LED Bulb Cooling

Introduction

This example simulates the thermodynamical behavior of a LED bulb, in order to evaluate the maximal temperature reached inside it while operating. It estimates the temperature of the system due to the heating of the LED chips, balanced by the cooling by a buoyancy driven air flow and radiation to the ambient surrounding.

In the past few years, light-emitting diode (LED) bulbs have progressively replaced incandescent light bulbs for both indoor and outdoor usage, due to their higher longevity and energy efficiency.

In LED bulbs, light emission is obtained by way of an electric current flowing through a semiconductor. In this physical process, called electroluminescence, 70% of the consumed electrical power is lost in heat. Therefore, to prevent from overheating, the geometry (size, number, and disposition of the LED chips, holes in the plastic bulb) and material properties (radiative properties of the aluminum PCB and plastic bulb) have to be optimized in the design of such devices.

In this model you compute the nonisothermal flow of air both inside and outside the bulb, by handling the coupling between energy transport — through conduction, radiation, and convection — and momentum transport induced by density variations of air.

Model Definition

The LED bulb considered in this study is made of acrylic plastic and has 6 holes at the top in order to let the air flow go through and remove heat.

It contains 8 lateral aluminum PCB supporting 18 LED chips each, and a top aluminum PCB supporting 12 LED chips. The total power of the bulb is 5 W. Radiation from the aluminum PCB to the ambient is handled in the computation. However, surface-to-surface radiation between the aluminum PCB and plastic bulb is neglected because most of the surfaces view factor is to the surrounding. Numerical simulations have shown that accounting for surface-to-surface radiation hardy changes the results while it increases significantly the computational cost.

The LED chips, made of epoxy resin, are welded on the aluminum PCB and dissipate each 70% of the electrical power they receive into heat, namely 3.5 W. The generated heat is transported to the surroundings through radiation, convection, and conduction. As air heats up, density and pressure changes induce a flow inside the bulb toward the top.

The base of the LED bulb is made of steel.

FLOW REGIME

Before starting to implement the model into COMSOL Multiphysics, it is recommended to estimate the flow regime. The Rayleigh, Reynolds, Grashof, and Prandtl dimensionless numbers compare the viscous, buoyancy, thermal, and inertia effects and allow to qualify the flow as laminar or turbulent, and to decide for modeling settings.

These numbers are based on the order of magnitude of the main physical and geometric parameters of the system, summarized in Table 1 below.

TABLE 1: ORDER OF MAGNITUDE OF THE PHYSICAL AND GEOMETRIC PARAMETERS OF THE SYSTEM.

PARAMETER	DESCRIPTION	VALUE
L	Typical dimension	10 cm
ΔT	Variation of temperature	50 K
μ	Dynamic viscosity of air	$1.8 \cdot 10^{-5}$ Pa·s
ρ	Density of air	1.2 kg/m^3
$\nu = \mu/\rho$	Cinematic viscosity of air	$1.5 \cdot 10^{-5}$ m$^2 \cdot$s^{-1}
α_p	Coefficient of thermal expansion of air	$3 \cdot 10^{-3}$ K^{-1}
α	Thermal diffusivity of air	$2 \cdot 10^{-5}$ m$^2 \cdot$s^{-1}
g	Gravity acceleration	10 m·s^{-2}

Grashof and Prandtl Numbers

The Grashof number compares the effect of buoyancy and viscous transportation, and is defined as

$$\mathrm{Gr} = \frac{g\alpha_p \Delta T L^3}{\nu^2}$$

Thus, for this model, take gravity into account. Considering the parameters from Table 1, gives $\mathrm{Gr} = 10^7$, which is below the critical number $\mathrm{Gr_c} = 10^9$ over which the flow turns to be turbulent.

The Prandtl number indicates whether a stationary thermal regime is reachable or not by comparing the velocity of thermal effects with the one of the hydrodynamical effects:

$$\mathrm{Pr} = \frac{\nu}{\alpha}$$

The parameters from Table 1 give $\mathrm{Pr} = 0.75$. This Prandtl number of order 1 indicates that the system will reach a thermal stationary state.

Rayleigh Number

The Rayleigh number $Ra = Pr \cdot Gr$ compares the buoyancy effects and the thermal diffusivity. It is needed to compute and estimate the typical velocity U of the fluid.

When the fluid is only led by buoyancy forces, its typical velocity U_0 can be estimated from the Grashof number as

$$U_0 = \sqrt{Gr}\frac{\nu}{L}$$

where ν/L is the speed due to viscous friction.

