water material and define your own tissue material.

# 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>Water, liquid.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

# MATERIALS

Define the material properties of tissue phantom and apply it to the tissue phantom domains.

Tissue phantom

- I In the Model Builder window, under Component I (comp1) right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Tissue phantom in the Label text field.
- **3** Select Domains 5–7, 9, and 10 only.
- 4 Locate the Material Contents section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1044	kg/m³	Basic
Speed of sound	c	1568	m/s	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.59	W/(m·K)	Basic
Heat capacity at constant pressure	Ср	3710	J/(kg·K)	Basic

# PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

- In the Model Builder window, under Component I (compl)>Pressure Acoustics, Frequency Domain (acpr) click Pressure Acoustics I.
- 2 In the Settings window for Pressure Acoustics, locate the Model Input section.
- **3** In the T text field, type T0.

Pressure Acoustics 2

- I In the Model Builder window, click Pressure Acoustics 2.
- 2 In the Settings window for Pressure Acoustics, locate the Model Input section.
- **3** In the T text field, type T0.

# **MESH I - ACOUSTICS**

In the following steps, create a first mesh for the pressure acoustics simulation and then create a mesh for the thermal problem. Since the two physics are different, the meshes need to have different properties. For the acoustic simulation the mesh should resolve the wavelength of the problem. Using second-order (quadratic) elements, 5 mesh elements per wavelength are adequate.

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, type Mesh 1 Acoustics in the Label text field.

Free Triangular 1

- I In the **Mesh** toolbar, click **Free Triangular**.
- 2 In the Settings window for Free Triangular, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domains 2–6 only.

Use a finer mesh in the focal region to resolve the large gradients in the pressure field. Use 8 elements per wavelength here.

# Size I

- I Right-click Free Triangular I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 Click Clear Selection.
- **4** Select Domain 6 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.

7 In the associated text field, type 1568[m/s]/f0/30.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I Acoustics 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 1483[m/s]/f0/5.

Set up a mapped mesh in the perfectly matched layer region.

Mapped I

- I In the Mesh toolbar, click III Mapped.
- 2 In the Settings window for Mapped, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- **4** Select Domains 1 and 7–10 only.

Distribution I

- I Right-click Mapped I and choose Distribution.
- 2 Select Boundaries 1, 22, and 23 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 10.

# 5 Click 📗 Build All.

The mesh should look like the one in the figure below.



Next, create a coarser mesh for thermal simulation.

# MESH 2 - BIOHEAT TRANSFER

- I In the Mesh toolbar, click Add Mesh and choose Add Mesh.
- 2 In the Settings window for Mesh, type Mesh 2 Bioheat Transfer in the Label text field.

Free Triangular 1

- I In the Mesh toolbar, click Kree Triangular.
- 2 In the Settings window for Free Triangular, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domains 5 and 6 only.

# Size I

- I Right-click Free Triangular I and choose Size.
- **2** Select Domain 6 only.
- 3 In the Settings window for Size, locate the Element Size section.

- 4 Click the **Custom** button.
- 5 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 6 In the associated text field, type 0.18[mm].

#### Size

- I In the Model Builder window, under Component I (compl)>Meshes>Mesh 2 -Bioheat Transfer 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, type5.
- 5 In the Maximum element growth rate text field, type 1.2.
- 6 Click 📗 Build All.

The mesh should look like the one in the figure below.



First, solve the Pressure Acoustics physics model in the frequency domain using the finer mesh **Mesh I - Acoustics**.

# STUDY I

Step 1: Frequency Domain

- I In the Model Builder window, under Study I click Step I: Frequency Domain.
- 2 In the Settings window for Frequency Domain, locate the Study Settings section.
- 3 In the Frequencies text field, type f0.
- 4 Locate the Physics and Variables Selection section. In the table, clear the Solve for check box for Bioheat Transfer (ht).
- 5 Click to expand the Mesh Selection section to check that Mesh I is selected.
- 6 In the Model Builder window, click Study I.
- 7 In the Settings window for Study, type Study 1 Acoustics in the Label text field.
- 8 In the **Home** toolbar, click **= Compute**.

