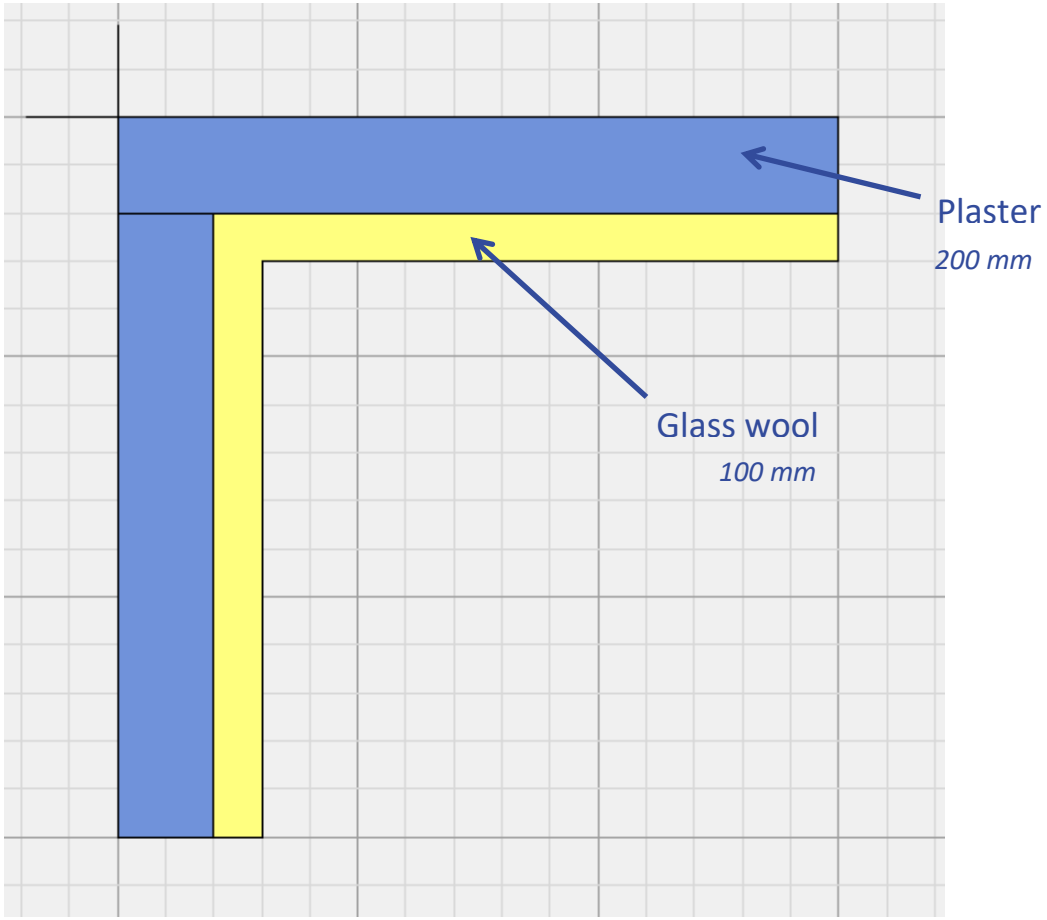
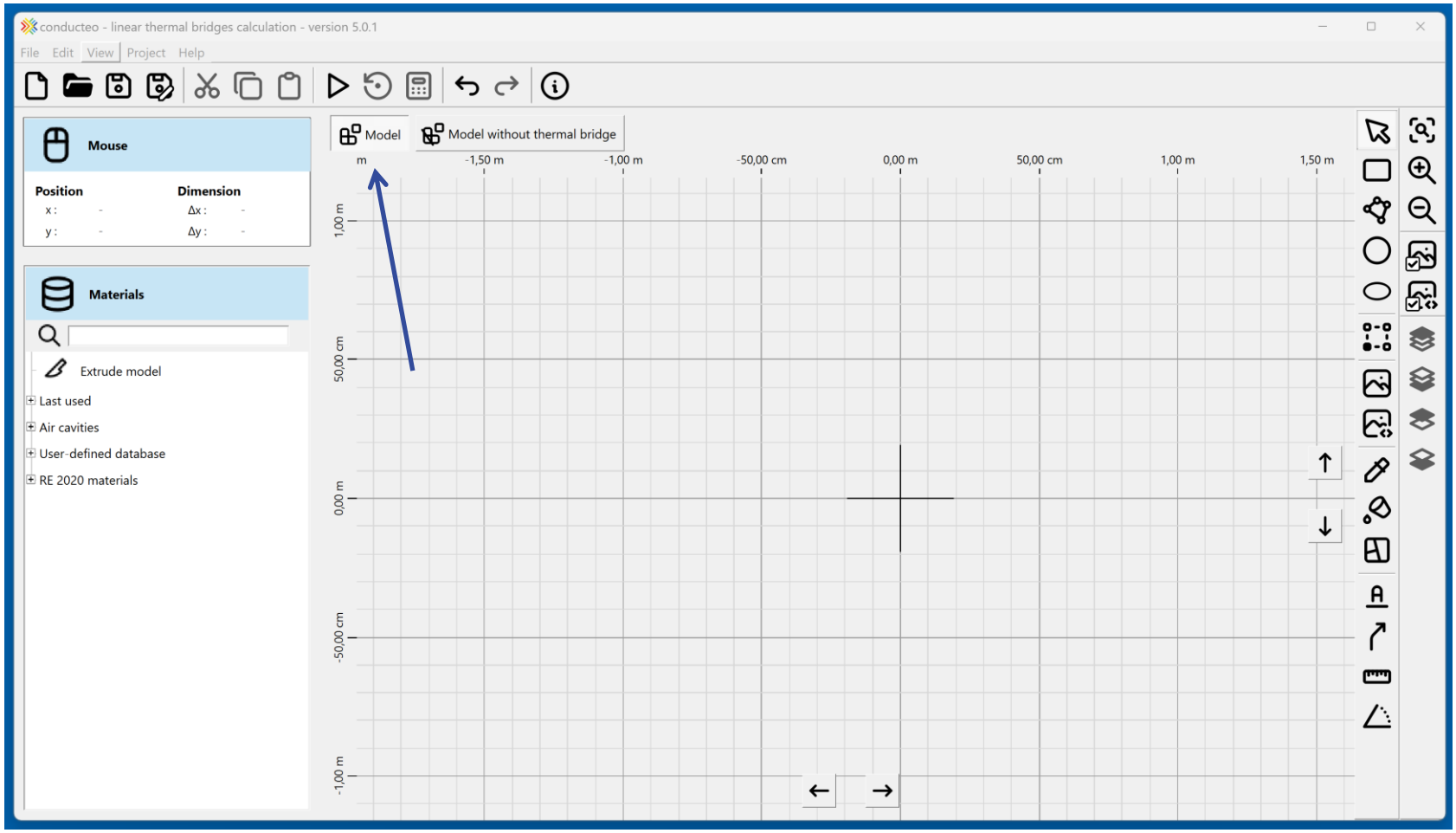
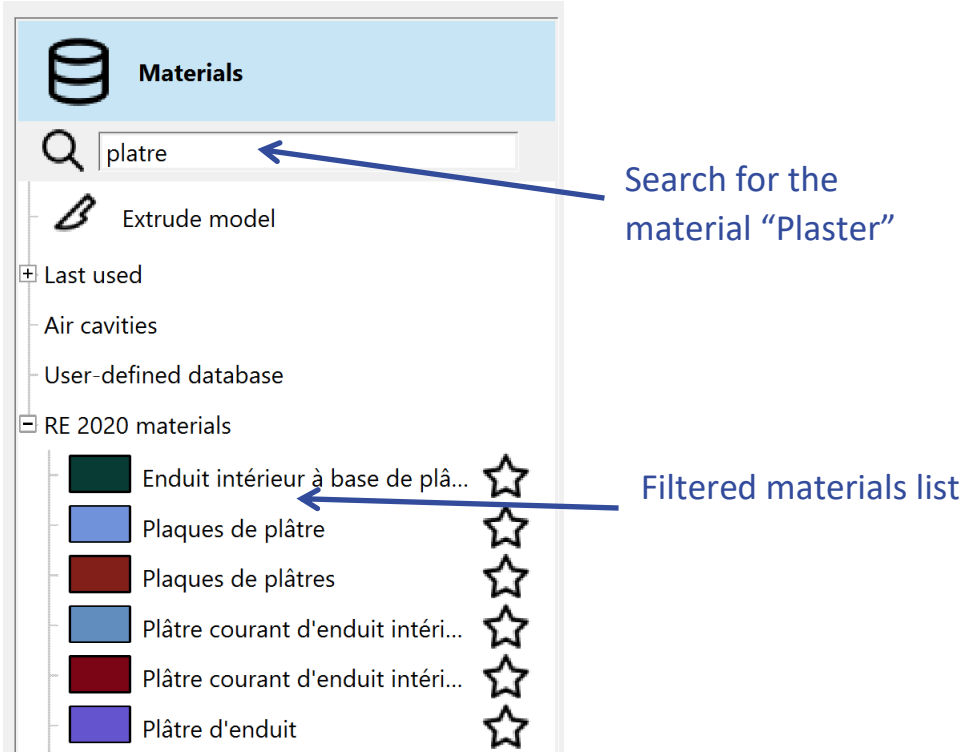
This quick start guide presents, in