Heat Generation in a Disc Brake

Introduction

This example models the heat generation and dissipation in a disc brake of an ordinary car during panic braking and the following release period. As the brakes slow the car, they transform kinetic energy into thermal energy, resulting in intense heating of the brake discs. If the discs overheat, the brake pads stop working and, in a worst-case scenario, can melt. Braking power starts to fade already at temperatures above 600 K.

In the model, the car (1800 kg) initially travels at 25 m/s (90 km/h) when the driver brakes hard for 2 s, causing the vehicle's eight brake pads to slow the car down at a rate of 10 m/s^2. The wheels are assumed not to skid against the road surface. After this period of time, the driver releases the brake and the car travels at 5 m/s for an additional 8 s without any braking. The questions to analyze with the model are:

- How hot do the brake discs and pads become during the braking stage?
- How much do they cool down during the subsequent rest?

Model Definition

Model the brake disc as a 3D solid with shape and dimensions as in Figure 1. The disc has a radius of 0.14 m and a thickness of 0.013 m.
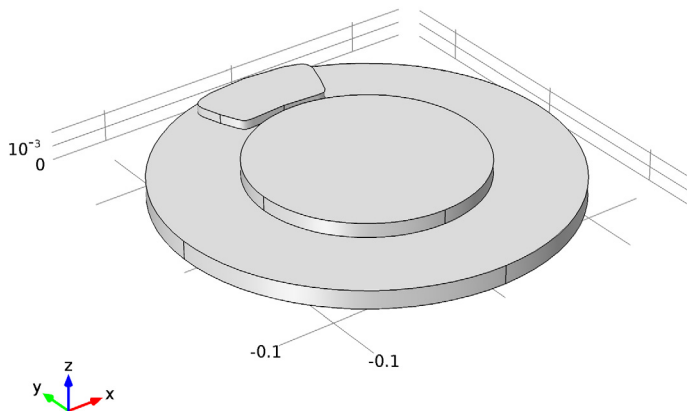

Figure 1: Model geometry, including disc and pad.

Neglecting drag and other losses outside the brakes, the brakes' retardation power is given by the negative of the time derivative of the car's kinetic energy:

$$P = -\frac{d}{dt}\left(\frac{mv^2}{2}\right) = -mv\frac{dv}{dt} = -mR^2\omega(t)\alpha$$

Here m is the car's mass, v denotes its speed, R equals the wheel radius (0.25 m), ω is the angular velocity, and α is the angular acceleration. The acceleration is constant in this case, so $\omega(t) = \omega_0 + \alpha t$.

By definition, the retardation power equals the negative of the work per unit time done by the friction forces on the discs at the interfaces between the pads and the discs for the eight brakes. You can calculate this work as eight times an integral over the contact surface of a single brake pad. The friction force per unit area, $\mathbf{f}_f$, is approximately constant over the surface and is directed opposite the disc velocity vector, $\mathbf{v}_d = v_d\mathbf{e}_\varphi$, where $\mathbf{e}_\varphi$ denotes a unit vector in the azimuthal (angular) direction and the magnitude of $\mathbf{v}_d$ at the distance r from the center equals $v_d(r, t) = \omega(t)r$. Thus, writing $\mathbf{f}_d = f_f\mathbf{e}_\varphi$ gives the following result for the retardation power:

$$P = -8\iint\mathbf{f}_f dA \cdot \mathbf{v}_d = 8f_f(t)\omega(t)\iint r dA$$

You can approximate the last integral with the pad's area, A (0.0035 m^2), multiplied by the distance from the center of the disc to the pad's center of mass, r_m (0.1143 m).

Combining the two expressions for P gives the following result for the magnitude of the friction force, f_f:

$$f_f = -\frac{mR^2\alpha}{8r_m A}$$

(Note that α is negative during retardation.)

