

FEBioHeat Plugin

Table of Contents

1	Introduction	2
2	Heat Module	2
3	Heat transfer constitutive model	2
4	Heat Transfer Boundary Conditions	2
4.1	Fixed and prescribed temperature	2
4.2	Heat flux	3
4.3	Convective heat flux	3
5	Heat source	3
6	Output variables	3
6.1	Plot file variables	3
6.2	Log file variables	4

1 Introduction

The FEBioHeat plugin adds the ability to FEBio to solve heat transfer problems. It currently supports steady-state and transient linear heat transfer analysis. This document describes the format of the FEBio input file for a heat-transfer analysis.

This manual assumes the 2.5 FEBio format specification.

2 Heat Module

The module type must be set to “heat” for solving a heat-transfer problem with this plugin.

```
<Module type="heat"/>
```

3 Heat transfer constitutive model

Currently, only one constitutive model is supported for heat-transfer analysis, namely the “isotropic Fourier”, which defines the heat flux \mathbf{q} as follows.

$$\mathbf{q} = -k\nabla T \quad (1)$$

Here, T is the temperature and k the thermal diffusivity.

This material has three parameters as shown in the following table.

parameter	description	units (SI)
density	The material density	kg/m ³
k	The thermal conductivity	W/m.K
c	The specific heat	J/kg.K

The units are just given as an example of a consistent set of units for these variables. Users can use their preferred unit system instead.

Note that density and specific heat are only used for transient analysis.

An example of material definition follows below.

```
<material id="1" name="myMaterial1" type="isotropic Fourier">
  <density>1.0</density>
  <k>0.4</k>
  <c>1.0</c>
</material>
```

4 Heat Transfer Boundary Conditions

4.1 Fixed and prescribed temperature

A prescribed or fixed temperature boundary condition is defined using “T” as the name of the degree of freedom. Aside from that it is defined similarly to any other prescribed or fixed boundary condition in FEBio and must be defined in the *Boundary* section of the FEBio input file. For example,

```
<prescribe bc="T" node_set="PrescribedBC1">
  <scale lc="1">1.0</scale>
```

```
<relative>0</relative>
</prescribe>
```

4.2 Heat flux

The heat flux surface load prescribes the heat flux on a surface of the mesh. It is defined as a *surface_load* with the type attribute set to “heatflux”. It takes a single parameter, namely “flux”, which specifies the value of the heat flux.

```
<surface_load type="heatflux" surface="Surface01">
  <flux lc="1">2.5</flux>
</surface_load>
```

4.3 Convective heat flux

A convective heat flux applies a flux boundary condition where the flux is proportional to the difference between the surface temperature T and the ambient temperature T_0 .

$$q_c = h_c (T - T_0) \quad (2)$$

This boundary load is defined as a *surface_load* with the type attribute set to “convective_heatflux” and requires two parameters.

parameter	description	units (SI)
hc	The proportionality constant h_c	W/m ² .K
Ta	The ambient temperature	K

An example is given below.

```
<surface_load type="convective_heatflux" surface="Surface01">
  <hc>60.0</hc>
  <Ta lc="1">25</Ta>
</surface_load>
```

5 Heat source

A heat source can be added by defining a *body_load* with the type set to “heat_source”. Only one parameter is required, “Q”, to define the heat source value.

```
<body_load type="heat_source">
  <Q lc="1">13.5</Q>
</body_load>
```

6 Output variables

The heat transfer plugin adds several output variables to both the plot file and the log file.

6.1 Plot file variables

The following table shows a list of all new variables defined by the plugin.

variable	description
temperature	The nodal temperature at the mesh nodes

heat flux	The average heat flux over each element.
-----------	--

6.2 Log file variables

The following table shows a list a log file variables.

variable	description	class
T	nodal temperature	node_data