Model created in COMSOL Multiphysics 6.4

# Focused Ultrasound Induced Heating in Tissue Phantom

## *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.

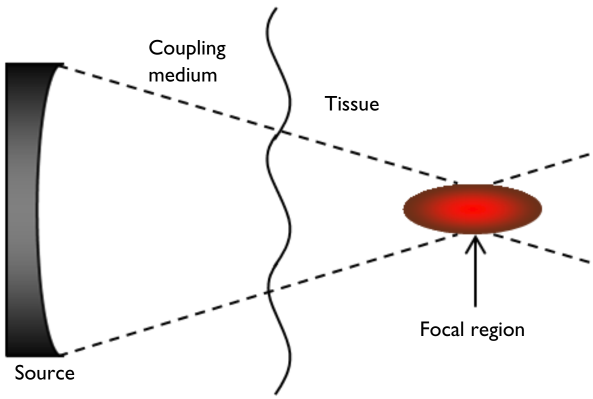



*Figure 1: Focused ultrasound makes selective and targeted heating possible: heating of tissues lying within the focal volume can be achieved with minimal damage to nearby healthy tissue and other structures lying elsewhere in the path of the beam.*

---

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

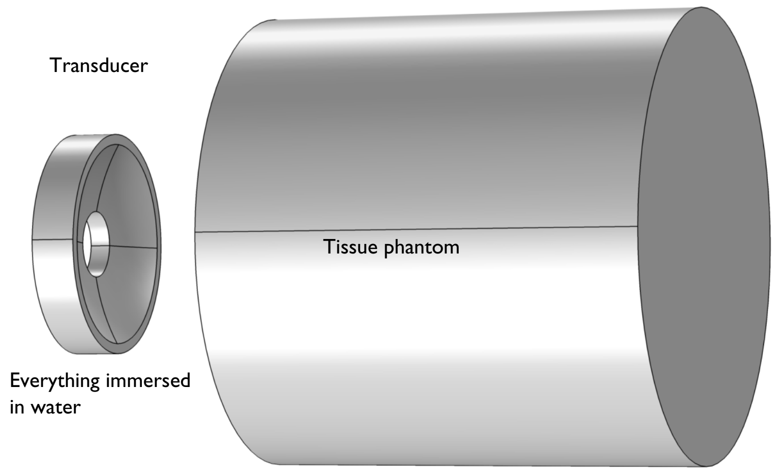



*Figure 2: The geometry of the acoustic transducer and tissue phantom. The transducer is bowl shaped with a hole in the center. Both the tissue phantom and the transducer are immersed in water. The axisymmetric geometry allows for a 2D axisymmetric simulation.*

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:

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

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

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

where $\alpha_{\mathrm{ABS}}$ 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

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

where $T$ is the temperature, $\rho$ is the density, $C_{\mathrm{p}}$ is the specific heat, $\mathrm{k}$ is the thermal conductivity, $\rho_{\mathrm{b}}$ is the density of blood, $C_{\mathrm{b}}$ is the specific heat of blood, $w_{\mathrm{b}}$ is the blood perfusion rate, $T_{\mathrm{b}}$ is the temperature of the blood, $Q$ is the heat source (the absorbed ultrasound energy calculated from Equation 2), and $Q_{\mathrm{met}}$ 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.

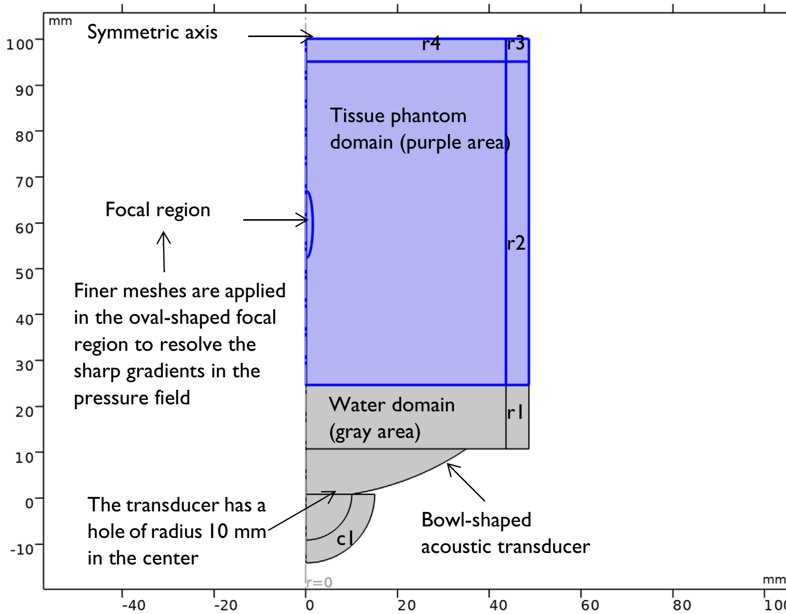



*Figure 3: Model geometry. Water domains are shown in blue and tissue phantom domains are shown in purple. Four cylindrical PMLs (*`r1-r4`*) and one spherical PML (*`c1`*) are used to absorb the outgoing waves.*

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.