Under the previously stated idealization that retardation is due entirely to friction in the brakes, the heat power generated per unit contact area at time t and the distance r from the center becomes

$$q(r, t) = -\mathbf{f}_f \cdot \mathbf{v}_d(r, t) = -\frac{mR^2\alpha}{8r_m A}r(\omega_0 + \alpha t)$$

The disc and pad dissipate the heat produced at the boundary between the brake pad and the disc by convection and radiation. This example models the rotation as convection in the disc. The local disc velocity vector is

$$\mathbf{v}_d = \omega(t)(-y, x)$$

The model also includes heat conduction in the disc and the pad through the transient heat transfer equation

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q - \rho C_P \mathbf{u} \cdot \nabla T$$

where k represents the thermal conductivity (W/(m·K)), C_p is the specific heat capacity (J/(kg·K)), and Q is the heating power per unit volume (W/m^3), which in this case is set to zero.

At the boundary between the disc and the pad, the brake produces heat according to the expression for q given earlier. The heat dissipation from the disc and pad surfaces to the surrounding air is described by both convection and radiation

$$q_{diss} = -h(T - T_{ref}) - \varepsilon \sigma(T^4 - T_{ref}^4)$$

In this equation, h equals the convective film coefficient (W/(m^2·K)), ε is the material's emissivity, and σ is the Stefan-Boltzmann constant ($5.67 \cdot 10^{-8}$ W/(m^2·K^4)).

To calculate the convective film coefficient as a function of the vehicle speed, v, use the following formula (Ref. 1):

$$h = \frac{0.037k}{l} \mathrm{Re}^{0.8} \mathrm{Pr}^{0.33} = \frac{0.037k}{l} \left(\frac{\rho l v}{\mu}\right)^{0.8} \left(\frac{C_p \mu}{k}\right)^{0.33}$$

Here l is the disc's diameter. The material properties—the thermal conductivity, k, the density, ρ, the viscosity, μ, and the specific heat capacity, C_p—are those for air.

Table 1 summarizes the thermal properties, which come from (Ref. 1). You calculate the density of air at a reference temperature of 300 K using the ideal gas law.

TABLE 1: MATERIAL PROPERTIES

PROPERTY	DISC	PAD	AIR
ρ (kg/m^3)	7870	2000	1.170
C_p (J/(kg·K))	449	935	1100
k (W/(m·K))	82	8.7	0.026
ε	0.28	0.8	-
μ (Pa·s)	-	-	$1.8 \cdot 10^{-5}$

Results and Discussion

The surface temperatures of the disc and the pad vary with both time and position. At the contact surface between the pad and the disc the temperature increases when the brake is engaged and then decreases again as the brake is released. You can best see these results in COMSOL Multiphysics by generating an animation. Figure 2 displays the surface temperatures just before the end of the braking. A "hot spot" is visible at the contact between the brake pad and disc, just at the pad's edge—this is where the temperature could become critical during braking. The figure also shows the temperature decreasing along the rotational trace after the pad. During the rest, the temperature becomes significantly lower and more uniform in the disc and the pad.
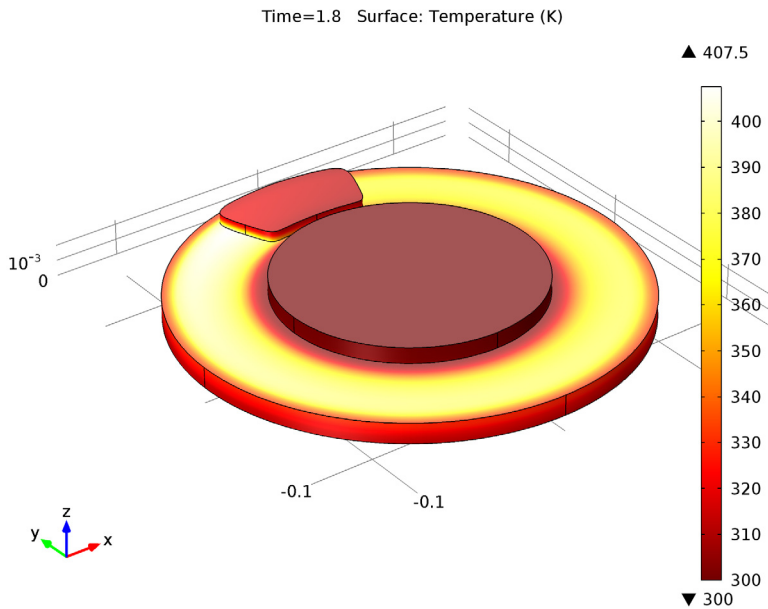

Figure 2: Surface temperature of the brake disc and pad just before releasing the brake (t = 1.8 s).