# ROOT

Now, add a transient analysis study type and solve the bioheat transfer model in the time domain using the coarser **Mesh 2 - Bioheat Transfer**. The acoustic model serves as input to calculate the heat source.

# ADD STUDY

- I In the Home toolbar, click 2 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

Preset Studies for Some Physics Interfaces>Time Dependent.

- 4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Pressure Acoustics, Frequency Domain (acpr).
- 5 Click Add Study in the window toolbar.
- 6 In the Home toolbar, click  $\sim 2$  Add Study to close the Add Study window.

# STUDY 2

Step 1: Time Dependent

- I In the Settings window for Time Dependent, locate the Study Settings section.
- 2 Click Range.
- 3 In the Range dialog box, type 0.2 in the Step text field.
- 4 In the **Stop** text field, type 5.

5 Click Replace.

These are the times where the solution is stored.

- 6 In the Settings window for Time Dependent, click to expand the Values of Dependent Variables section.
- 7 Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 8 From the Method list, choose Solution.
- 9 From the Study list, choose Study I Acoustics, Frequency Domain.
- 10 Click to expand the Mesh Selection section to check that Mesh 2 is selected.
- II In the Model Builder window, click Study 2.
- 12 In the Settings window for Study, type Study 2 Bioheat Transfer in the Label text field.

Specify the time stepping method and the tolerence for the internal time steps taken by the time-dependent solver.

Solution 2 (sol2)

- I In the Study toolbar, click **here** Show Default Solver.
- 2 In the Model Builder window, expand the Solution 2 (sol2) node, then click Time-Dependent Solver 1.
- **3** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 4 From the Maximum BDF order list, choose 5.
- 5 From the Maximum step constraint list, choose Constant.
- 6 In the Maximum step text field, type 0.02.
- **7** In the **Study** toolbar, click **= Compute**.

# RESULTS

Having solved the model, proceed to the results analysis. Follow the steps below to generate plots of the acoustic pressure and the temperature fields.

First, create a mirror dataset to better visualize the results in a full cut plane through the axisymmetric geometry.

Mirror 2D I

In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

Acoustic Pressure (acpr)

- I In the Model Builder window, under Results click Acoustic Pressure (acpr).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Mirror 2D I.
- 4 Locate the Plot Settings section. Select the x-axis label check box.
- 5 In the associated text field, type r coordinate (mm).
- 6 Select the y-axis label check box.
- 7 In the associated text field, type z coordinate (mm).
- 8 Clear the **Plot dataset edges** check box.

Use the wave color table to get an enhanced view of the acoustic field.

#### Surface 1

- I In the Model Builder window, expand the Acoustic Pressure (acpr) node, then click Surface I.
- 2 In the Acoustic Pressure (acpr) toolbar, click 💿 Plot.
- **3** Click the  $4 \rightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

The plot should like look that shown in Figure 4.

# Sound Pressure Level (acpr)

This second plot shows the sound pressure level (in the acoustics domain as well as in the PML region); it should look like that in the figure below. Note that the acoustic field is damped by about 150 dB in the PML region.



Deactivate plotting in the unphysical PML region for the remaining of the results analysis.

# Study I - Acoustics/Solution I (soll)

```
In the Model Builder window, under Results>Datasets click Study I - Acoustics/
Solution I (soll).
```

# Selection

- I In the Results toolbar, click 🐐 Attributes and choose Selection.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Click in the Graphics window and then press Ctrl+A to select all domains.
- **5** Select Domains 2–6 only.

Generate the acoustic intensity plot (the magnitude of the intensity vector) as shown in Figure 5.

Acoustic Intensity field

- I In the **Results** toolbar, click **2D Plot Group**.
- 2 In the Settings window for 2D Plot Group, type Acoustic Intensity field in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Mirror 2D I.
- 4 Locate the Plot Settings section. Select the x-axis label check box.
- **5** In the associated text field, type r coordinate (mm).
- 6 Select the y-axis label check box.
- 7 In the associated text field, type z coordinate (mm).
- 8 Clear the **Plot dataset edges** check box.