TABLE 1: MATERIAL PROPERTIES USED IN THE MODEL.

| **Property** | **Density ($kg/m^3$)** | **Speed of sound (m/s)** | **Attenuation (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 |

## *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.

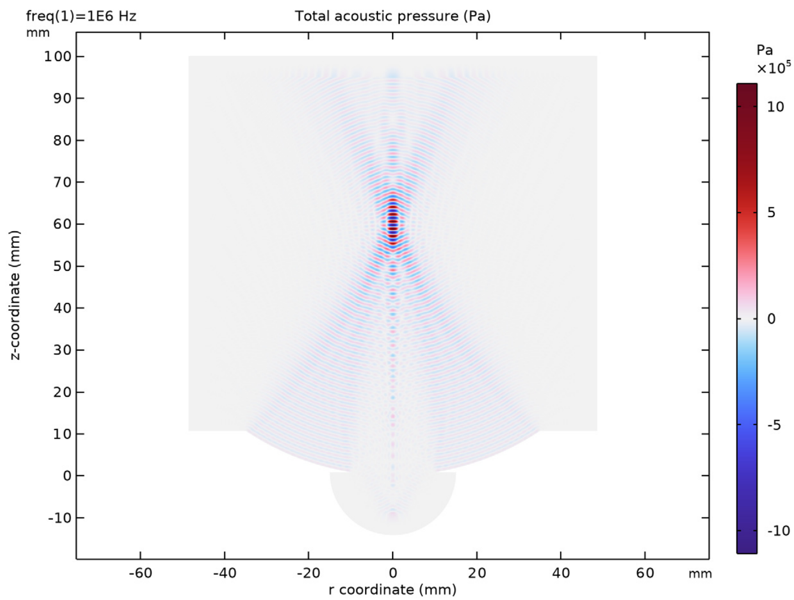



*Figure 4: The acoustic pressure field in the water and tissue domains.*

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.

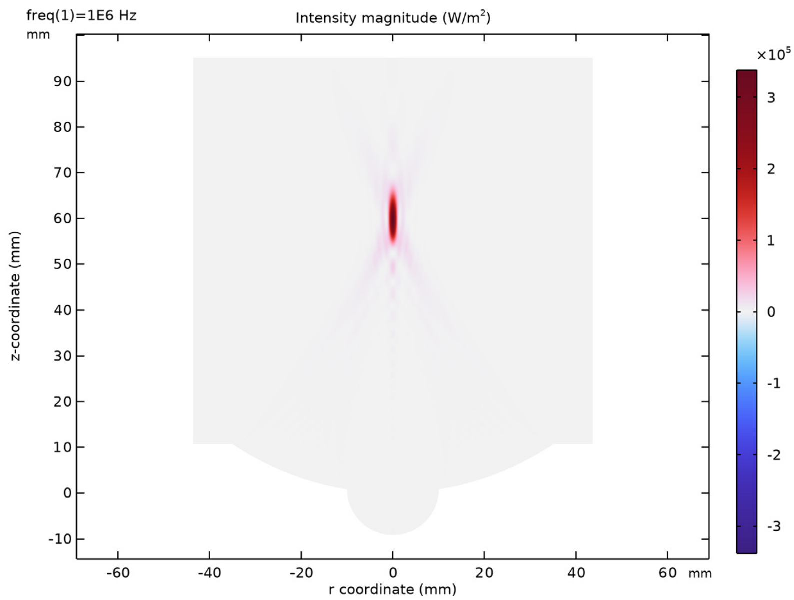



*Figure 5: Surface plot of the acoustic intensity field, showing the acoustic energy to be concentrated to the focal region.*

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.