To investigate the position of the hot spot and the time of the temperature maximum, it is helpful to plot temperature versus time along the line from the center to the pad's edge depicted in Figure 3. The result is displayed in Figure 4. You can see that the maximum temperature is approximately 415 K. The hot spot is positioned close to the radially outer edge of the pad. The highest temperature occurs approximately 1 s after engaging the brake.
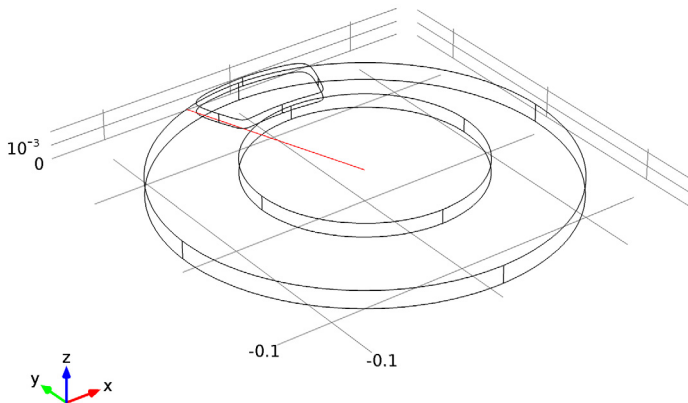
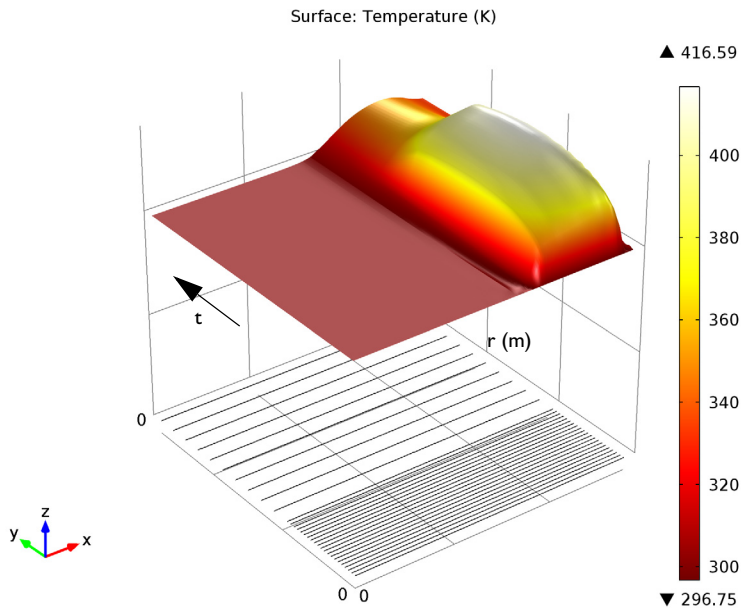*Figure 3: The radial line probed in the temperature vs. time plot in Figure 4.*

Surface: Temperature (K)

Figure 4: Temperature profile along the line indicated in Figure 3 at the disc surface (z = 0.013 m) as a function of time.