under 10 minutes, how to model the thermal bridge shown below.



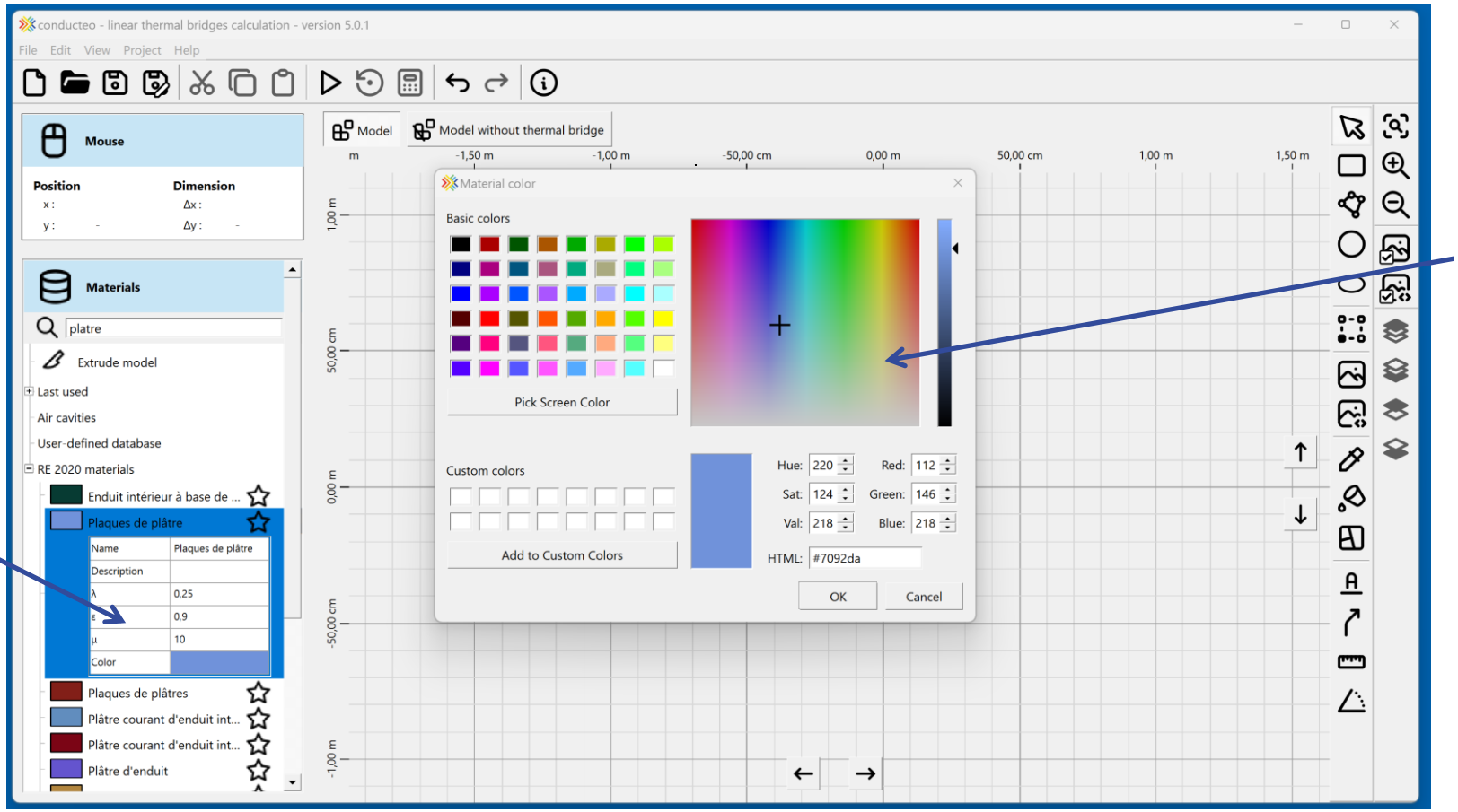
The thermal bridge is modeled in the “Model” tab.