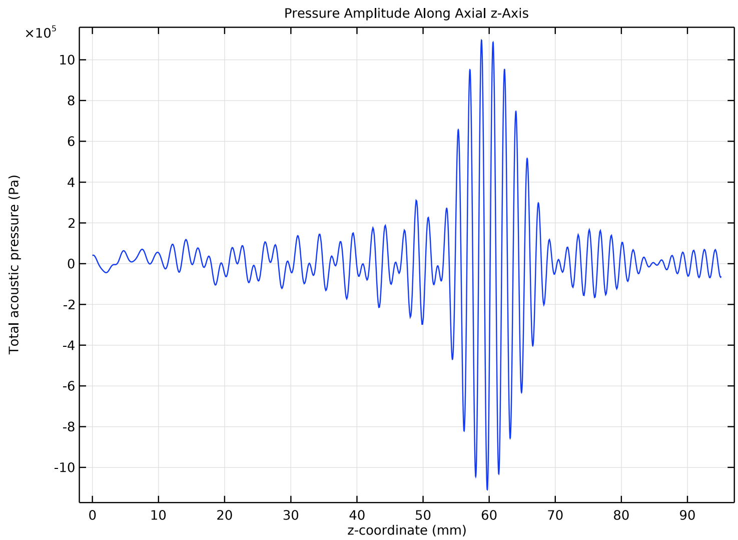



*Figure 6: Acoustic pressure amplitude profile along the symmetry axis (r = 0).*

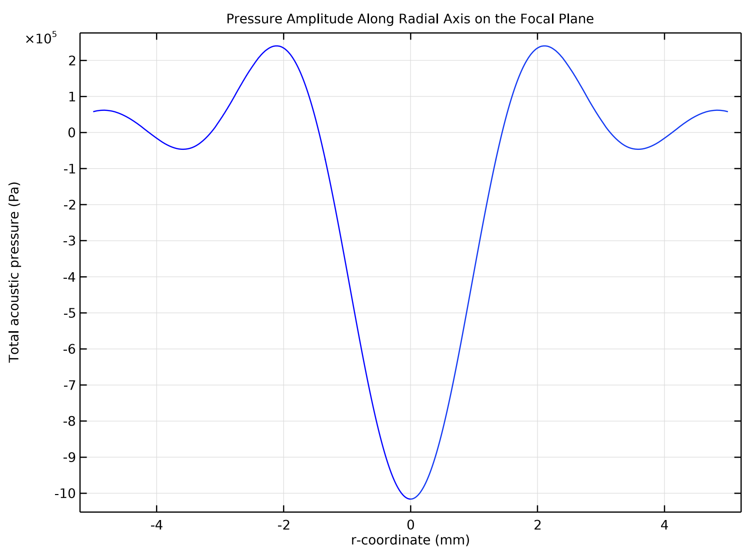



*Figure 7: Acoustic pressure amplitude profile along the radial direction in the focal plane.*

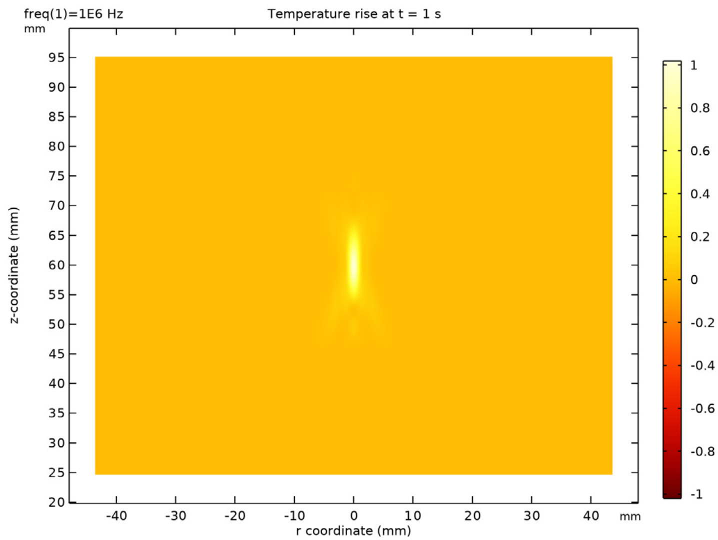



*Figure 8: Surface plot of the temperature rise in the tissue phantom after 1 second insonation for a focal pressure amplitude of 1.11 MPa.*

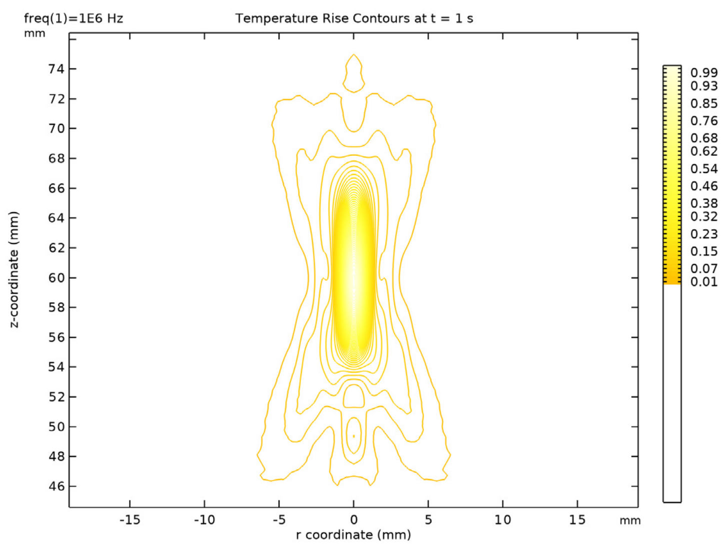



*Figure 9: Contour plot of the temperature rise in the tissue phantom after 1 second of insonation for a focal pressure amplitude of 1.11 MPa. The oval-shaped heated spot is about the same size as that of the acoustic focal area.*