To investigate how much of the generated heat is dissipated to the air, study the surface integrals of the produced heat and the dissipated heat. These integrals give the total heat flux (J/s) for heat production, Q_{prod}, and heat dissipation, Q_{diss}, as functions of time for the brake disc. The time integrals of these two quantities, W_{prod} and W_{diss}, give the total heat (J) produced and dissipated, respectively, in the brake disc. Figure 5 shows a plot of the total produced heat and dissipated heat versus time. You can see that 8 s after disengagement the brake has dissipated only a fraction of the produced heat. The plot indicates that the resting time must be extended significantly to dissipate all the generated heat.
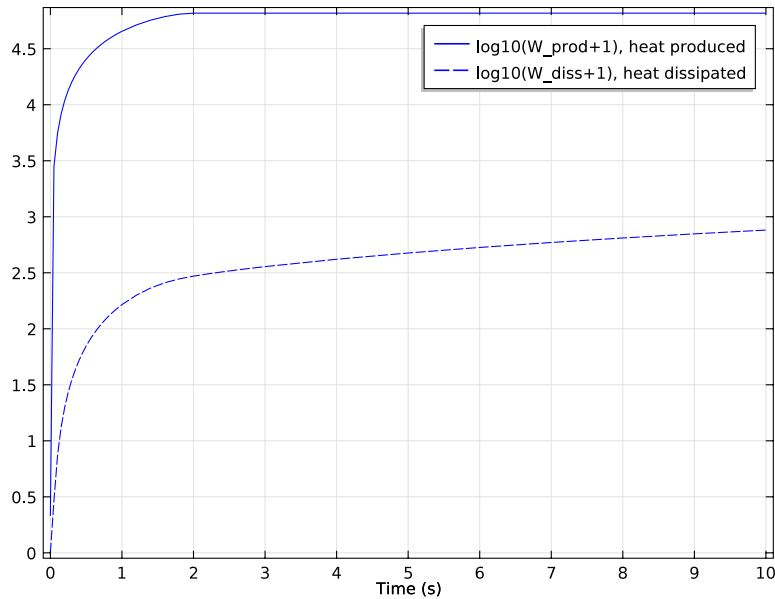

Figure 5: Comparison of total heat produced (solid line) and dissipated (dashed).

The results of this model can help engineers investigate how much abuse, in terms of specific braking sequences, a certain brake-disc design can tolerate before overheating. It is also possible to vary the parameters affecting the heat dissipation and investigate their influence.

Reference

1. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, eq. 9.88; material properties from appendix A2.

Model Library path: Heat_Transfer_Module/Tutorial_Models/brake_disc

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

6 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

Define the global parameters by loading the corresponding text file provided.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file brake_disc_parameters.txt.

GEOMETRY 1

Cylinder 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 0.14.

4 In the **Height** edit field, type 0.013.

5 Click the **Build Selected** button.

Cylinder 2

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

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 0.08.

4 In the **Height** edit field, type 0.01.

5 Locate the **Position** section. In the **z** edit field, type 0.013.

6 Click the **Build Selected** button.

Work Plane 1

1 Right-click **Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Work Plane** section.

3 In the **z-coordinate** edit field, type 0.013.

4 Click the **Show Work Plane** button.

Bézier Polygon 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Cubic** button.

4 Find the **Control points** subsection. In row **1**, set **yw** to 0.135.

5 In row **2**, set **xw** to 0.02.

6 In row **2**, set **yw** to 0.135.

7 In row **3**, set **xw** to 0.05.

8 In row **3**, set **yw** to 0.13.

9 In row **4**, set **xw** to 0.04.

10 In row **4**, set **yw** to 0.105.

11 Find the **Added segments** subsection. Click the **Add Cubic** button.

12 Find the **Control points** subsection. In row **2**, set **xw** to 0.03.

13 In row **2**, set **yw** to 0.08.

14 In row **3**, set **xw** to 0.035.

15 In row **3**, set **yw** to 0.09.

16 In row **4**, set **xw** to 0.

17 In row **4**, set **yw** to 0.09.

18 Find the **Added segments** subsection. Click the **Add Cubic** button.

19 Find the **Control points** subsection. In row **2**, set **xw** to -0.035.

20 In row **2**, set **yw** to 0.09.

21 In row **3**, set **xw** to -0.03.

22 In row **3**, set **yw** to 0.08.

23 In row **4**, set **xw** to -0.04.

24 In row **4**, set **yw** to 0.105.

25 Find the **Added segments** subsection. Click the **Add Cubic** button.

26 Find the **Control points** subsection. In row **2**, set **xw** to -0.05.

27 In row **2**, set **yw** to 0.13.

28 In row **3**, set **xw** to -0.02.

29 In row **3**, set **yw** to 0.135.

30 Click the **Close Curve** button.

To complete the pad cross section, you must make the top-left and top-right corners sharper. Do so by changing the weights of the Bézier curves.

31 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 1 (cubic)**.

32 Find the **Weights** subsection. In the **3** edit field, type 2.5.

33 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 4 (cubic)**.

