Created in COMSOL Multiphysics 6.2

Optimal Control for Heating of a Rod

Introduction

Optimal control involves is type of optimization specific to transient problems. Typically, a number of scalar parameters is allowed to vary in time within certain bounds, and the one then has to find the optimal transient variation of these parameters with respect to some objective, and possibly constraints. This models demonstrates optimal control for a simple heating example using the **Control Function** feature.

Model Definition

The model geometry consists of a rod that is 3 m long and 1 cm in diameter. The rod is made of structural steel and the idea is to heat it as fast as possible without exceeding some maximum temperature. Once a certain minimum temperature is reached, the process is complete, and the objective is to minimize the processing time. The simple geometry of the problem means that the minimum temperature will occur in the center, while the maximum will occur on the boundary. If infinite power is available, one can thus solve the problem by applying the maximum temperature on the boundary and computing the power as a postprocessing step. For a finite power, one has to apply the maximum power and switch to temperature control, when the maximum temperature is reached on the boundary.

In this example we will use a **Control Function** feature for the power and minimize the difference between the temperature and a target temperature on the boundary. The target temperature should thus be slightly smaller than the maximum temperature. In order to arrive at a scalar objective, φ, the squared temperature difference is integrated over time:

$$\varphi = \int_T (T - T_{\mathrm{target}})^2 dt$$

The model uses 260°C as the target temperature and 20°C as the initial temperature. The maximum heating power is 50 kW, and the simulation time is 10 s.

Results and Discussion

Figure 1 displays the temperature on the inside and the outside of the rod for the initial uniform heating. The temperatures increases linearly with time resulting in excessive heating.

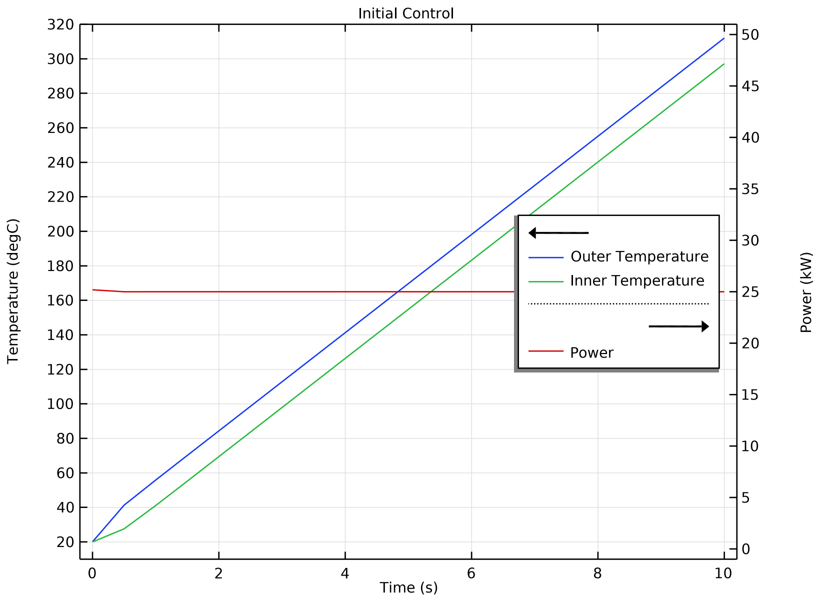

Figure 1: The plot shows the initial temperature at the surface of the cylinder for a heating power corresponding to 50% of the maximum power. The temperature for the optimization results is also shown. The control function values are plotted on the 2nd y-axis.