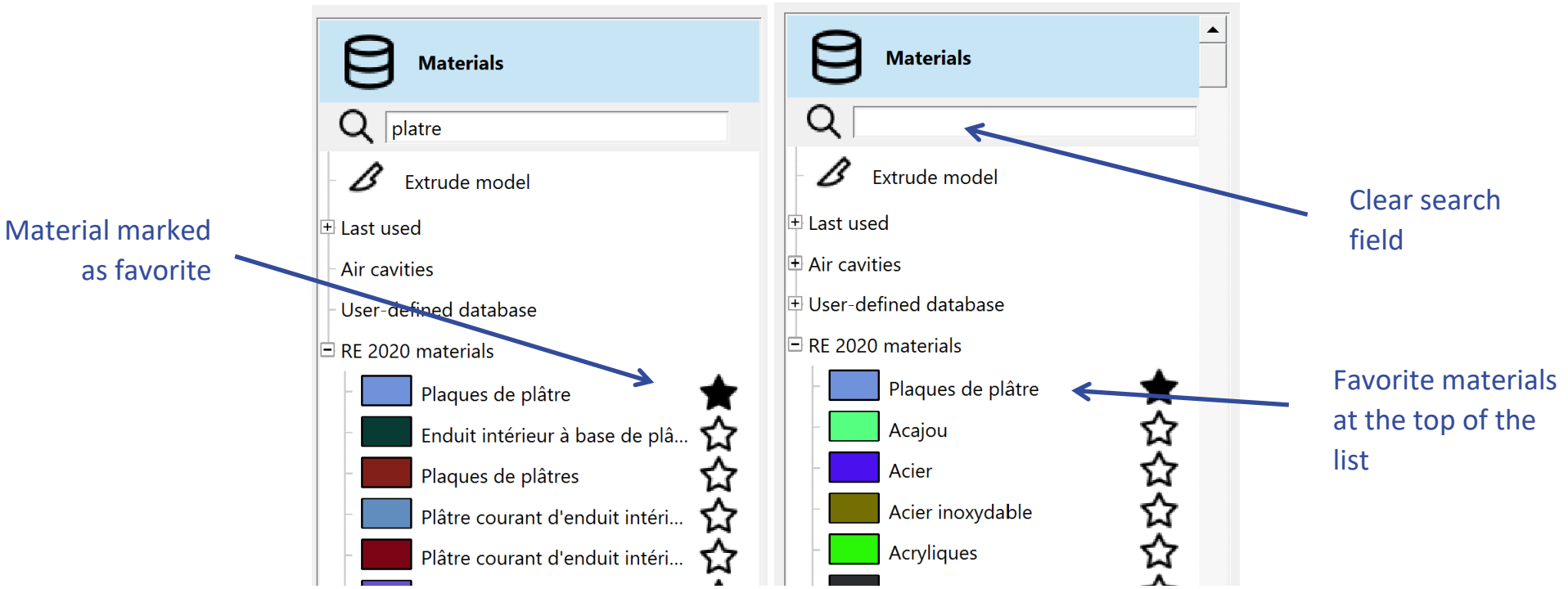
The “Materials” tab provides several libraries, with a search field.