34 Find the **Weights** subsection. In the **2** edit field, type 2.5.

35 Click the **Build Selected** button.

Extrude 1

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Work Plane 1** and choose **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (m)
0.0065

4 Click the **Build Selected** button.

The model geometry is now complete.

Next, define some selections of certain boundaries. You will use them when defining the settings for model couplings, boundary conditions, and so on.

DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 1** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Disc faces` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Select Boundaries 1, 2, 4–6, 8, 13–15, and 18 only.

Explicit 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 2** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Pad faces` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 Select Boundaries 9, 10, 12, 16, and 17 only.

Explicit 3

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 3** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Contact surface` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

6 From the **Geometric entity level** list, choose **Boundary**.

To select the contact surface boundary, it is convenient to temporarily switch to wireframe rendering.

7 Click the **Wireframe Rendering** button on the Graphics toolbar.

8 Select Boundary 11 only.

To select this boundary, you typically need to click twice before right-clicking; the first click highlights the pad's top surface.

9 Click the **Wireframe Rendering** button on the Graphics toolbar.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 11 only.

Integration 2

1 Right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

**4** From the **Selection** list, choose **Disc faces**.

Integration 3

1 Right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Pad faces**.

Define a step function for use in variables.

Step 1

1 Right-click **Definitions** and choose **Functions>Step**.

2 In the **Step** settings window, click to expand the **Smoothing** section.

3 In the **Size of transition zone** edit field, type 2*0.01.

4 Click the **Plot** button.

step1
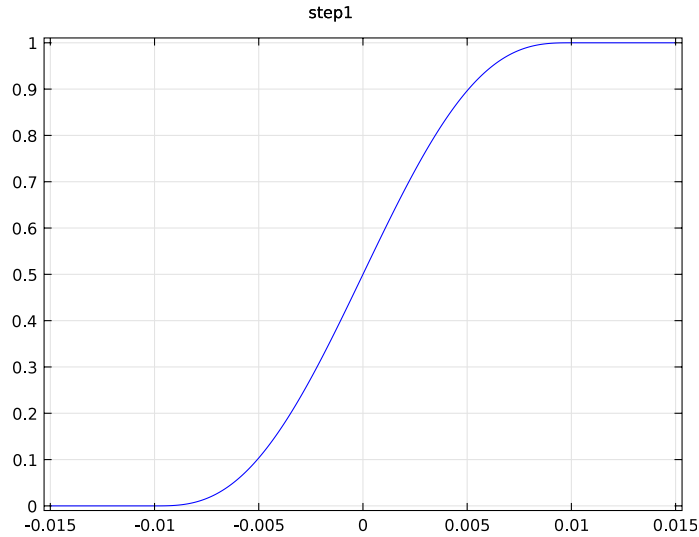
Define some variables by loading the corresponding text file provided.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file
`brake_disc_variables.txt`.

In the table, add variables for the produced heating power and dissipated heating power defined in terms of these variables and the integration operators you defined previously:

5 In the table, enter the following settings:

Name	Expression	Description
Q_prod	intop1(q_prod)	Produced heating power
Q_diss	intop2(-q_d_disc)+intop 3(-q_d_pad)	Total dissipated heating power

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose 📃**rial**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	k_disc
Density	rho	rho_disc
Heat capacity at constant pressure	Cp	C_disc

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Select Domain 3 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	k_pad
Density	rho	rho_pad
Heat capacity at constant pressure	Cp	C_pad

HEAT TRANSFER IN SOLIDS

Heat Transfer in Solids 1

In the **Model Builder** window, expand the **Model 1>Heat Transfer in Solids** node.

Translational Motion 1

1 Right-click **Heat Transfer in Solids 1** and choose **Translational Motion**.

2 Select Domains 1 and 2 only.

3 In the **Translational Motion** settings window, locate the **Translational Motion** section.

4 In the $\mathbf{u}_{trans}$ table, enter the following settings:

-y*omega	x
x*omega	y
0	z

Symmetry 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose **Symmetry**.

2 Select Boundary 3 only.

Heat Flux 1

1 Right-click **Heat Transfer in Solids** and choose **Heat Flux**.

2 In the **Heat Flux** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Disc faces**.

4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.