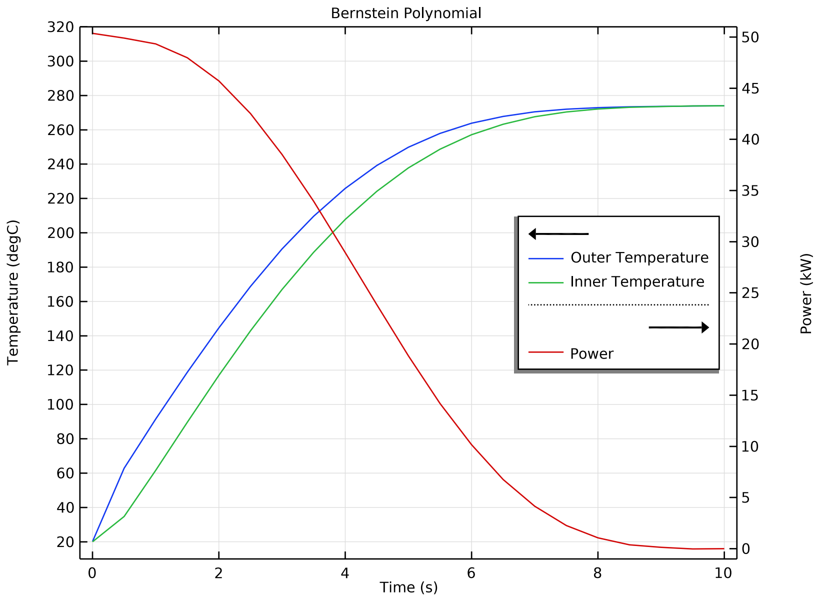

Figure 2: The plot shows the temperature and heating as function of time for the case of control function regularized via Bernstein polynomial. The heating is plotted on the 2nd y-axis.

The result of the optimization for a 7th order Bernstein polynomial is shown in Figure 2, while Figure 3 shows the case of Helmholtz filter with a maximum slope 10 times the slope corresponding to a steady increase from the minimum to the maximum. The resulting controls and temperature are similar, and excessive heating is avoided in both cases.

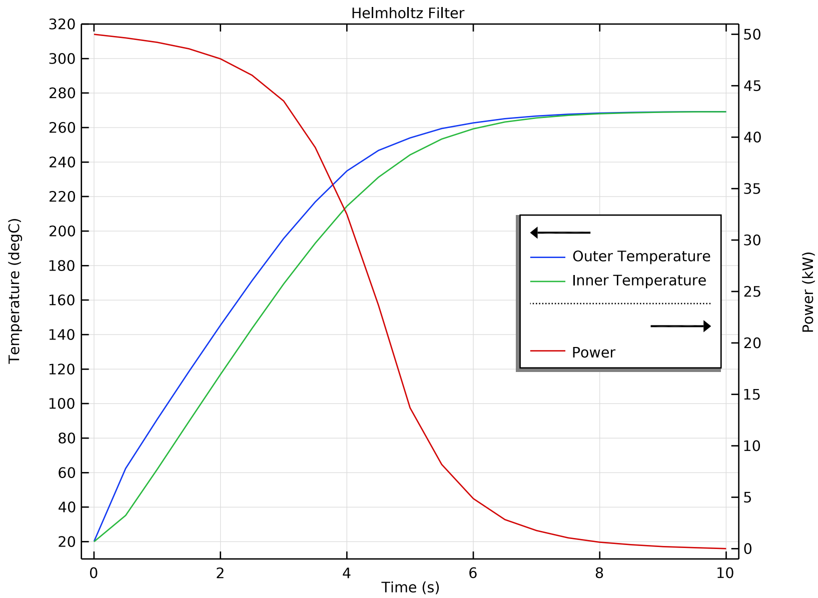

Figure 3: The plot shows the temperature and heating as function of time for the case of control function regularized via Helmholtz filtering. The heating is plotted on the 2nd y-axis.

Application Library path: `Optimization_Module/Optimal_Control/optimal_heating_control`

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 **1D Axisymmetric**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

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
radius	5[mm]	0.005 m	Cylinder radius
tmax	10[s]	10 s	Simulation time
Pmax	50[kW]	50000 W	Maximum power
Ttarget	260[degC]	533.15 K	Target temperature
T0	20[degC]	293.15 K	Initial temperature
Lz	3[m]	3 m	Out-of-plane length scale

GEOMETRY 1

Interval 1 (i1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.