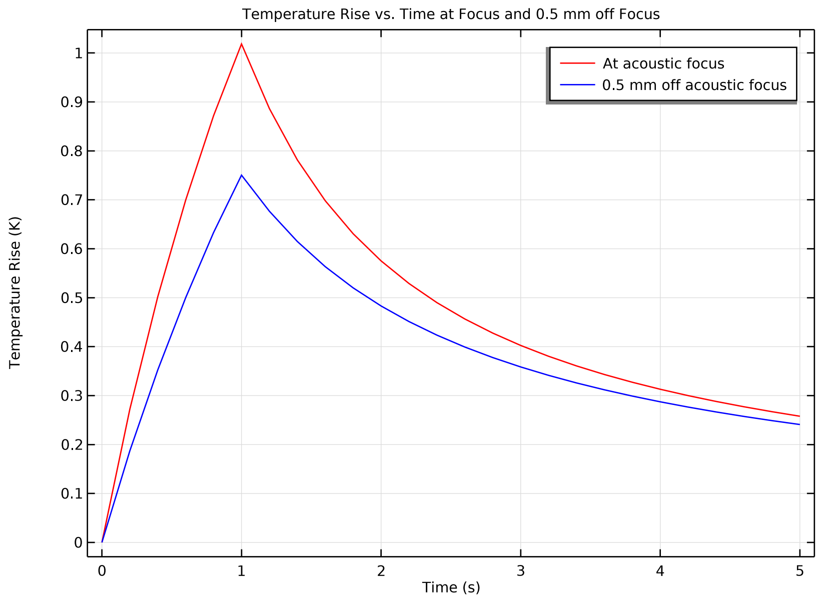



*Figure 10: Heating and cooling curves at acoustic focus and 0.5 mm off-axis in the focal plane for 1 second of insonation with a focal pressure amplitude of 1.11 MPa.*

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 sidelobe heating effect at those surrounding locations with pressure maxima outside the focal region (which is the main lobe). The results also show that the sidelobes 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.

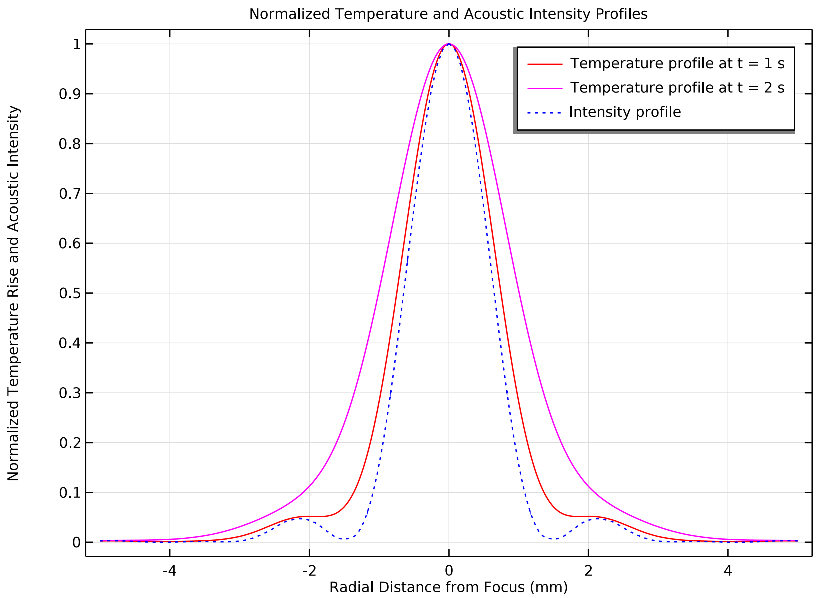



*Figure 11: The temperature and intensity profiles show that the sidelobes in the intensity field are mitigated in the temperature response due to the smoothing effect of the thermal conduction.*

## *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

1 In the **Model Wizard** window, click **2D Axisymmetric**.

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

**3** Click **Add**.

**4** In the **Select Physics** tree, select **Heat Transfer > Bioheat Transfer (ht)**.

**5** Click **Add**.

**6** Click **Study**.

**7** In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces > Frequency Domain**.

**8** Click **Done**.

## GLOBAL DEFINITIONS

Define the parameters used in the model.

### *Parameters 1*

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

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

**3** 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 |
| T0 | 293.7[K] | 293.7 K | Initial temperature value |
| alpha_water | 0.025[1/m] | 0.025 1/m | Absorption coefficient of water |
| alpha_tissue | 8.55[1/m] | 8.55 1/m | Absorption coefficient of tissue phantom |
| f0 | 1[MHz] | 1E6 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 1 (step1)*

**1** 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 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

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 1 (c1)*

1 In the **Geometry** toolbar, click **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)*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `35`.

4 In the **Height** text field, type `10.69`.

5 Click **Build Selected**.

*Intersection 1 (int1)*

1 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 **Build Selected**.

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

*Rectangle 2 (r2)*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `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** checkbox.

8 Select the **Layers on top** checkbox.

9 Clear the **Layers on bottom** checkbox.

10 Click **Build Selected**.

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

*Rectangle 3 (r3)*

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 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** checkbox.