5 In the h edit field, type h_air.

6 In the T_{ext} edit field, type T_air.

Heat Flux 2

1 Right-click **Heat Transfer in Solids** and choose **Heat Flux**.

2 In the **Heat Flux** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pad faces**.

4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.

5 In the h edit field, type h_air.

6 In the T_{ext} edit field, type T_air.

Boundary Heat Source 1

1 Right-click **Heat Transfer in Solids** and choose **Boundary Heat Source**.

2 In the **Boundary Heat Source** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Contact surface**.

4 Locate the **Boundary Heat Source** section. In the Q_b edit field, type q_prod.

Initial Values 1

1 In the **Model Builder** window, under **Model 1>Heat Transfer in Solids** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

3 In the T edit field, type T_air.

Surface-to-Ambient Radiation 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose **Surface-to-Ambient Radiation**.

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

3 From the **Selection** list, choose **Disc faces**.

4 Locate the **Surface-to-Ambient Radiation** section. From the ε list, choose **User defined**. In the associated edit field, type `e_disc`.

5 In the T_{amb} edit field, type `T_air`.

Surface-to-Ambient Radiation 2

1 Right-click **Heat Transfer in Solids** and choose **Surface-to-Ambient Radiation**.

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

3 From the **Selection** list, choose **Pad faces**.

4 Locate the **Surface-to-Ambient Radiation** section. From the ε list, choose **User defined**. In the associated edit field, type `e_pad`.

5 In the T_{amb} edit field, type `T_air`.

To compute the heat produced and the heat dissipated, integrate the corresponding heating-power variables, `Q_prod` and `Q_diss`, over time. For this purpose, define two ODEs using a Global Equations node.

6 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 Right-click **Heat Transfer in Solids** and choose **Global>Global Equations**.

2 In the **Global Equations** settings window, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t)	Initial value (u_0)	Initial value (u_t0)	Description
W_prod	W_prodt-Q_prod	0	0	Produced heat
W_diss	W_disst-Q_diss	0	0	Dissipated heat

Here, `W_prodt` (`W_disst`) is COMSOL Multiphysics syntax for the time derivative of `W_prod` (`W_diss`). The table thus defines the two uncoupled initial value problems

$$\dot{W}_{prod/diss}(t) = Q_{prod/diss}(t)$$
$$W_{prod/diss}(0) = 0$$

To obtain the first-order ODEs, take the time derivative of the integrals.

$$W_{\text{prod/diss}}(t) = \int_0^t Q_{\text{prod/diss}}(t')dt'$$

The initial values follow from setting $t = 0$.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Click the **Transparency** button on the Graphics toolbar.

3 Select Boundaries 4, 7, and 11 only.

4 Click the **Transparency** button on the Graphics toolbar again to return to the original state.

Size

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click the **Build All** button.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Swept** settings window, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 1 only.

Distribution 1

1 Right-click **Model 1>Mesh 1>Swept 1** and choose **Distribution**.

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type 2.

4 Click the **Build Selected** button.

Swept 2

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 2** and choose **Distribution**.

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type 2.

4 Click the **Build Selected** button.

The complete mesh consists of roughly 5700 elements.
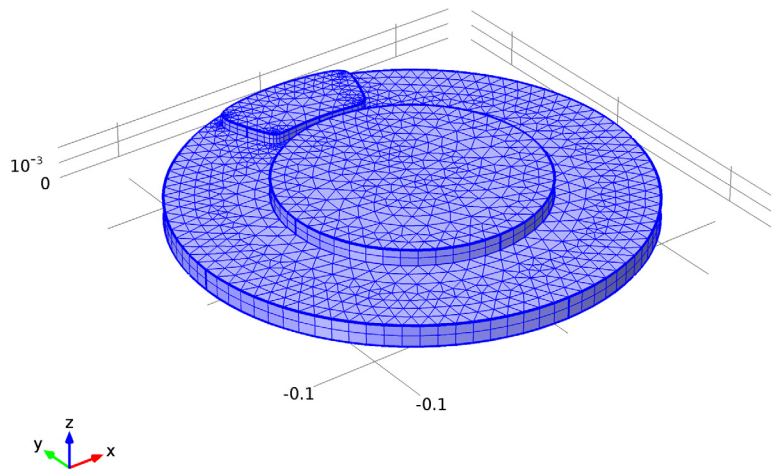
STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

2 In the **Time Dependent** settings window, locate the **Study Settings** section.

3 In the **Times** edit field, type range(0,0.05,1) range(1.2,0.2,3) range(4,10).

4 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

5 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.

