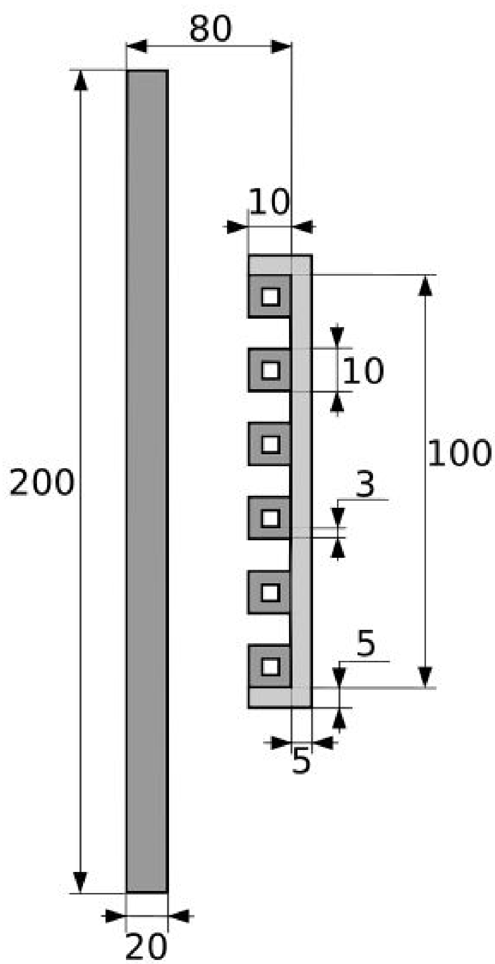
Induction Heating Templates tutorial

When you need to simulate a **2D axial symmetric induction heating** case with billet and coil of standard, regular shape, it is convenient to use **Induction Heating Templates.**

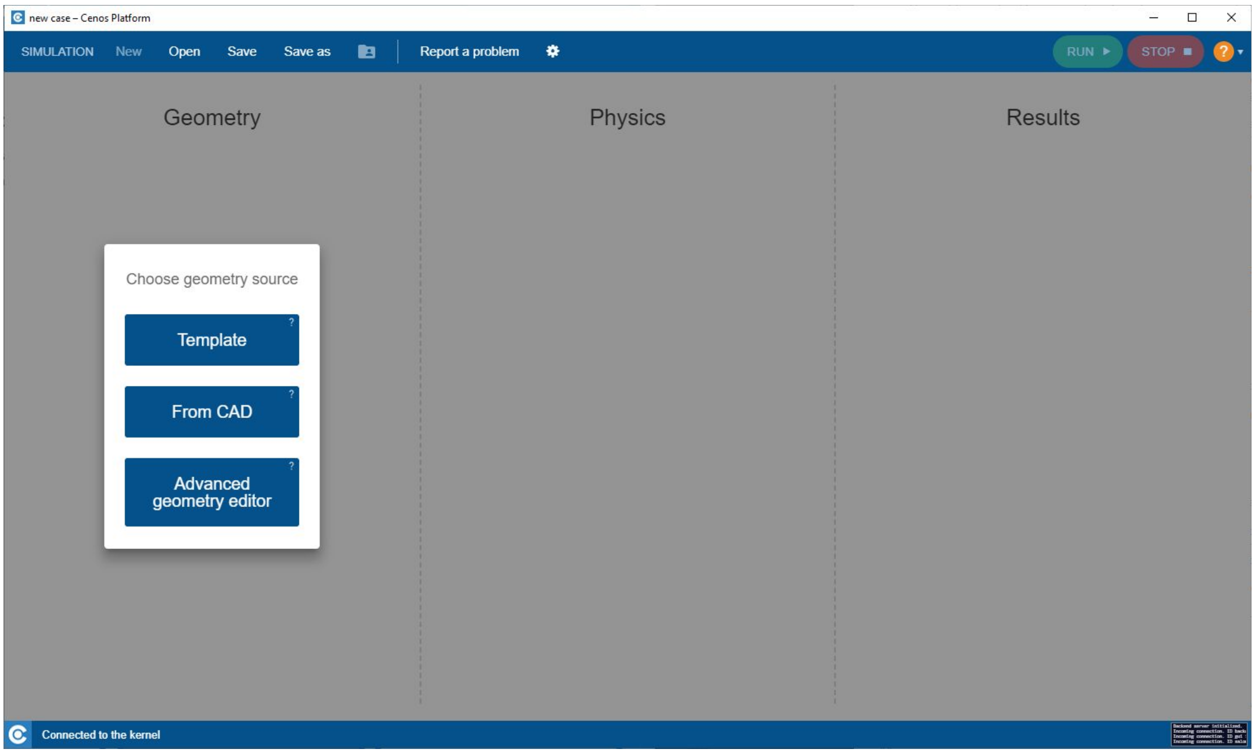
In this tutorial you will learn how to set up and use the Induction Heating Templates for 2D axial symmetric simulation of a cylindrical rod with a rectangular profile coil and a flux concentrator around it. We will create the geometry by choosing one of the available templates for each domain, define the mesh quality, enter the specific values and boundary conditions in the physics setup and in the end evaluate the results using our post-processor.