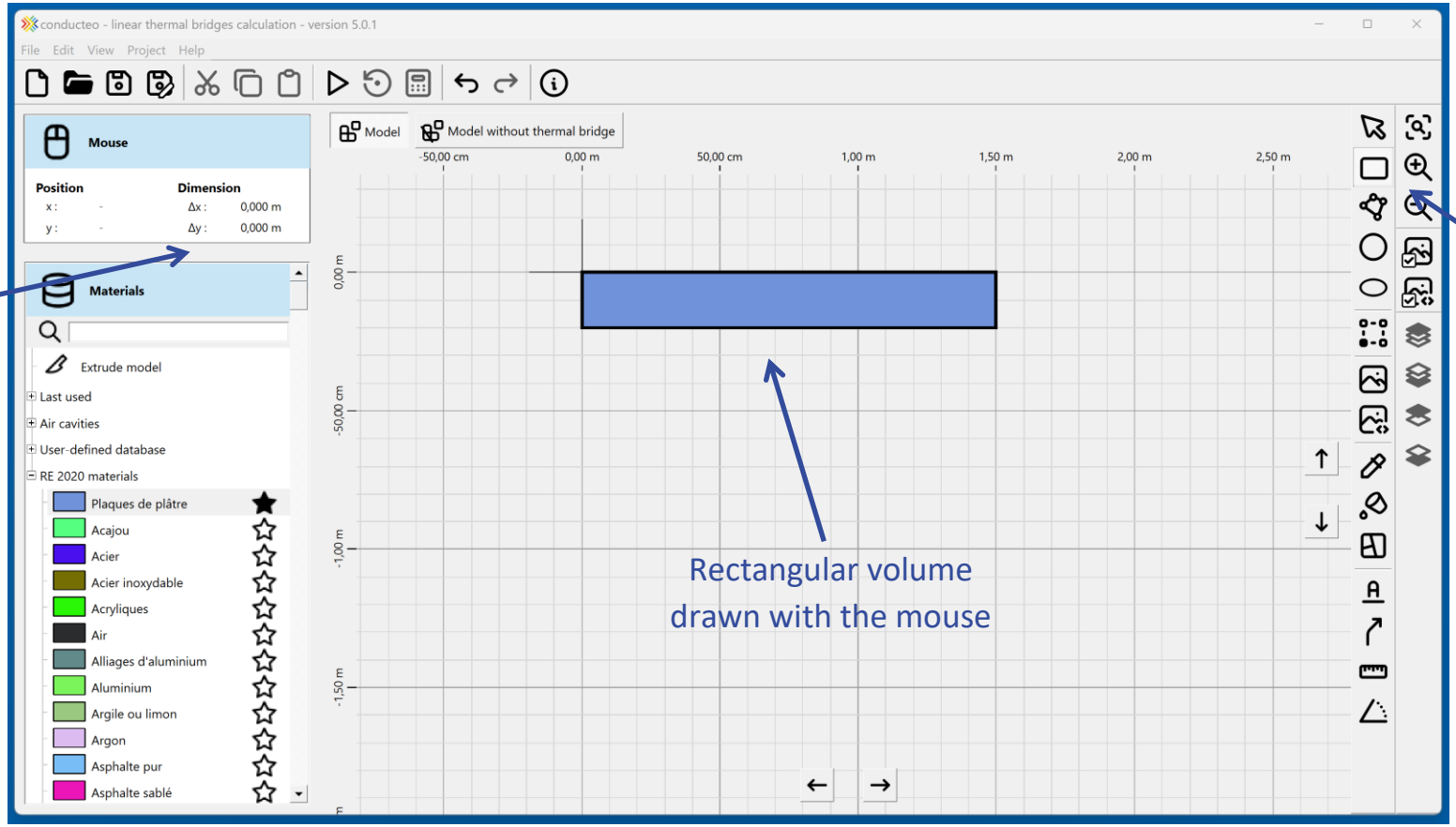
Double-clicking a material allows you to edit its properties.
Double-clicking the color opens a dialog to modify the material color.



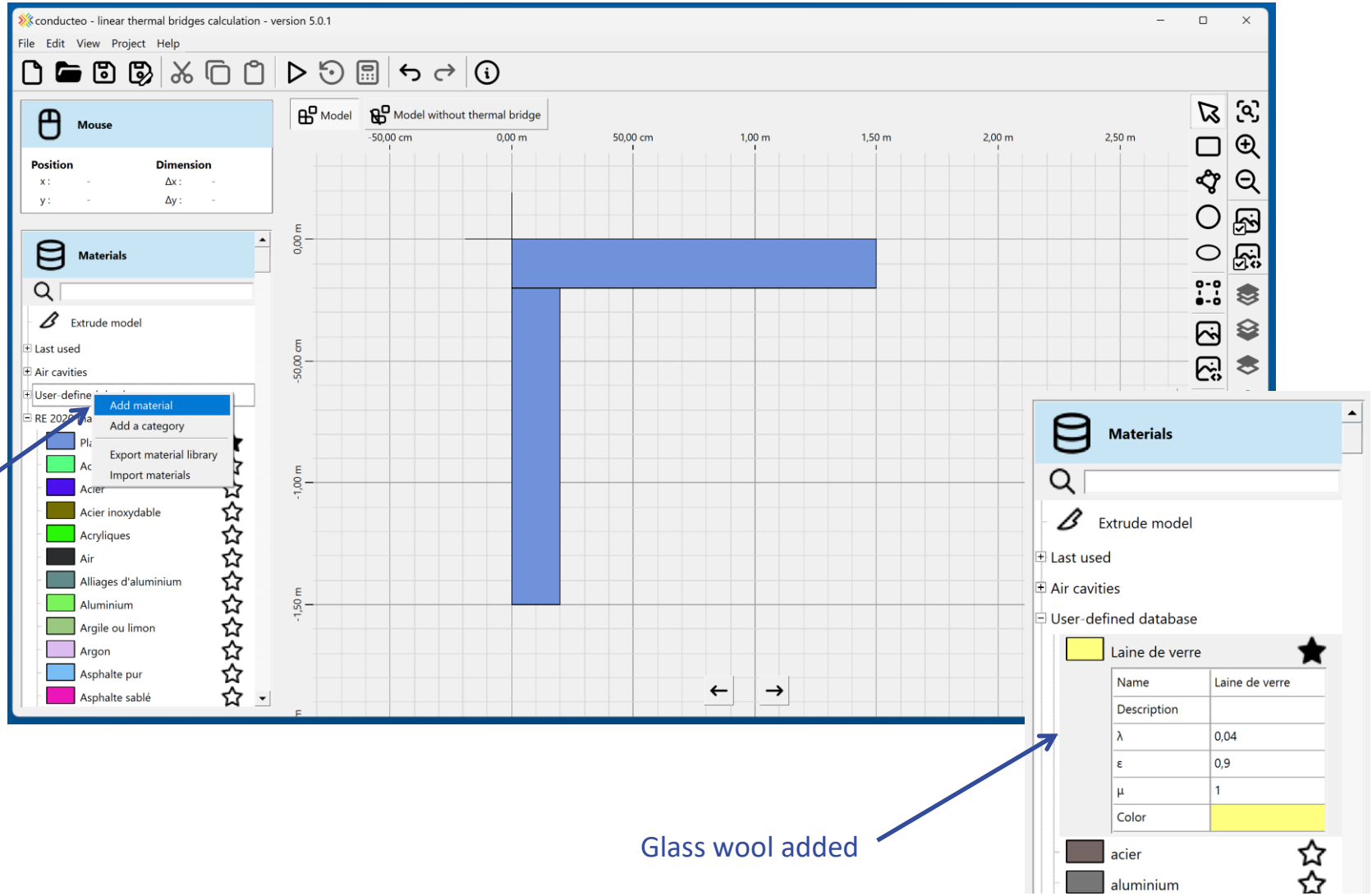
Clicking the star of a material marks it as a favorite.