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

3 In the table, enter the following settings:

Coordinates (m)
0
radius

DEFINITIONS

Outer Temperature

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `Outer Temperature` in the **Label** text field.

3 In the **Variable name** text field, type `Tout`.

4 Select Boundary 2 only.

Inner Temperature

1 Right-click **Outer Temperature** and choose **Duplicate**.

2 In the **Settings** window for **Point Probe**, type `Inner Temperature` in the **Label** text field.

3 In the **Variable name** text field, type `Tin`.

4 Select Boundary 1 only.

Temperature Error

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Temperature Error` in the **Label** text field.

3 In the **Variable name** text field, type `Terror`.

4 Locate the **Expression** section. In the **Expression** text field, type `(1-(Tout-T0)/(Ttarget-T0))^2`.

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>Structural steel**.

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

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

HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (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`.

DEFINITIONS (COMP1)

Control Function (Polynomial)

1. In the **Definitions** toolbar, click **Control Variables** and choose **Control Function**.
2. In the **Settings** window for **Control Function**, type `Control Function (Polynomial)` in the **Label** text field.
3. Locate the **Input** section. In the x_{end} text field, type `tmax`.
4. In the n text field, type `7`.

Control Function (Helmholtz)

1. Right-click **Control Function (Polynomial)** and choose **Duplicate**.
2. In the **Settings** window for **Control Function**, type `Control Function (Helmholtz)` in the **Label** text field.
3. Locate the **Input** section. From the **Control type** list, choose **Helmholtz filter**.
4. In the f_{max} text field, type `10/tmax`.

HEAT TRANSFER IN SOLIDS (HT)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.
2. In the **Settings** window for **Heat Transfer in Solids**, locate the **Physical Model** section.
3. In the d_z text field, type `Lz`.

Heat Flux 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
4. From the **Flux type** list, choose **Heat rate**.
5. In the P_0 text field, type `cfunc1(t)*Pmax`.

Heat Flux 2

1. Right-click **Heat Flux 1** and choose **Duplicate**.
2. In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
3. In the P_0 text field, type `cfunc2(t)*Pmax`.

GLOBAL ODES AND DAES (GE)

Global Equations 1

Use the ODE to integrate the squared error over time.

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt, t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`obj`	`objt*tmax-Terror`	`0`	`0`	

MESH 1

Edge 1

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

Size

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

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

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

4 Click **Build All**.

INITIAL CONTROL

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

2 In the **Settings** window for **Study**, type `Initial Control` in the **Label** text field.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Initial Control** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,tmax/20,tmax)`.

4 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Heat Flux 2**.

7 Click **Disable**.

8 In the **Home** toolbar, click **Compute**.

RESULTS

Line Graph

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

2 Right-click **Line Graph** and choose **Delete**.

Global 1

1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Tout - Outer Temperature - K**.

3 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Tin - Inner Temperature - K**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Tout	degC	Outer Temperature
Tin	degC	Inner Temperature

5 Click to expand the **Legends** section.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click  **Clear Table**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Heat and energy balance> Energy balance>Net powers, boundary features>ht.hf1.ntefluxInt - Total net energy rate - W**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
-ht.hf1.ntefluxInt	kW	Power

Initial Control

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

2 In the **Settings** window for **1D Plot Group**, type `Initial Control` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Temperature (degC)`.

6 Select the **Two y-axes** check box.

7 In the table, select the **Plot on secondary y-axis** check box for **Global 2**.

8 Select the **Secondary y-axis label** check box.

9 Locate the **Axis** section. Select the **Manual axis limits** check box.

10 In the **x minimum** text field, type `-tmax/50`.

11 In the **x maximum** text field, type `tmax*1.02`.

12 In the **y minimum** text field, type `10`.

13 In the **y maximum** text field, type `320`.

14 In the **Secondary y minimum** text field, type `-Pmax*2e-5`.

15 In the **Secondary y maximum** text field, type `Pmax*1.02e-3`.

16 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

17 In the **Initial Control** toolbar, click **Plot**.

1D Plot Group 2

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

ADD STUDY

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

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

3 Click **Add Study** in the window toolbar twice.

INITIAL CONTROL

Step 1: Time Dependent

In the **Model Builder** window, under **Initial Control** right-click **Step 1: Time Dependent** and choose **Copy**.