Surface 1

- I Right-click Acoustic Intensity field and choose Surface.
- 2 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
   Pressure Acoustics, Frequency Domain>Intensity>acpr.I\_mag Intensity magnitude W/m<sup>2</sup>.

**3** In the Acoustic Intensity field toolbar, click **O** Plot.

Next, generate a line plot of the acoustic pressure amplitude along the axis of symmetry, as shown in Figure 6.

# Pressure Amplitude Along Axial z-Axis

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Pressure Amplitude Along Axial z-Axis in the Label text field.
- 3 Click to expand the Title section. From the Title type list, choose Label.

#### Line Graph 1

I Right-click Pressure Amplitude Along Axial z-Axis and choose Line Graph.



- 3 In the Settings window for Line Graph, locate the x-Axis Data section.
- 4 From the Parameter list, choose Expression.

**2** Select Boundaries 3, 4, 6, 8, and 10–12 only.

- **5** In the **Expression** text field, type **z**.
- 6 In the Pressure Amplitude Along Axial z-Axis toolbar, click 🗿 Plot.

You can use the Zoom Box and Zoom Extents tools to zoom in around the acoustic focal point. The maximum pressure amplitude is located at z = 59.6 mm.

Define a line dataset and generate a plot of the acoustic pressure amplitude along the radial direction in the focal plane, as shown in Figure 7.

Cut Line 2D I

- I In the **Results** toolbar, click  $\frown$  **Cut Line 2D**.
- 2 In the Settings window for Cut Line 2D, locate the Line Data section.
- 3 In row Point I, set Z to 59.6.
- 4 In row **Point 2**, set **R** to 5 and **z** to 59.6.
- 5 Click 🗿 Plot.

Pressure Amplitude Along Radial Axis on the Focal Plane

I In the Results toolbar, click  $\sim$  ID Plot Group.

- 2 In the Settings window for ID Plot Group, type Pressure Amplitude Along Radial Axis on the Focal Plane in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Cut Line 2D I.
- 4 Locate the Title section. From the Title type list, choose Label.
- 5 Locate the Plot Settings section. Select the x-axis label check box.
- 6 In the associated text field, type r-coordinate (mm).

#### Line Graph I

- I Right-click Pressure Amplitude Along Radial Axis on the Focal Plane and choose Line Graph.
- 2 In the Settings window for Line Graph, locate the x-Axis Data section.
- 3 From the Parameter list, choose Expression.
- 4 In the **Expression** text field, type r.

You can increase the resolution of the plot. The resolution sets the number of interpolation points used inside each finite element.

- 5 Click to expand the Quality section. From the Resolution list, choose Finer.
- 6 In the Pressure Amplitude Along Radial Axis on the Focal Plane toolbar, click 🗿 Plot.

Line Graph 2

- I Right-click Line Graph I and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the x-Axis Data section.
- **3** In the **Expression** text field, type r.
- 4 Click to expand the Coloring and Style section. From the Color list, choose Blue.
- 5 In the Pressure Amplitude Along Radial Axis on the Focal Plane toolbar, click 💽 Plot.

Now, use the following steps to generate the temperature field plots as shown in the results.

First, create a mirror dataset for better visualization of the temperature field.

Mirror 2D 2

- I In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.
- 2 In the Settings window for Mirror 2D, locate the Data section.
- 3 From the Dataset list, choose Study 2 Bioheat Transfer/Solution 2 (sol2).

Temperature Rise at t = I s

I In the **Results** toolbar, click **2D Plot Group**.

- 2 In the Settings window for 2D Plot Group, type Temperature Rise at t = 1 s in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Mirror 2D 2.
- 4 From the Time (s) list, choose I.
- 5 Click to expand the Title section. From the Title type list, choose Manual.
- 6 In the Title text area, type Temperature rise at t = 1 s.
- 7 Locate the Plot Settings section. Select the x-axis label check box.
- 8 In the associated text field, type r coordinate (mm).
- 9 Select the y-axis label check box.
- **IO** In the associated text field, type z coordinate (mm).
- II Clear the **Plot dataset edges** check box.