Favorite materials are displayed at the top of the list.



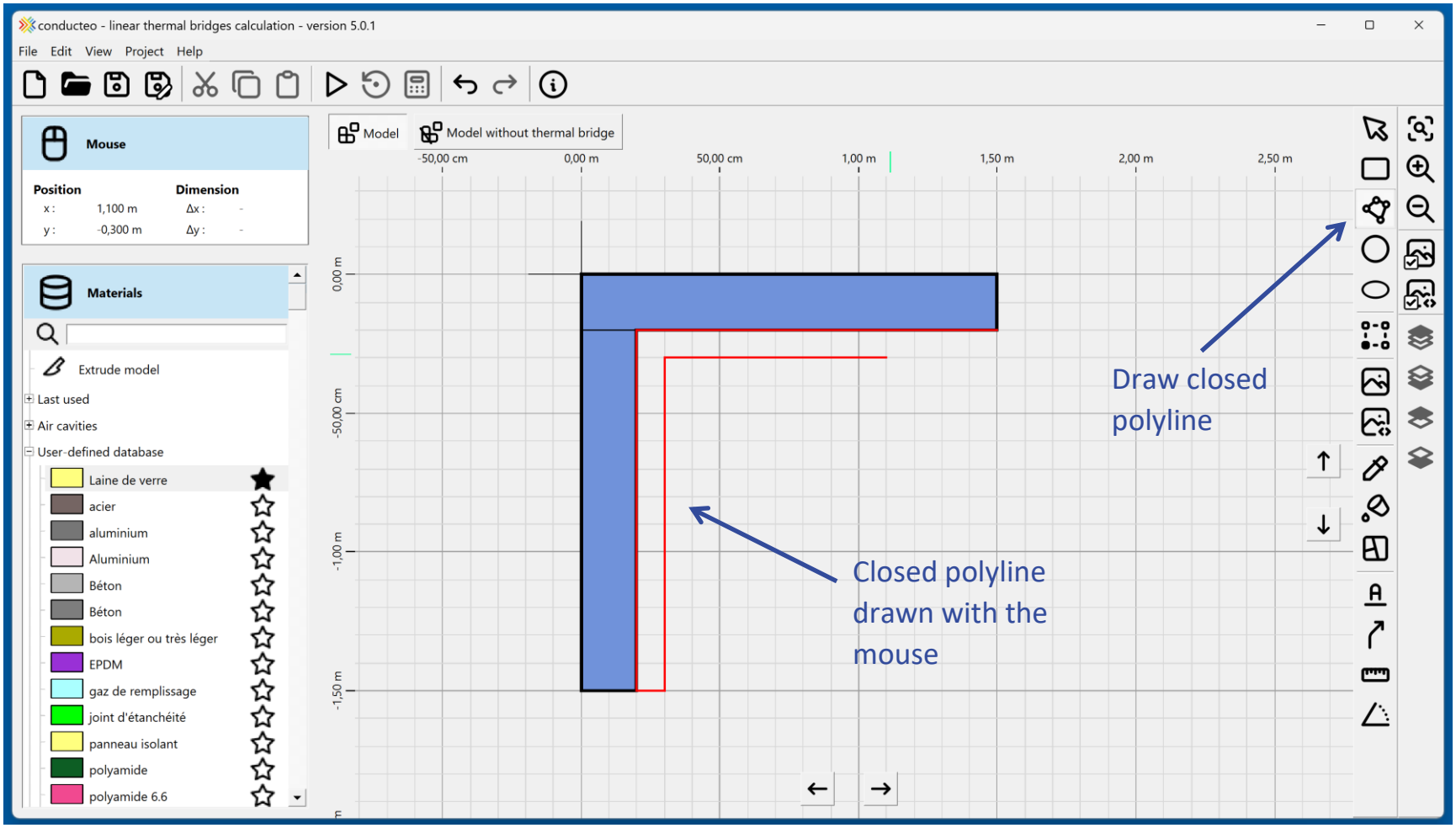
By selecting the “Draw rectangle” option, it is possible to add rectangular volumes with the mouse in the model.