8 Clear the **Layers on bottom** checkbox.

9 Click **Build Selected**.

*Circle 2 (c2)*

1 In the **Geometry** toolbar, click **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 **Build Selected**.

*Union 1 (uni1)*

**1** In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

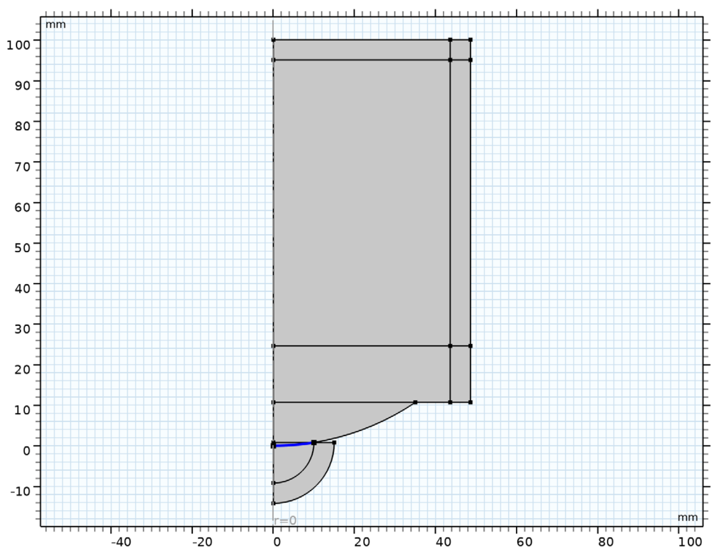
**2** Select the objects **c2** and **int1** only.

**3** In the **Settings** window for **Union**, click **Build Selected**.

*Delete Entities 1 (del1)*

**1** In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

**2** On the object **uni1**, 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 1 (e1)*

**1** In the **Geometry** toolbar, click **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 **Build Selected**.

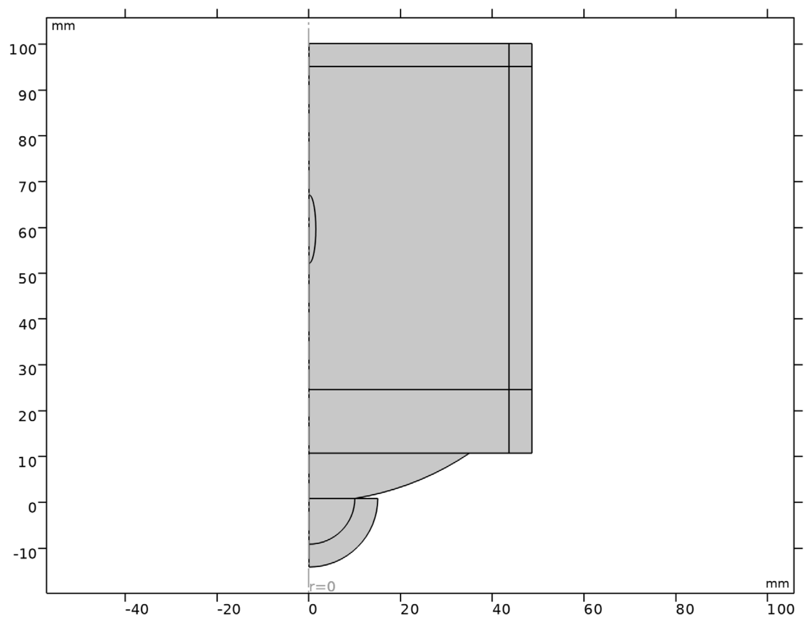
*Form Union (fin)*

**1** 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.

**3** In the **Model Builder** window, click **Form Union (fin)**.




**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.

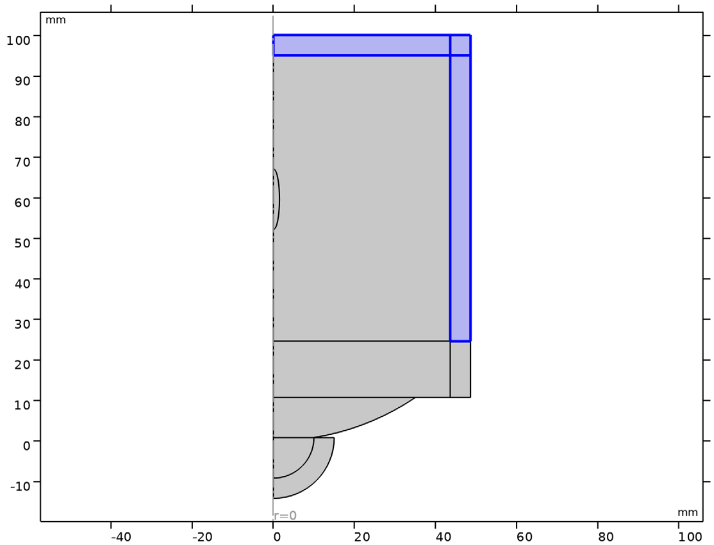
*Perfectly Matched Layer 1 (pml1)*

**1** In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Perfectly Matched Layer**.

3 Select Domains 7, 9, and 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)*

1 Right-click **Perfectly Matched Layer 1 (pml1)** and choose **Duplicate**.

It is best to define a separate Perfectly Matched Layer domain for each different material present in the model.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 8 only.