# Surface 1

- I Right-click **Temperature Rise at t = I s** and choose **Surface**.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type T-TO.
- 4 Locate the Coloring and Style section. From the Color table list, choose Thermal.
- 5 In the Temperature Rise at t = I s toolbar, click **O** Plot.
- **6** Click the  $4 \rightarrow$  **Zoom Extents** button in the **Graphics** toolbar.

The plot should like look that shown in Figure 8.

Next, generate the isothermal contours plot as shown in Figure 9.

Isothermal Contours at t = I s

- I In the Home toolbar, click 🚛 Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, type Isothermal Contours at t = 1 s in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Mirror 2D 2.
- 4 From the Time (s) list, choose I.
- 5 Locate the Title section. From the Title type list, choose Manual.
- 6 In the Title text area, type Temperature Rise Contours at t = 1 s.
- 7 Locate the Plot Settings section. Select the x-axis label check box.
- 8 In the associated text field, type r coordinate (mm).
- 9 Select the y-axis label check box.

**IO** In the associated text field, type z coordinate (mm).

II Clear the **Plot dataset edges** check box.

# Contour I

- I Right-click Isothermal Contours at t = I s and choose Contour.
- 2 In the Settings window for Contour, locate the Expression section.
- **3** In the **Expression** text field, type T-TO.
- 4 Locate the Levels section. In the Total levels text field, type 50.
- 5 Locate the Coloring and Style section. From the Color table list, choose Thermal.
- 6 In the Isothermal Contours at t = I s toolbar, click **I** Plot.
- 7 Click the |+| Zoom Extents button in the Graphics toolbar.

Define two point datasets, one at the focus and the other at 0.5 mm off the acoustic focus. Then generate two point graphs (within a 1D Plot Group) of the temperature rise as function of time, as shown in Figure 10.

# Cut Point 2D 1

- I In the **Results** toolbar, click **Cut Point 2D**.
- 2 In the Settings window for Cut Point 2D, locate the Data section.
- 3 From the Dataset list, choose Study 2 Bioheat Transfer/Solution 2 (sol2).
- 4 Locate the Point Data section. In the R text field, type 0.
- 5 In the Z text field, type 59.6.

#### Cut Point 2D 2

- I Right-click Cut Point 2D I and choose Duplicate.
- 2 In the Settings window for Cut Point 2D, locate the Point Data section.
- **3** In the **R** text field, type **0.5**.

Temperature Rise vs. Time at Focus and 0.5 mm off Focus

- I In the Results toolbar, click  $\sim$  ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Temperature Rise vs. Time at Focus and 0.5 mm off Focus in the Label text field.
- 3 Locate the Data section. From the Dataset list, choose Study 2 Bioheat Transfer/ Solution 2 (sol2).
- 4 Locate the Title section. From the Title type list, choose Label.
- 5 Locate the Plot Settings section. Select the y-axis label check box.
- 6 In the associated text field, type Temperature Rise (K).

# Point Graph 1

- I Right-click Temperature Rise vs. Time at Focus and 0.5 mm off Focus and choose Point Graph.
- 2 In the Settings window for Point Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Point 2D I.
- 4 From the Solution parameters list, choose From parent.
- 5 Locate the y-Axis Data section. In the Expression text field, type T-TO.
- 6 Click to expand the Coloring and Style section. From the Color list, choose Red.
- 7 Click to expand the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

#### Legends

At acoustic focus

Point Graph 2

- I Right-click Point Graph I and choose Duplicate.
- 2 In the Settings window for Point Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Point 2D 2.
- 4 From the Solution parameters list, choose From parent.
- 5 Locate the Coloring and Style section. From the Color list, choose Blue.
- 6 Locate the Legends section. In the table, enter the following settings:

#### Legends

0.5 mm off acoustic focus

7 In the Temperature Rise vs. Time at Focus and 0.5 mm off Focus toolbar, click 💽 Plot.

**8** Click the **Comextents** button in the **Graphics** toolbar.

Define a line dataset and generate a 1D Line Graph of the temperature rise along the radial direction on the focal plane. Plot this after 1 and 2 seconds of insonation, as shown in Figure 11.