The velocity U, however, takes into account the thermal diffusivity. The Rayleigh number $Ra = Pr(U_0 L/\nu)^2$ also reads $Ra = U^2 L/\alpha$ with

$$U = \frac{U_0}{\sqrt{Pr}}$$

U provides a more accurate velocity estimate as it takes the viscosity into account. The parameters in Table 1 give $U = 0.28$ m/s.

Reynolds Number

This number compares the effects of inertial forces to the viscous forces to which the fluid is submitted. It indicates whether the regime is laminar or turbulent, and is defined as

$$Re = \nu^{-1} U L$$

For a flow in a pipe, the threshold value from laminar to turbulent is about $Re = 10^3$. Using the parameters in Table 1 and the velocity estimate obtained from the Rayleigh number gives $Re = 10^3$, which corresponds to the transition threshold.

Flow Regime

All the dimensionless numbers computed above are summarized in Table 2 below.

TABLE 2: INDICATORS OF THE FLOW REGIME.

Dimensionless Number	Value
Reynolds	$Re = 10^3$
Grashof	$Gr = 10^7$
Prandtl	$Pr = 0.75$
Rayleigh	$Ra = 10^7$

Here the values of the Reynolds, the Grashof, and the estimated velocity justify to work under the hypothesis of a laminar flow. See Buoyancy Flow in Water tutorial for more details about the dimensionless numbers and their use.

SYMMETRY

The LED geometry and the operating conditions have two obvious symmetry planes. As the flow regime is moderate, it is reasonable to assume that it shows the same symmetry. Hence it is possible to solve the equations on only the quarter of the geometry showed in Figure 1 and Figure 2, and to use symmetry conditions for the fluid flow and heat transfer equations. Numerical simulations using the full geometry have confirmed that the symmetry assumption for the flow does not modify the estimation of the cooling performance of the LED bulb.

GEOMETRY

Figure 1 and Figure 2 show the LED bulb geometry used in the model. The LED chips and the aluminum PCB are represented in orange and brown, respectively.
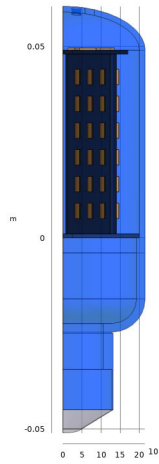
Figure 1: A quarter of the LED bulb geometry. Front view, showing the LED chips position on the lateral PCB.
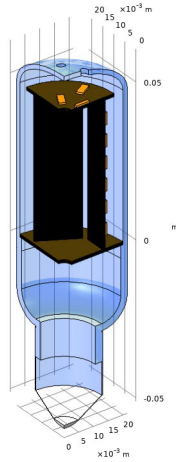
Figure 2: A quarter of the LED bulb geometry. View showing the LED chips position on the top PCB and the hole at the top of the bulb.

You can find LED bulbs similar to the one studied in this model in any hardware store. This section presents an experimental-like setup made up of two LED bulbs fixed on a vertical board (see Figure 3).

The LEDs are powered through a 230 V AC domestic current that is converted to an adapted current by the LED bulb electronic part that does not play any role in this thermodynamical study.

Figure 3: Device for experimental measurement.

The temperature is measured through a basic infrared camera (see Figure 4). Once the device reaches a stationary state, the maximal temperature observed through this disposal is around 72°C.
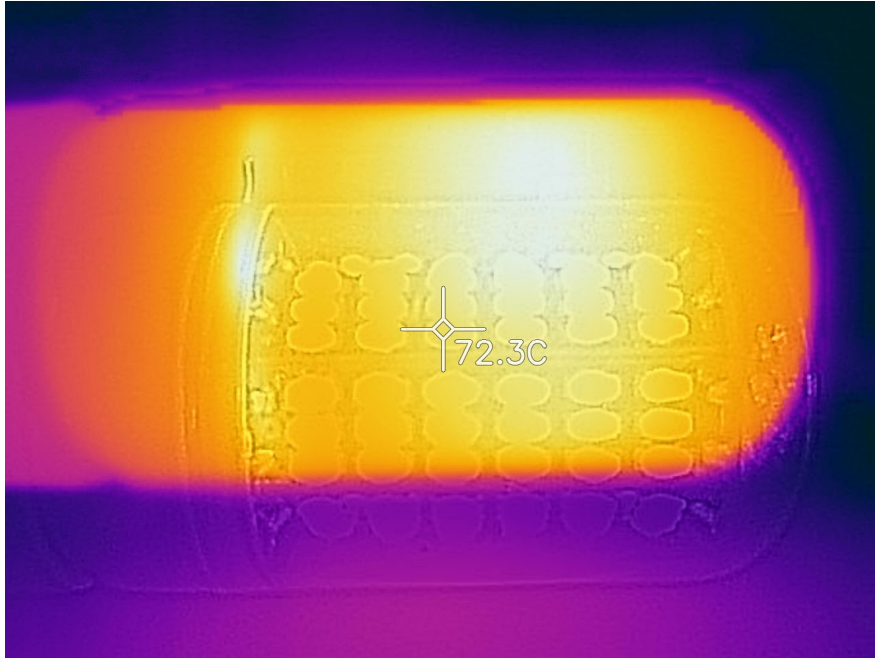
Figure 4: Temperature field given by an infrared camera.