Induction Heating Template is an easy and fast way to set up and calculate induction heating cases for standard geometries. In the next pages an induction heating example of an **AISI 1020** workpiece at **15 kHz** and **2 kA** with radiation and convection boundary conditions and **FLUXTROL A concentrator** is presented.



1. Open pre-processor

To enter geometry templates, in CENOS home window click **Template.**



Template automatically defines the non-critical parts of the simulation setup and offers new users a **preset simulation physical and geometrical values**, so you only need to change the parameters that are already there.

2. Create Geometry

2.1 Choose units and mesh

You can **define case units** in lower left corner of *Create Geometry* screen. For this tutorial, choose *meters* as case units.

If you want to **change mesh density**, click SHOW MESH OPTIONS.

Then choose from *Mesh density* dropdown.

IMPORTANT: When using templates, mesh is generated automatically before the start of the calculation. With *Mesh density* you define how fine or coarse the mesh will be.

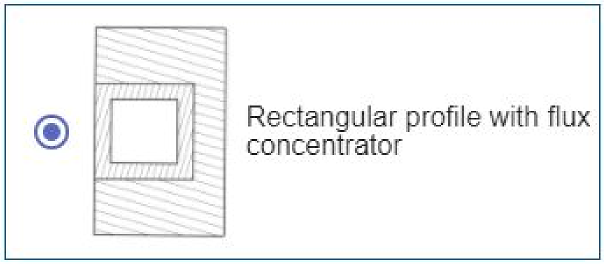
2.2 Set the workpiece geometry

Switch to the WORKPIECE domain. Select **Billet** for *Shape*.

Under *Properties* enter **0.02** for *Diameter* and **0.2** for *Height*.

2.3 Set the inductor geometry

Switch to the INDUCTOR domain. Select **Rectangular profile with flux concentrator** for *Shape*.



Under *Properties* make sure that the *Hollow coil* box is checked and enter the values for the coil and concentrator as follows:

Properties

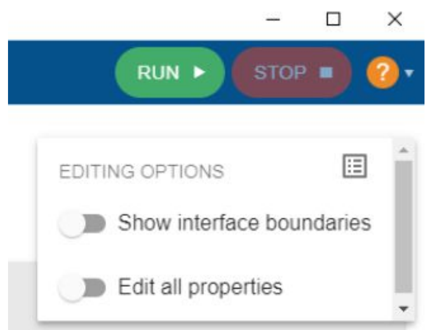
D	0.08	*Diameter (outer)*
h	0.1	*Height*
N	6	*Number of windings*
a	0.01	*Profile width*
b	0.01	*Profile height*
☑	Hollow coil	
δ	0.003	*Profile thickness*
y	0	*Coil center offset*
A	0.005	*Concentrator thickness (horizontal)*
B	0.005	*Concentrator thickness (vertical)*

When the geometry is set, click **Go to PHYSICS** in the upper right corner of the window.

3. Define physics and boundary conditions

IMPORTANT: Part of the window is painted grey meaning that every setting in the grey area is pre-set for this specific template, leaving only the core settings to your mind.

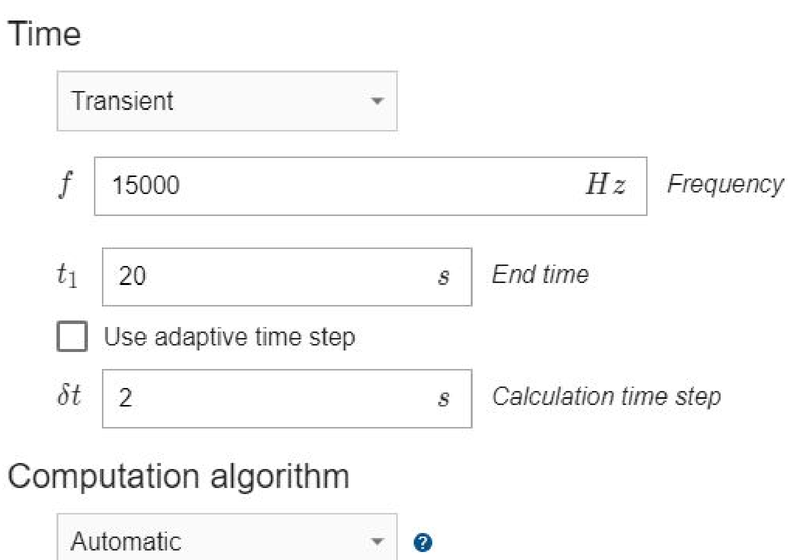
If you want to **edit settings in the grey area**, enable **Edit all properties.**