### *Perfectly Matched Layer 3 (pml3)*

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 1 only.

**3** In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

**4** From the **Coordinate stretching type** list, choose **Rational**.

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

### PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

**1** In the **Model Builder** window, under **Component 1 (comp1)** 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**.

Now, specify the physical properties for the water domains.

*Pressure Acoustics 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.

**2** In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.

**3** From the **Fluid model** list, choose **User-defined attenuation**.

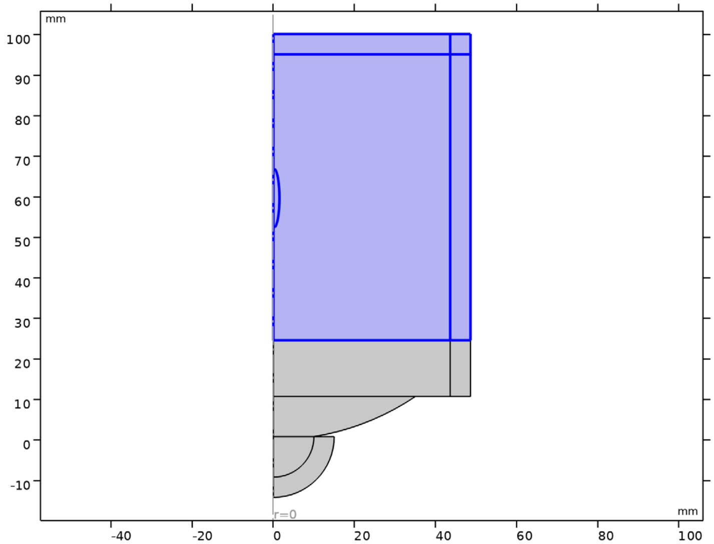
**4** In the $\alpha$ text field, type `alpha_water`.

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

*Pressure Acoustics 2*

**1** 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 1*

**1** 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)

**1** In the **Model Builder** window, under **Component 1 (comp1)** 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*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Bioheat Transfer (ht)** click **Initial Values 1**.
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*

1 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*

1 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

1 In the **Materials** 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 the **Add to Component** button in the window toolbar.
5 In the **Materials** 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*

**1** In the **Model Builder** window, under **Component 1 (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 | Cp | 3710 | J/(kg·K) | Basic |

## PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

*Pressure Acoustics 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.

**2** In the **Settings** window for **Pressure Acoustics**, locate the **Model Input** section.

**3** In the $T$ text field, type `T0`.

*Pressure Acoustics 2*

**1** 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 1 - 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.

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

**2** In the **Settings** window for **Mesh**, type `Mesh 1 - Acoustics` in the **Label** text field.

### *Free Triangular 1*

1. 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 1*

1. Right-click **Free Triangular 1** 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.
7. Select the **Maximum element size** checkbox. In the associated text field, type `1568[m/s]/f0/30`.

### *Size*

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1 - 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 1*

1. In the **Mesh** toolbar, click **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.

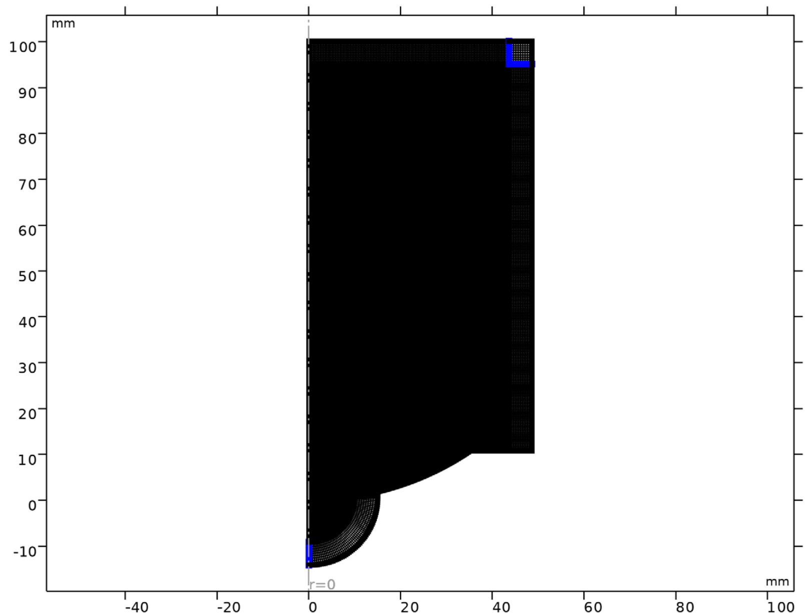
### *Distribution 1*

1. Right-click **Mapped 1** 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

**1** 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*

**1** 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 5 and 6 only.

#### *Size 1*

**1** Right-click **Free Triangular 1** 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.

6 Select the **Maximum element size** checkbox. In the associated text field, type `0.18[mm]`.

*Size*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Mesh 2 - Bioheat Transfer** click **Size**.