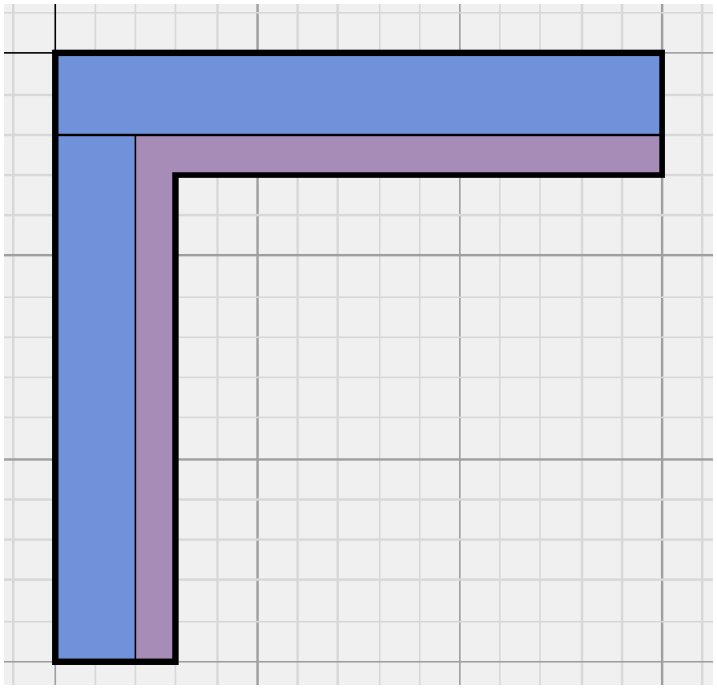
Right-clicking the user library allows you to add materials and categories.

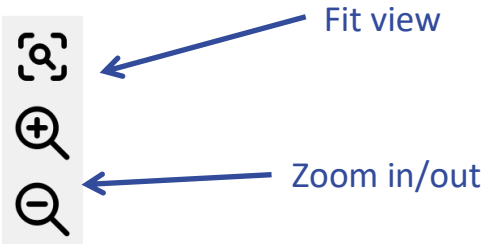


By selecting the “Draw closed polyline” option, it is possible to draw a non-rectangular volume. The volume is completed when returning to the starting point.

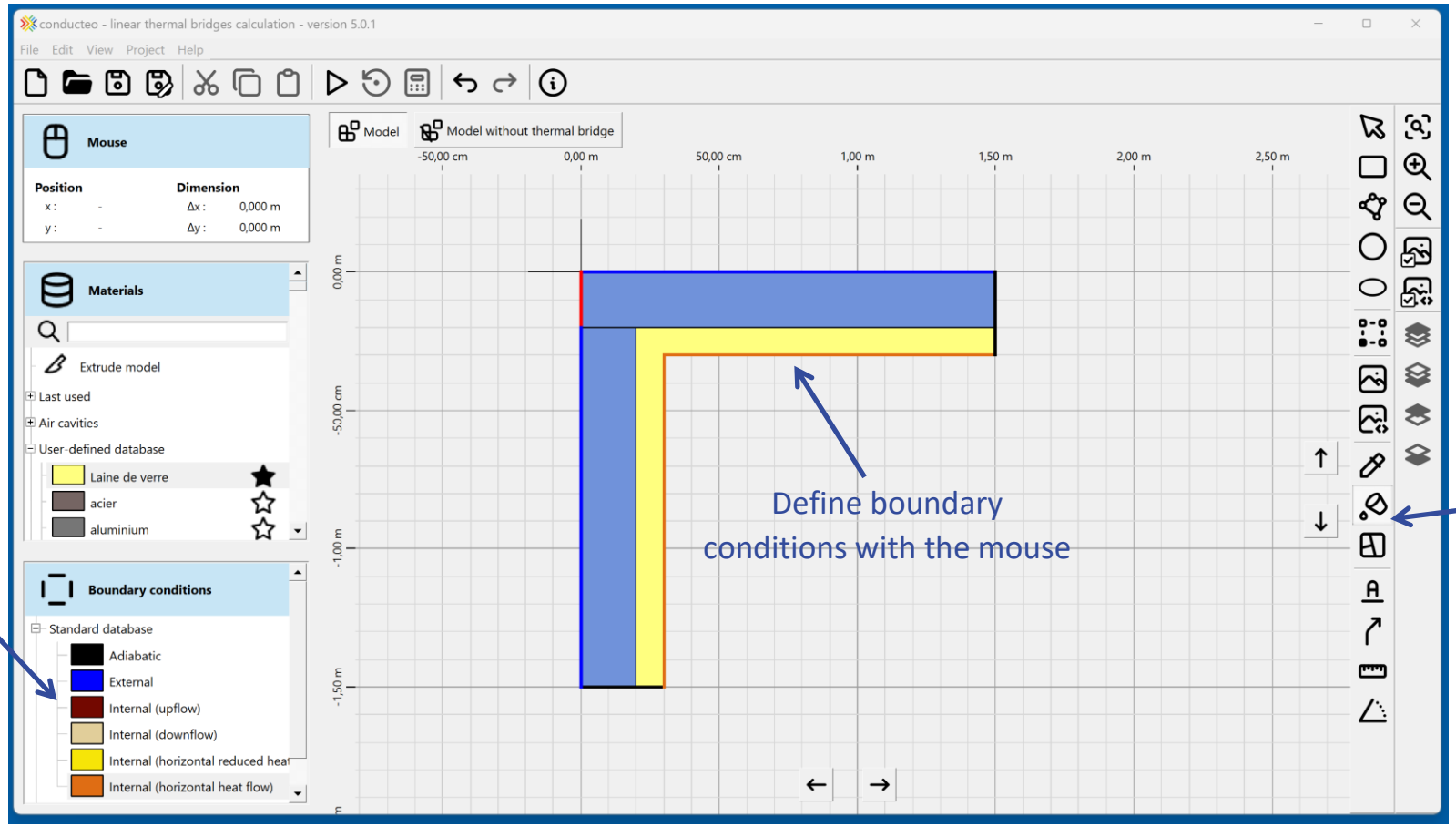


It is possible to adjust the zoom and the view position. The mouse wheel allows you to change the zoom level. The view can be moved by dragging the mouse while holding the wheel button down.