3.1 Simulation control

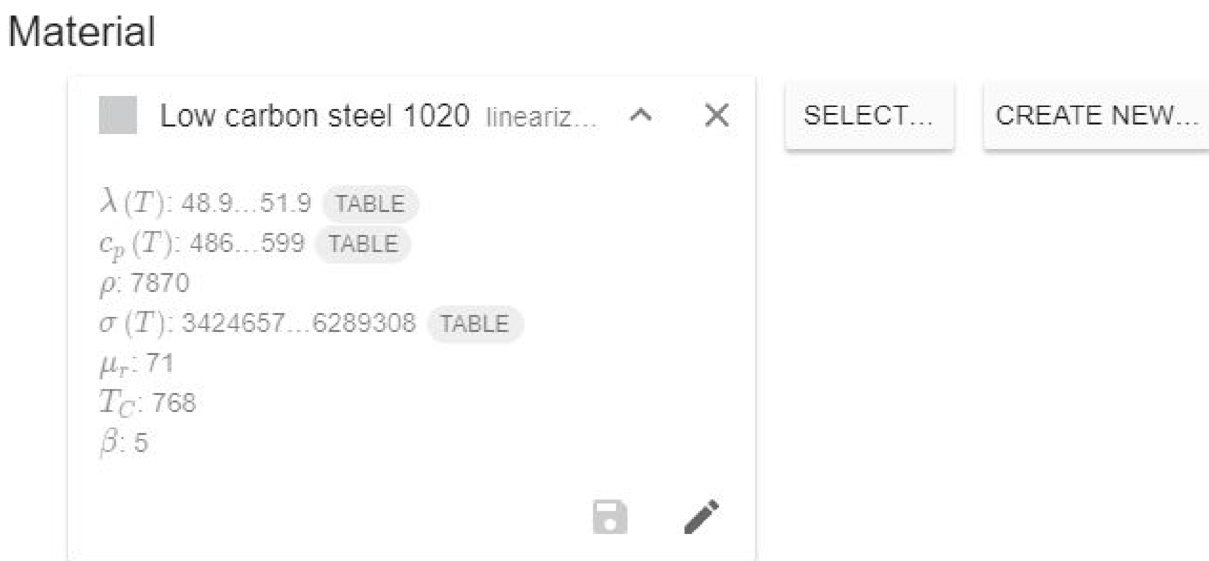
In the SIMULATION CONTROL window you can access and **define the global simulation parameters** such as frequency, calculation time, time step, computational algorithms and many more.

Define the simulation as *Transient* with **15 kHz** *frequency*, **20 s** *End time* and **2 s** *time step.* For *Computation algorithm* choose **Accurate.**