Even if a more precise experiment would be needed in order to have a precise quantitative agreement between model and reality. This, nonetheless, acts like a proof of concept. In particular it confirmed, in agreement with the numerical results, that the cooling of the LED bulb is hardly influence by the orientation (vertical or horizontal).

Results and Discussion

The temperature distribution in the LED bulb is shown in Figure 5. A maximal temperature of about **76**°C is obtained on the LED chips. This value is close to the experimental value displayed in Figure 4.
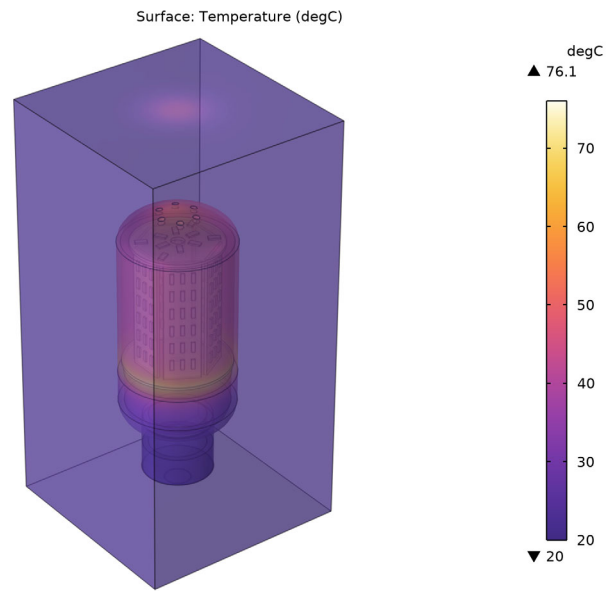
Figure 5: Temperature distribution with nominal power 5 W.

The velocity magnitude of the natural convection flow is shown in Figure 6. Its maximal value is about 0.2 m/s, which is close to the value estimated for U in the Flow regime section.

Slice: Velocity magnitude (m/s)
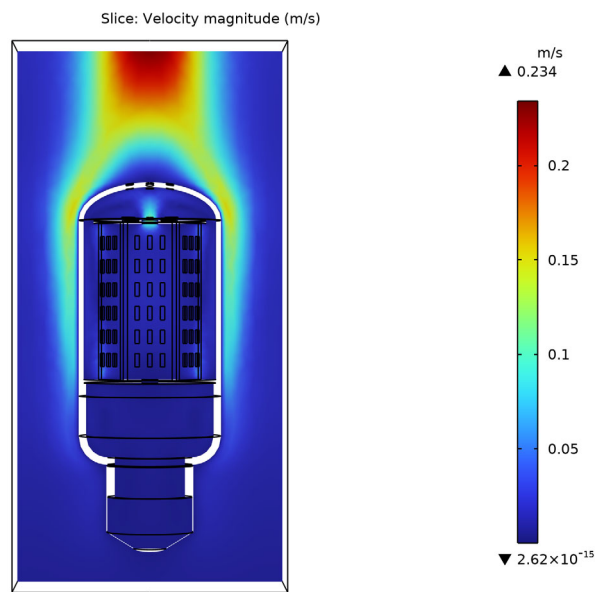
Figure 6: Velocity field magnitude.

Notes About the COMSOL Implementation

For this kind of buoyancy driven flow, a mainly vertical velocity is expected. Hence, in order to converge more easily toward the stationary solution, you can define a small positive value for vertical velocity initial condition. According to the calculation above, the initial velocity can be taken around 0.01 m/s.