Cut Line 2D 2

- I In the Model Builder window, under Results>Datasets right-click Cut Line 2D I and choose Duplicate.
- 2 In the Settings window for Cut Line 2D, locate the Data section.

# 3 From the Dataset list, choose Study 2 - Bioheat Transfer/Solution 2 (sol2).

Normalized Temperature and Acoustic Intensity Profiles

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- **2** In the **Settings** window for **ID Plot Group**, type Normalized Temperature and Acoustic Intensity Profiles in the **Label** text field.
- 3 Locate the Title section. From the Title type list, choose Label.
- 4 Locate the Plot Settings section. Select the x-axis label check box.
- 5 In the associated text field, type Radial Distance from Focus (mm).
- 6 Select the y-axis label check box.
- **7** In the associated text field, type Normalized Temperature Rise and Acoustic Intensity.

Line Graph 1

- I Right-click Normalized Temperature and Acoustic Intensity Profiles and choose Line Graph.
- 2 In the Settings window for Line Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Line 2D 2.
- 4 From the Time selection list, choose From list.
- **5** In the **Times (s)** list, select **I**.
- 6 Locate the y-Axis Data section. In the Expression text field, type (T-T0)/1.018.
- 7 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 8 In the Expression text field, type r.
- 9 Locate the Coloring and Style section. From the Color list, choose Red.
- **IO** Click to expand the **Legends** section. Select the **Show legends** check box.
- II From the Legends list, choose Manual.

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

#### Legends

Temperature profile at t = 1 s

Line Graph 2

- I Right-click Line Graph I and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the x-Axis Data section.
- **3** In the **Expression** text field, type r.

4 Locate the Legends section. Clear the Show legends check box.

#### Line Graph 3

I In the Model Builder window, under Results>

Normalized Temperature and Acoustic Intensity Profiles right-click Line Graph I and choose Duplicate.

- 2 In the Settings window for Line Graph, locate the Data section.
- 3 In the Times (s) list, select 2.
- 4 Locate the y-Axis Data section. In the Expression text field, type (T-T0)/0.5751.
- 5 Locate the Coloring and Style section. From the Color list, choose Magenta.
- 6 Locate the Legends section. In the table, enter the following settings:

#### Legends

Temperature profile at t = 2 s

Line Graph 4

- I Right-click Line Graph 3 and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the x-Axis Data section.
- **3** In the **Expression** text field, type r.
- 4 Locate the Legends section. Clear the Show legends check box.

#### Line Graph 5

I In the Model Builder window, under Results>

Normalized Temperature and Acoustic Intensity Profiles right-click Line Graph I and choose Duplicate.

- 2 In the Settings window for Line Graph, locate the Data section.
- 3 From the Dataset list, choose Cut Line 2D I.
- 4 Click Replace Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (compl)>Pressure Acoustics, Frequency Domain>Intensity> acpr.I\_mag Intensity magnitude W/m<sup>2</sup>.
- 5 Locate the y-Axis Data section. In the Expression text field, type acpr.I\_mag/3.376e5.
- 6 Locate the Coloring and Style section. From the Color list, choose Blue.
- 7 Find the Line style subsection. From the Line list, choose Dotted.

8 Locate the Legends section. In the table, enter the following settings:

# Legends

Intensity profile

Line Graph 6

- I Right-click Line Graph 5 and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the x-Axis Data section.
- **3** In the **Expression** text field, type r.
- 4 Locate the Legends section. Clear the Show legends check box.
- **5** In the Normalized Temperature and Acoustic Intensity Profiles toolbar, click **O** Plot.