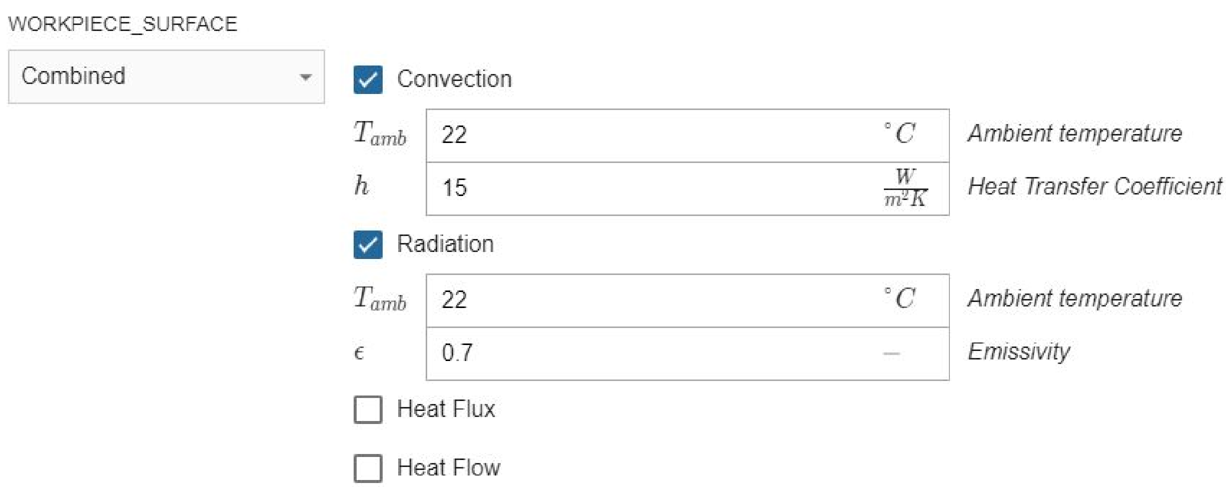


3.2 Workpiece definition

Switch to WORKPIECE in *Domain bar*. For *Material* click SELECT... and choose **Low carbon steel 1020 linearized (H=100000A/m), t depend**.



Under THERMAL ANALYSIS choose **Combined** for WORKPIECE_SURFACE – check the *Convection* and *Radiation* boxes and enter **15** for *Heat Transfer Coefficient* and **0.7** for *Emissivity*.



3.3 Coil definition

Switch to WINDING_0.. in *Domain bar*. Enter **2000 A** for *Current (Amplitude)*.

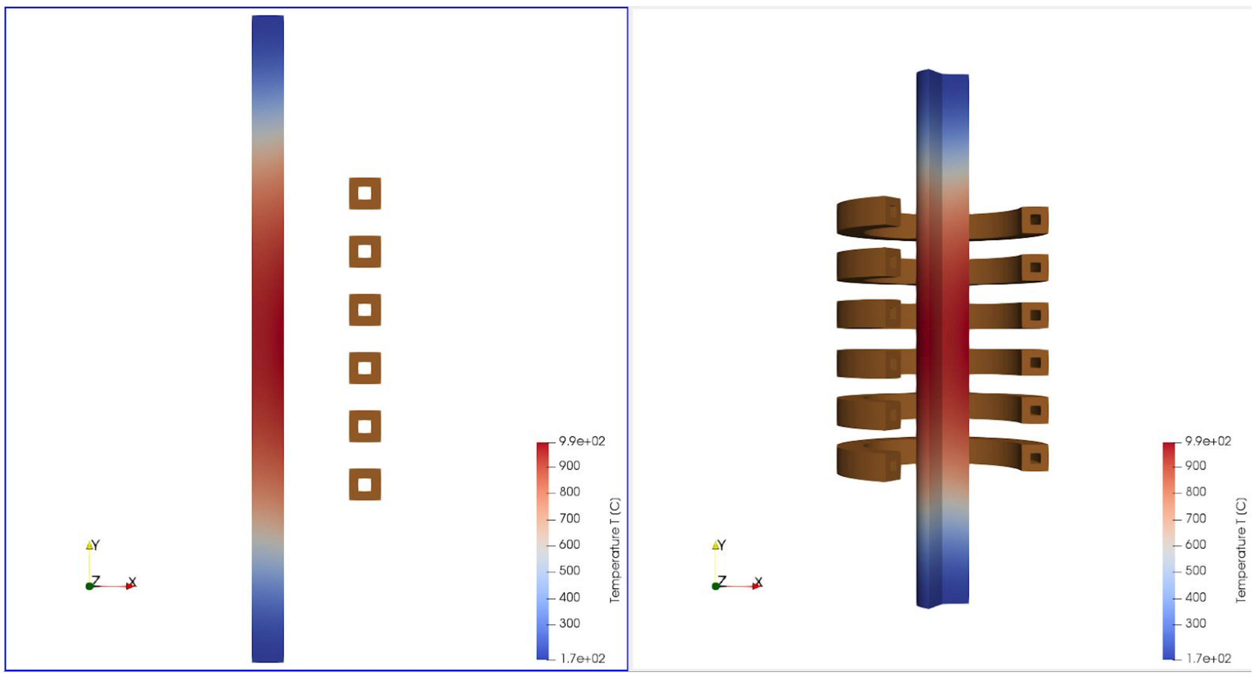
3.4 Concentrator definition

Switch to CONCENTRATOR in *Domain bar*. For *Material* select **Flux concentrator FLUXTROL A**.

Physical definition for the air domain is set fully automatically, so after you define concentrator and conductor domains, **click RUN** in the right upper corner of the window.

4. Evaluate results

In the end of the calculation the post-processing tool *ParaView* will automatically **open with a pre-set temperature state**, and you will be able to see the **temperature field distribution** in the workpiece in the last time step.



This concludes our Induction Heating Template tutorial. For any recommendations or questions contact our support.