Also due to the main direction of the flow, the normal dynamic pressure on the studied box of air surrounding the led bulb should not be ignored. The dynamic pressure is approximated to $P_{\text{flow}} = \rho_{\text{ref}} w^2 /2$, where w is the vertical velocity, and contributes to the total pressure, in addition to the reference and the static pressure that are already accounted for by default.

Application Library path: `Heat_Transfer_Module/`
`Power_Electronics_and_Electronic_Cooling/led_bulb`

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 ▨ **3D**.

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 Click ➡ **Study**.

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

6 Click ☑ **Done**.

First, enter the parameters that will be needed to run the simulation.

GLOBAL DEFINITIONS

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
power	5[W]/4	1.25 W	Nominal power of a quarter of a 5W LED bulb
eta	0.3	0.3	Efficiency
heat_losses	(1-eta)*power	0.875 W	Part of the power dissipated through heat

Import 1 (imp1)

1 In the **Home** toolbar, click ⊞ **Import**.

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

3 Click ▸ **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file
led_bulb.mphbin.

5 Click ⊞ **Import**.

You should now see the geometry shown in Figure 1 and Figure 2. You can hide some
parts to visualize the interior of the LED bulb.

6 Click the 👁 **Click and Hide** button in the **Graphics** toolbar.

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

8 On the object **imp1**, select Boundary 44 only.

9 On the object **imp1**, select Boundary 261 only.

10 On the object **imp1**, select Boundary 43 only.

11 On the object **imp1**, select Boundary 41 only.

12 On the object **imp1**, select Boundary 42 only.

To simplify further selections, create some sets from now.

LED

1 In the **Geometry** toolbar, click 🔲 **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type LED in the **Label** text field.

3 On the object **imp1**, select Domains 8–28 and 30–41 only.

PCB

1 In the **Geometry** toolbar, click 🔲 **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type PCB in the **Label** text field.

3 Click the 👁 **Click and Hide** button in the **Graphics** toolbar.

4 On the object **imp1**, select Domain 1 only.

5 On the object **imp1**, select Domain 3 only.

6 Click the 👁 **Click and Hide** button in the **Graphics** toolbar.

7 On the object **imp1**, select Domains 4, 5, 7, and 29 only.

PCB Radiating Boundaries

1 In the **Geometry** toolbar, click 🗏 **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type PCB Radiating Boundaries in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 21, 49, 53, 82–87, 98, 126–131, 143, 169–174, 188, 189, and 249–260 only.

Outlet

1 In the **Geometry** toolbar, click 🗏 **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type Outlet in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundary 13 only.

Inlet

1 In the **Geometry** toolbar, click 🗏 **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type Inlet in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **imp1**, select Boundaries 3, 44, and 261 only.

5 In the **Geometry** toolbar, click ▦ **Build All**.

Before adding the materials, define which part of the system is solid and which one is fluid, in order to clarify the physics context.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.

3 Click 📋 **Paste Selection**.

4 In the **Paste Selection** dialog box, type imp1: 1 in the **Selection** text field.

5 Click **OK**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 In the list, choose **2**, **3 (hidden)**, **4**, **5**, **6**, **7**, **8**, **9**, **10**, **11**, **12**, **13**, **14**, **15**, **16**, **17**, **18**, **19**, **20**, **21**, **22**, **23**, **24**, **25**, **26**, **27**, **28**, **29**, **30**, **31**, **32**, **33**, **34**, **35**, **36**, **37**, **38**, **39**, **40**, and **41**.

5 Click ━ **Remove from Selection**.

6 Select Domain 1 only.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Steel AISI 4340**.

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

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

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

7 In the tree, select **Built-in>Aluminum**.

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

9 In the tree, select **Built-in>Acrylic plastic**.

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

11 In the tree, select **Material Library>Epoxies, Acrylics and PMMA>Filled epoxy resin (X238)> Filled epoxy resin (X238) [solid]**.

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

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

MATERIALS

Bulb base

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat1)**.

2 In the **Settings** window for **Material**, type Bulb base in the **Label** text field.

Air (mat2)

1 Click the 🔲 **Transparency** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Air (mat2)**.

3 In the **Settings** window for **Material**, in the **Graphics** window toolbar, click ▼ next to 👁 **View Unhidden**, then choose **View All**.

4 Select Domain 1 only.

PCB

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat3)**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **PCB**.