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

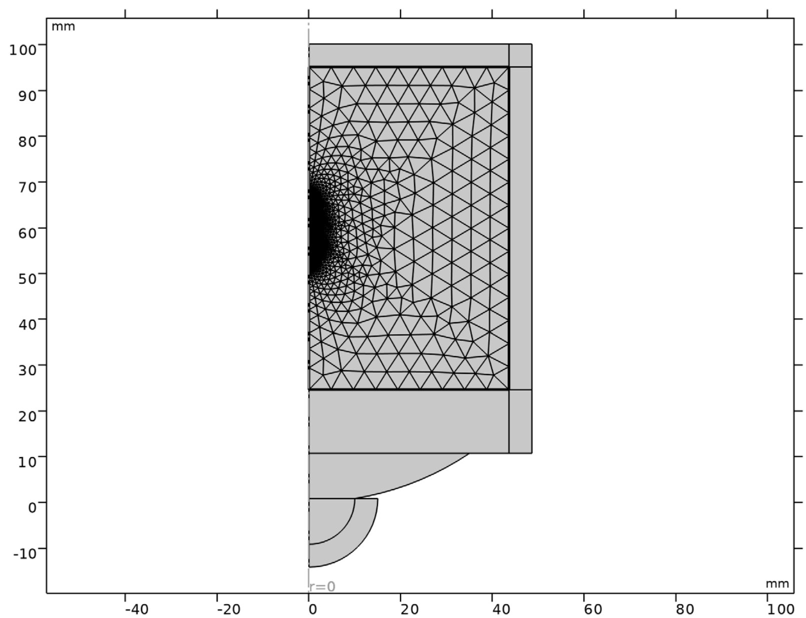
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `5`.

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 1 - Acoustics**.

### STUDY 1

*Step 1: Frequency Domain*

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

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

3 In the **Frequencies** text field, type `f0`.

4 Click to expand the **Mesh Selection** section to check that **Mesh 1** is selected.

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

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

7 In the **Study** 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

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Pressure Acoustics, Frequency Domain (acpr)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

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

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

### STUDY 2

*Step 1: Time Dependent*

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog, 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 1 - Acoustics, Frequency Domain**.

10 Click to expand the **Mesh Selection** section to check that **Mesh 2** is selected.

11 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 tolerance for the internal time steps taken by the time-dependent solver.

*Solution 2 (sol2)*

1 In the **Study** toolbar, click **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 1*

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

*Acoustic Pressure (acpr)*

1 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 1**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `r coordinate (mm)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `z-coordinate (mm)`.

7 Clear the **Plot dataset edges** checkbox.

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

### *Surface 1*

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.

2 In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

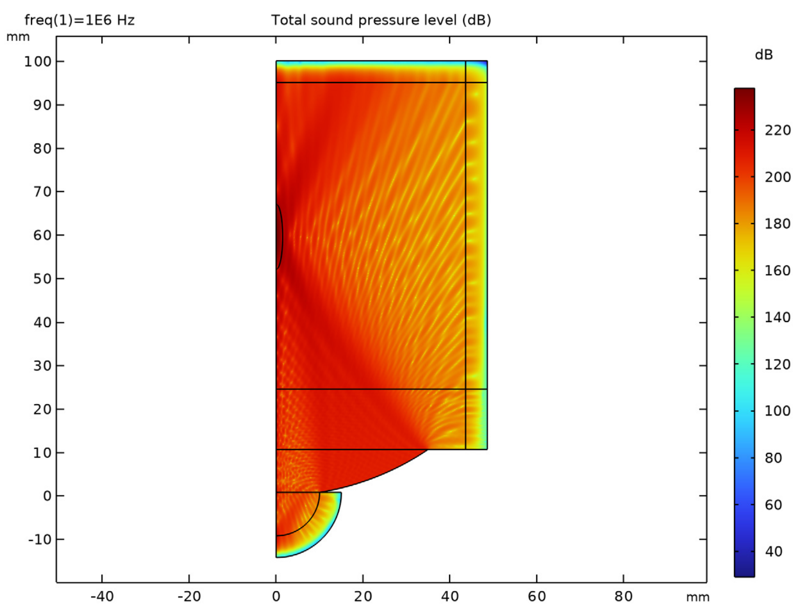
3 Click the **Zoom Extents** button in the **Graphics** toolbar.

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.

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.



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

### *Study 1 - Acoustics/Solution 1 (sol1)*

In the **Model Builder** window, under **Results** > **Datasets** click **Study 1 - Acoustics/ Solution 1 (sol1)**.

### *Selection*

1 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*

1 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 1**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `r coordinate (mm)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `z-coordinate (mm)`.