OPTIMIZATION (BERNSTEIN POLYNOMIAL)

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

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Optimization (Bernstein Polynomial)`.

5 Right-click **Optimization (Bernstein Polynomial)** and choose **Paste Time Dependent**.

Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **IPOPT**.

4 Find the **Solver settings** subsection. In the **Maximum number of model evaluations** text field, type `50`.

5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Global ODEs and DAEs>comp1.obj - State variable obj - 1**.

6 Locate the **Control Variables and Parameters** section. In the table, clear the **Solve for** check box for **Control Function (Helmholtz) (cfunc2)**.

7 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

8 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Bernstein Polynomial

1 In the **Model Builder** window, right-click **Initial Control** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Bernstein Polynomial` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Optimization (Bernstein Polynomial)/Solution 2 (sol2)**.

OPTIMIZATION (BERNSTEIN POLYNOMIAL)

Optimization

1 In the **Model Builder** window, under **Optimization (Bernstein Polynomial)** click **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Bernstein Polynomial**.

5 In the **Study** toolbar, click **Compute**.

Optimization, Step 1: Time Dependent

1 In the **Model Builder** window, under **Optimization (Bernstein Polynomial)**, Ctrl-click to select **Optimization** and **Step 1: Time Dependent**.

2 Right-click and choose **Copy**.

STUDY 3

Optimization

In the **Model Builder** window, right-click **Study 3** and choose **Paste Multiple Items**.

Optimization, Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3**, Ctrl-click to select **Optimization** and **Step 1: Time Dependent**.

2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **MMA**.

4 In the **Optimality tolerance** text field, type `0.01`.

5 From the **Study step** list, choose **Time Dependent**.

6 Locate the **Control Variables and Parameters** section. In the table, enter the following settings:

Control variables from physics interface	Solve for
Control Function (Polynomial) (cfunc1)	
Control Function (Helmholtz) (cfunc2)	√

7 Locate the **Output While Solving** section. From the **Output table** list, choose **New**.

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Heat Flux 1**.

4 Click **Disable**.

5 In the tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Heat Flux 2**.

6 Click **Enable**.

7 In the **Model Builder** window, click **Study 3**.

8 In the **Settings** window for **Study**, locate the **Study Settings** section.

9 Clear the **Generate default plots** check box.

10 In the **Label** text field, type `Optimization (Helmholtz Filter)`.

11 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Bernstein Polynomial

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

2 In the **Bernstein Polynomial** toolbar, click **Plot**.

Bernstein Polynomial 1

Right-click **Bernstein Polynomial** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, expand the **Bernstein Polynomial 1** node, then click **Global 2**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click **Clear Table**.

Helmholtz Filter

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

2 In the **Settings** window for **1D Plot Group**, type `Helmholtz Filter` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Optimization (Helmholtz Filter)/ Solution 3 (sol3)**.

Global 2

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

2 In the **Settings** window for **Global**, click **Section toolbar** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Heat and energy balance>Energy balance>Net powers, boundary features>ht.hf2.ntefluxInt - Total net energy rate - W**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`-ht.hf2.ntefluxInt`	`kW`	`Power`

OPTIMIZATION (HELMHOLTZ FILTER)

Optimization

1 In the **Model Builder** window, under **Optimization (Helmholtz Filter)** click **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Output While Solving** section.

3 From the **Plot group** list, choose **Helmholtz Filter**.

4 In the **Home** toolbar, click **Compute**.

RESULTS

Helmholtz Filter

1 In the **Helmholtz Filter** toolbar, click **Plot**.

The optimal power seems to take a similar shape regardless of which **Control Function** is used.