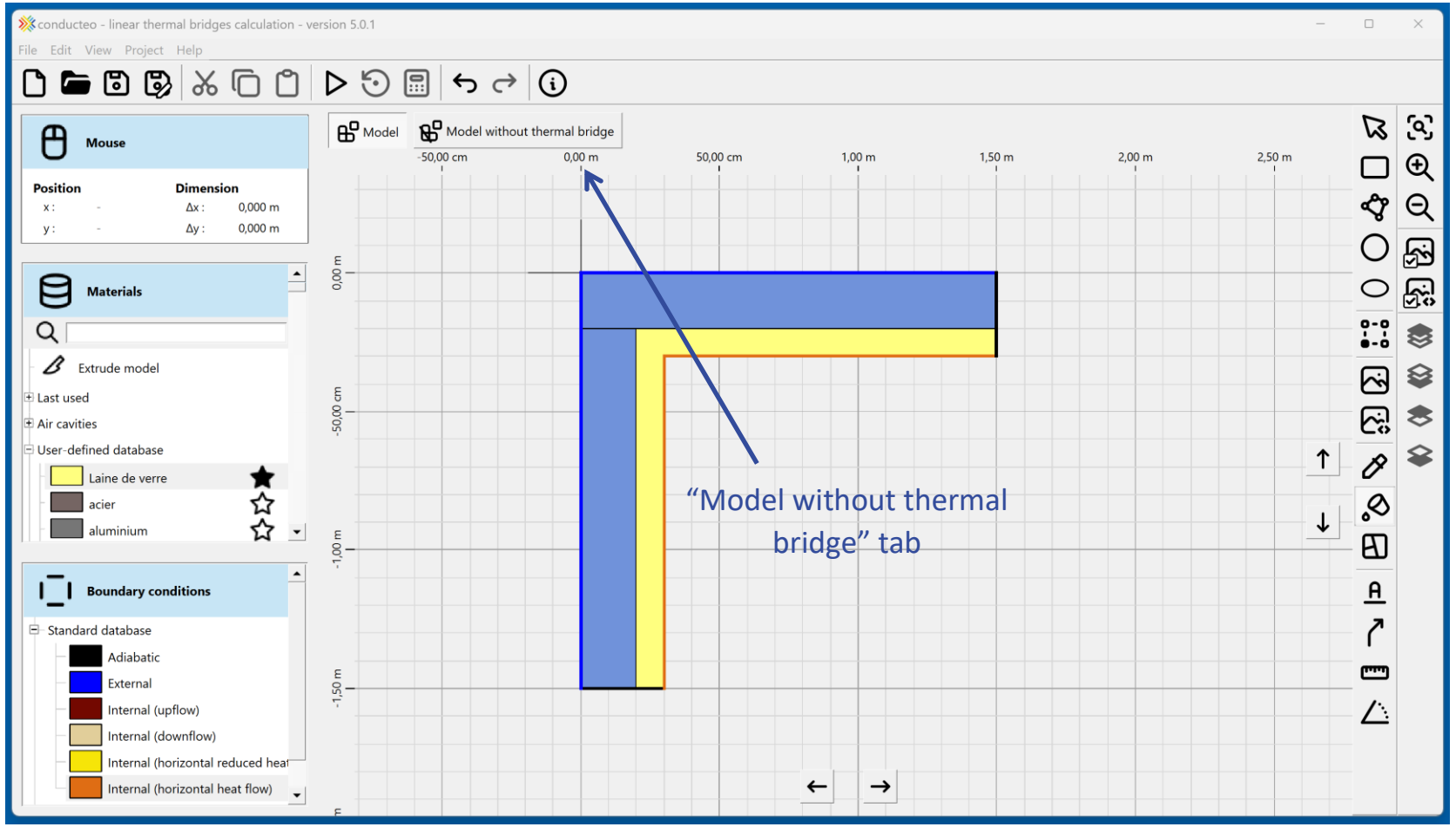




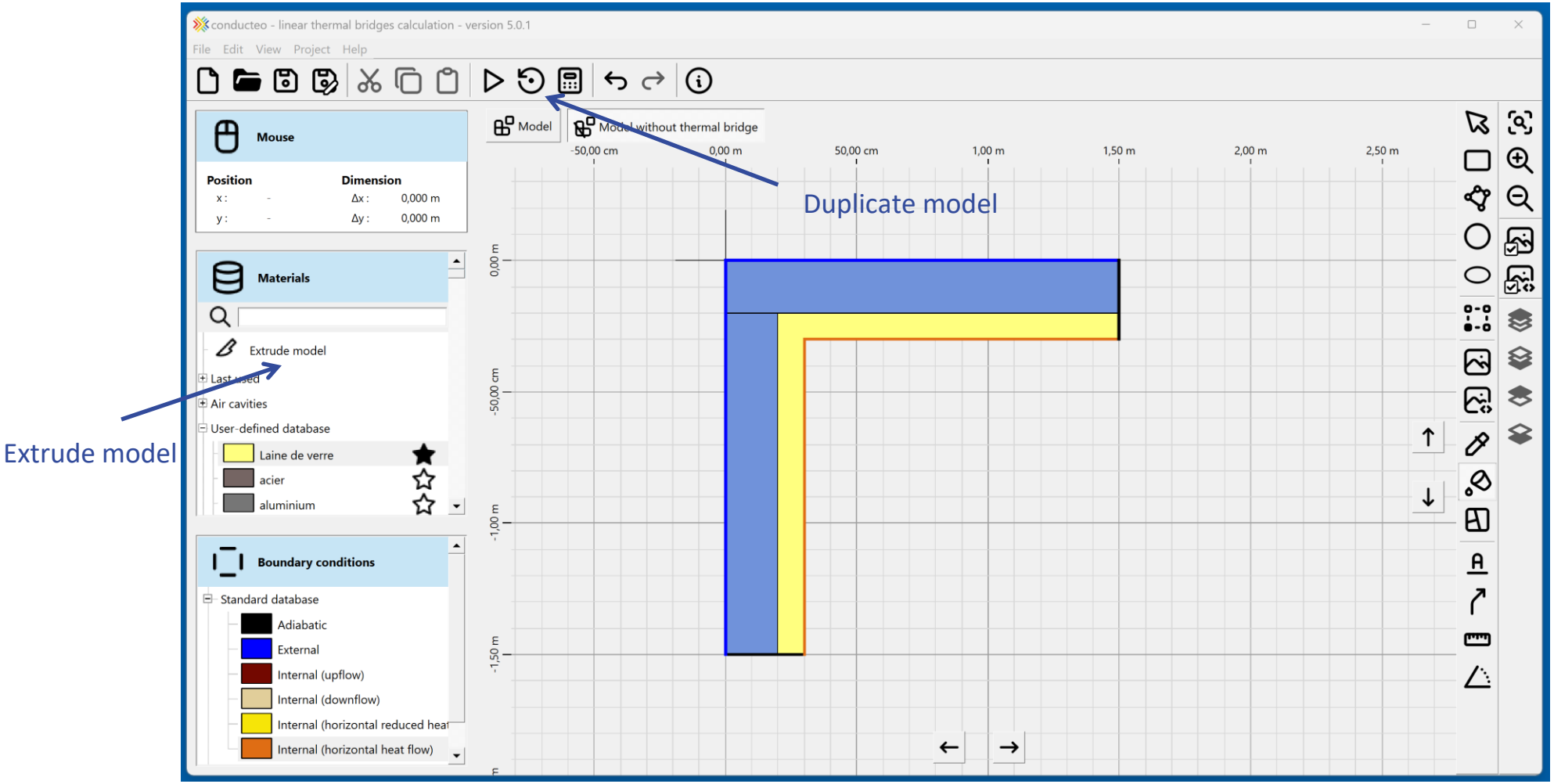
The “Materials and boundary conditions” button allows you to define boundary conditions on surfaces. The list of boundary conditions is displayed on the left.