7 Clear the **Plot dataset edges** checkbox.

### *Surface 1*

1 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 1 (comp1)** > **Pressure Acoustics, Frequency Domain** > **Intensity** > **acpr.I_mag - Intensity magnitude - W/m²**.

3 In the **Acoustic Intensity field** toolbar, click **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*

1 In the **Results** toolbar, click **1D Plot Group**.

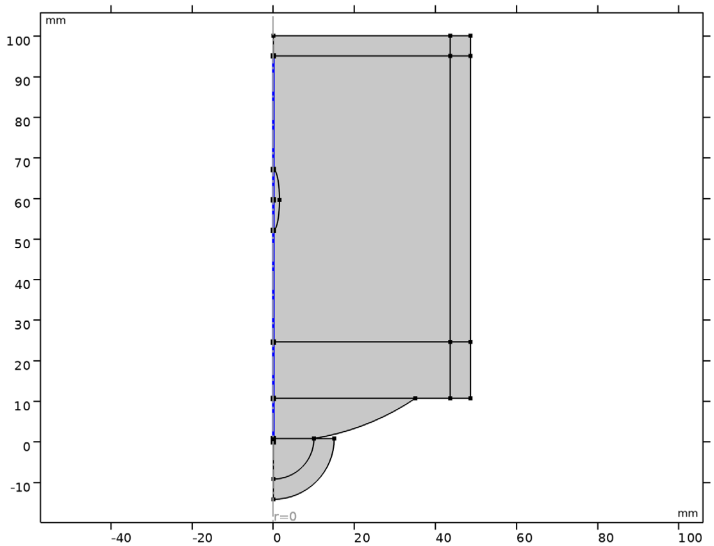
2 In the **Settings** window for **1D Plot Group**, type `Pressure Amplitude Along Axial z-Axis` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

*Line Graph 1*

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

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




3 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

4 From the **Parameter** list, choose **Expression**.

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 1*

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, 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*

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D 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 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `r-coordinate (mm)`.

*Line Graph 1*

1 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 **Evaluation settings** list, choose **Manual**.

6 From the **Resolution** list, choose **Finer**.

7 In the **Pressure Amplitude Along Radial Axis on the Focal Plane** toolbar, click **Plot**.

*Line Graph 2*

1 Right-click **Line Graph 1** 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*

1 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 = 1 s*

1 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 **1**.

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.

8 Select the **x-axis label** checkbox. In the associated text field, type `r coordinate (mm)`.

9 Select the **y-axis label** checkbox. In the associated text field, type `z-coordinate (mm)`.

10 Clear the **Plot dataset edges** checkbox.

### *Surface 1*

1 Right-click **Temperature Rise at t = 1 s** and choose **Surface**.

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

3 In the **Expression** text field, type `T-T0`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

5 In the **Temperature Rise at t = 1 s** toolbar, click **Plot**.

6 Click the **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 = 1 s*

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

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.

8 Select the **x-axis label** checkbox. In the associated text field, type `r coordinate (mm)`.

9 Select the **y-axis label** checkbox. In the associated text field, type `z-coordinate (mm)`.

10 Clear the **Plot dataset edges** checkbox.

*Contour 1*

1 Right-click **Isothermal Contours at t = 1 s** and choose **Contour**.

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

3 In the **Expression** text field, type `T-T0`.

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 = 1 s** toolbar, click **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*

1 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*

1 Right-click **Cut Point 2D 1** 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*

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D 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.

6 Select the **y-axis label** checkbox. In the associated text field, type `Temperature Rise (K)`.

*Point Graph 1*

1 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 1**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `T-T0`.

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** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

| Legends |
| --- |
| At acoustic focus |

*Point Graph 2*

1 Right-click **Point Graph 1** 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 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

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

| Legends |
| --- |
| 0.5 mm off acoustic focus |

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

7 Click the **Zoom Extents** 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*

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Cut Line 2D 1** 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*

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D 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.

5 Select the **x-axis label** checkbox. In the associated text field, type `Radial Distance from Focus (mm)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Normalized Temperature Rise and Acoustic Intensity`.

*Line Graph 1*

1 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 box, select **1**.

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**.

10 Click to expand the **Legends** section. Select the **Show legends** checkbox.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

| Legends |
| --- |
| `Temperature profile at t = 1 s` |

*Line Graph 2*

1 Right-click **Line Graph 1** 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** checkbox.

*Line Graph 3*

1 In the **Model Builder** window, under **Results** > **Normalized Temperature and Acoustic Intensity Profiles** right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 In the **Times (s)** list box, 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*

1 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** checkbox.

*Line Graph 5*

1 In the **Model Builder** window, under **Results** > **Normalized Temperature and Acoustic Intensity Profiles** right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Line 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Pressure Acoustics, Frequency Domain** > **Intensity** > **acpr.I_mag - Intensity magnitude - W/m²**.

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*

1 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** checkbox.

5 In the **Normalized Temperature and Acoustic Intensity Profiles** toolbar, click **Plot**.