2 In the **Time-Dependent Solver** settings window, click to expand the **Absolute Tolerance** section.

3 In the **Tolerance** edit field, type 1e-4.

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

This setting forces the solver to take at least one step in each specified interval.

5 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The first of the two default plots displays the surface temperature of the brake disc and pad at the end of the simulation interval. Modify this plot to show the time just before releasing the brake.

Temperature

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

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

3 From the **Time** list, choose **1.8**.

4 Click the **Plot** button.

5 Click the **Zoom Extents** button on the Graphics toolbar.

Compare the result to the plot shown in Figure 2.

To compare the total heat produced and the heat dissipated, as done in Figure 5, follow the steps given below.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Click to collapse the **Title** section. Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated edit field, type Time (s).

6 Right-click **Results>1D Plot Group 3** and choose **Point Graph**.

7 Select Point 1 only.

8 In the **Point Graph** settings window, locate the **y-Axis Data** section.

9 In the **Expression** edit field, type log10(W_prod+1).

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Blue**.

11 Click to expand the **Legends** section. Select the **Show legends** check box.

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

13 In the table, enter the following settings:

Legends
log10(W_prod+1), heat produced

14 Right-click **Results>1D Plot Group 3>Point Graph 1** and choose **Duplicate**.

15 In the **Point Graph** settings window, locate the **y-Axis Data** section.

16 In the **Expression** edit field, type log10(W_diss+1).

17 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

| Legends |
| --- |
| log10(W_diss+1), heat dissipated |

19 Click the **Plot** button.

20 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Rename**.

21 Go to the **Rename 1D Plot Group** dialog box and type Dissipated and produced heat in the **New name** edit field.

22 Click **OK**.

Finally, follow the steps below to reproduce the plot in Figure 3.

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Line 3D**.

2 In the **Cut Line 3D** settings window, locate the **Line Data** section.

3 In row **Point 1**, set **z** to 0.013.

4 In row **Point 2**, set **x** to -0.047.

5 In row **Point 2**, set **y** to 0.1316.

6 In row **Point 2**, set **z** to 0.013.

7 Click the **Plot** button.
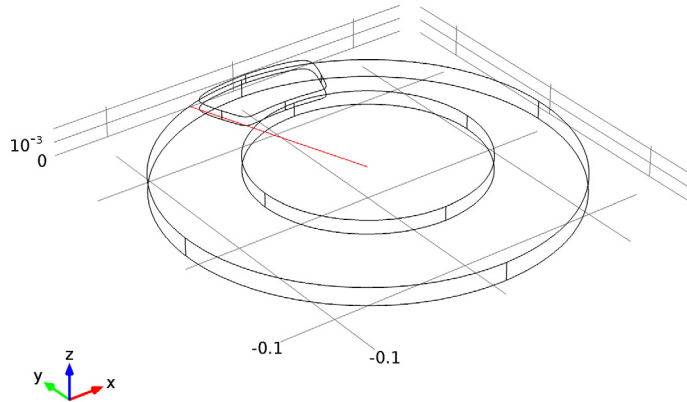
8 In the **Model Builder** window, right-click **Data Sets** and choose **More Data Sets>Parametric Extrusion 1D**.

9 In the **Parametric Extrusion 1D** settings window, locate the **Data** section.

10 Click and Shift-click in the list to select all time steps from 0 through 3 s.

2D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 4** and choose **Surface**.

3 In the **Surface** settings window, locate the **Coloring and Style** section.

4 From the **Color table** list, choose **ThermalLight**.

5 Right-click **Results>2D Plot Group 4>Surface 1** and choose **Height Expression**.

6 Right-click **2D Plot Group 4** and choose **Rename**.

7 Go to the **Rename 2D Plot Group** dialog box and type `Temperature profile vs time` in the **New name** edit field.

8 Click **OK**.