Plastic bulb

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Acrylic plastic (mat4)**.

2 In the **Settings** window for **Material**, type Plastic bulb in the **Label** text field.

3 Select Domain 3 only.

LED chips

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Filled epoxy resin (X238) [solid] (mat5)**.

2 In the **Settings** window for **Material**, type LED chips in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **LED**.

Add the heat capacity at constant pressure to the list of properties of the epoxy resin material.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	1200[J/kg/K]	J/(kg·K)	Basic

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domain 1 only.

Since the density variation is not small, the flow cannot be regarded as incompressible. Therefore set the flow to be compressible.

3 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

4 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

As the flow is driven by buoyancy, gravity matters.

5 Select the **Include gravity** check box.

6 Select the **Use reduced pressure** check box.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

0	x
0	y
1e-2	z

Open Boundary 1

1 In the **Physics** toolbar, click ▭ **Boundaries** and choose **Open Boundary**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Open Boundary**, locate the **Boundary Condition** section.

4 In the f_0 text field, type `-0.5*spf.rhoref*w^2`.

Open Boundary 2

1 In the **Physics** toolbar, click ▭ **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 13, 44, and 261 only.

Symmetry 1

1 In the **Physics** toolbar, click ▭ **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 2, and 7 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

Heat Source 1

1 In the **Physics** toolbar, click ▭ **Domains** and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **LED**.

4 Locate the **Heat Source** section. From the **Heat source** list, choose **Heat rate**.

5 In the P_0 text field, type `heat_losses`.

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **PCB Radiating Boundaries**.

4 In the list, select **21**.

5 Locate the **Surface-to-Ambient Radiation** section. From the ε list, choose **User defined**. In the associated text field, type 0.9.

Open Boundary 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.

2 Select Boundaries 3, 13, 44, and 261 only.

Symmetry 1

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

2 Select Boundaries 1, 2, 4, 5, 7, 9, 10, 14, 18, 22, 55, 56, 96, 100, 138, 184, 190, 194, 198, 202, 206, and 210 only.

Define the mesh manually to have enough cells within the thin parts of the geometry and precise boundary layers, without having too many elements.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

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

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

3 From the **Predefined** list, choose **Coarse**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type 4e-4.

Size 1

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

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

3 From the **Predefined** list, choose **Coarser**.

Size 2

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

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

3 Click the **Custom** button.

4 Click to collapse the **Element Size Parameters** section. Click to expand the **Element Size Parameters** section. Select the **Minimum element size** check box.

5 Click ▦ **Build All**.

6 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Temperature (ht)

The plots shown in Figure 5 and Figure 6 can be generated after the entire geometry is recovered using a **Mirror 3D**. Follow the next steps to create these plots.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, click 🔲 **Plot**.

4 Click to expand the **Advanced** section. Select the **Remove elements on the symmetry plane** check box.

5 Click 🔲 **Plot**.

Mirror 3D 2

1 In the **Results** toolbar, click ▦ **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 Locate the **Advanced** section. Select the **Remove elements on the symmetry plane** check box.

6 Click 🔲 **Plot**.

DEFINITIONS

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

Hide for Geometry 1

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

2 Right-click **Hide for Geometry 1** and choose **Disable**.

Hide for Geometry 2

In the **Model Builder** window, right-click **Hide for Geometry 2** and choose **Disable**.

RESULTS

Mirror 3D 2

1 In the **Model Builder** window, under **Results>Datasets** click **Mirror 3D 2**.

2 In the **Settings** window for **Mirror 3D**, click ▣ **Plot**.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 Locate the **Color Legend** section. Select the **Show units** check box.

5 Select the **Show maximum and minimum values** check box.

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

4 In the **Temperature (ht)** toolbar, click ▣ **Plot**.

5 Click the ▣ **Transparency** button in the **Graphics** toolbar.

6 Click the ⬦ **Show Axis Orientation** button in the **Graphics** toolbar.

7 Click the ▦ **Show Grid** button in the **Graphics** toolbar.

Velocity (spf)

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Velocity (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

4 Select the **Show units** check box.

Slice

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

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 In the **Planes** text field, type 1.

4 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 2**.

5 In the **Velocity (spf)** toolbar, click **Plot**.

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 In the **Velocity (spf)** toolbar, click **Plot**.

5 Click the **Go to YZ View** button in the **Graphics** toolbar.

6 Click the **Transparency** button in the **Graphics** toolbar.