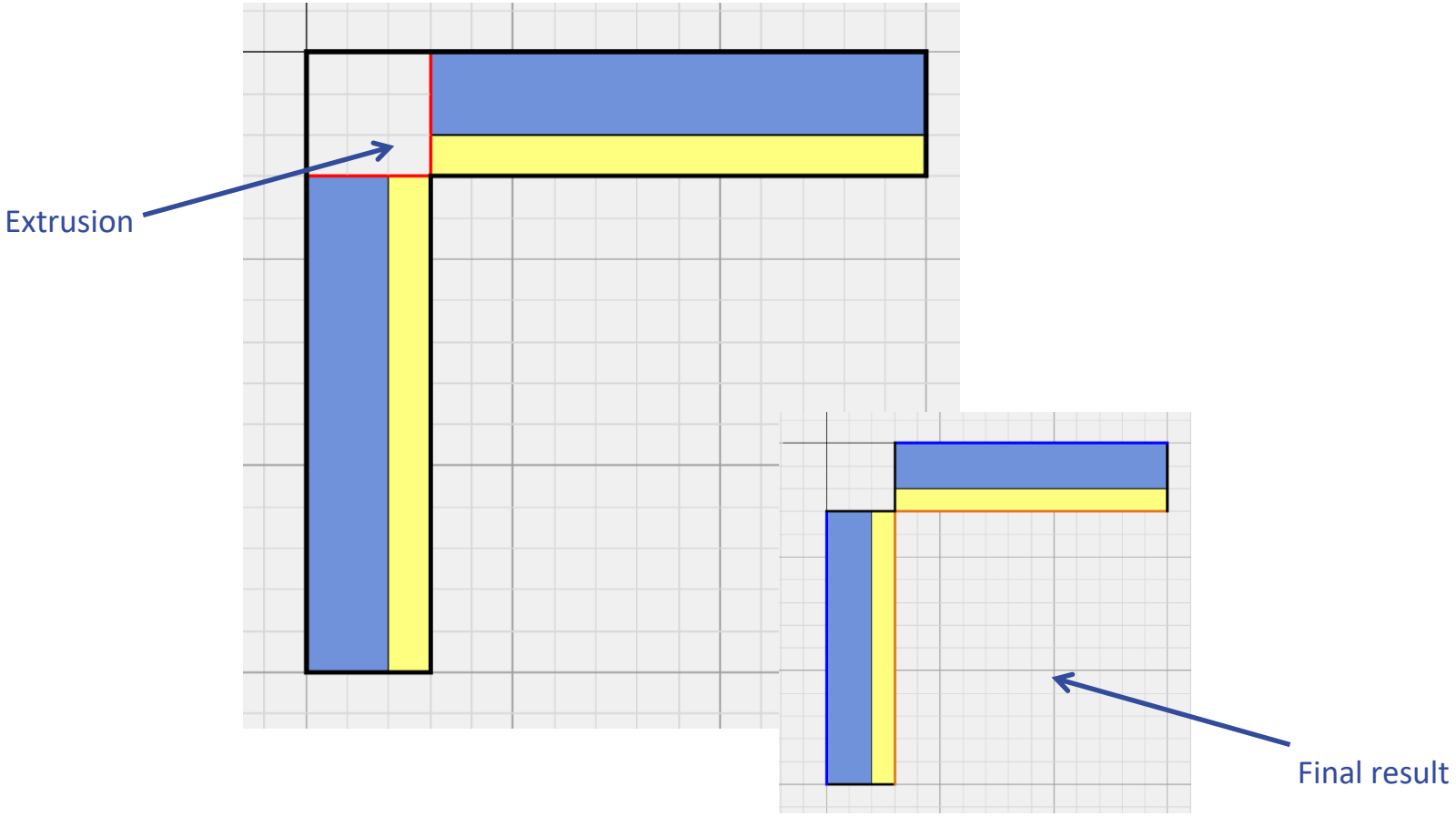
The modeling of the element with a thermal bridge is complete.
It is now necessary to model the element without a thermal bridge.



The “Duplicate model” option allows you to copy the model with a thermal bridge into the “Model without thermal bridge” tab.
“Extrude model” allows you to add extrusions to remove the junction creating the thermal bridge.

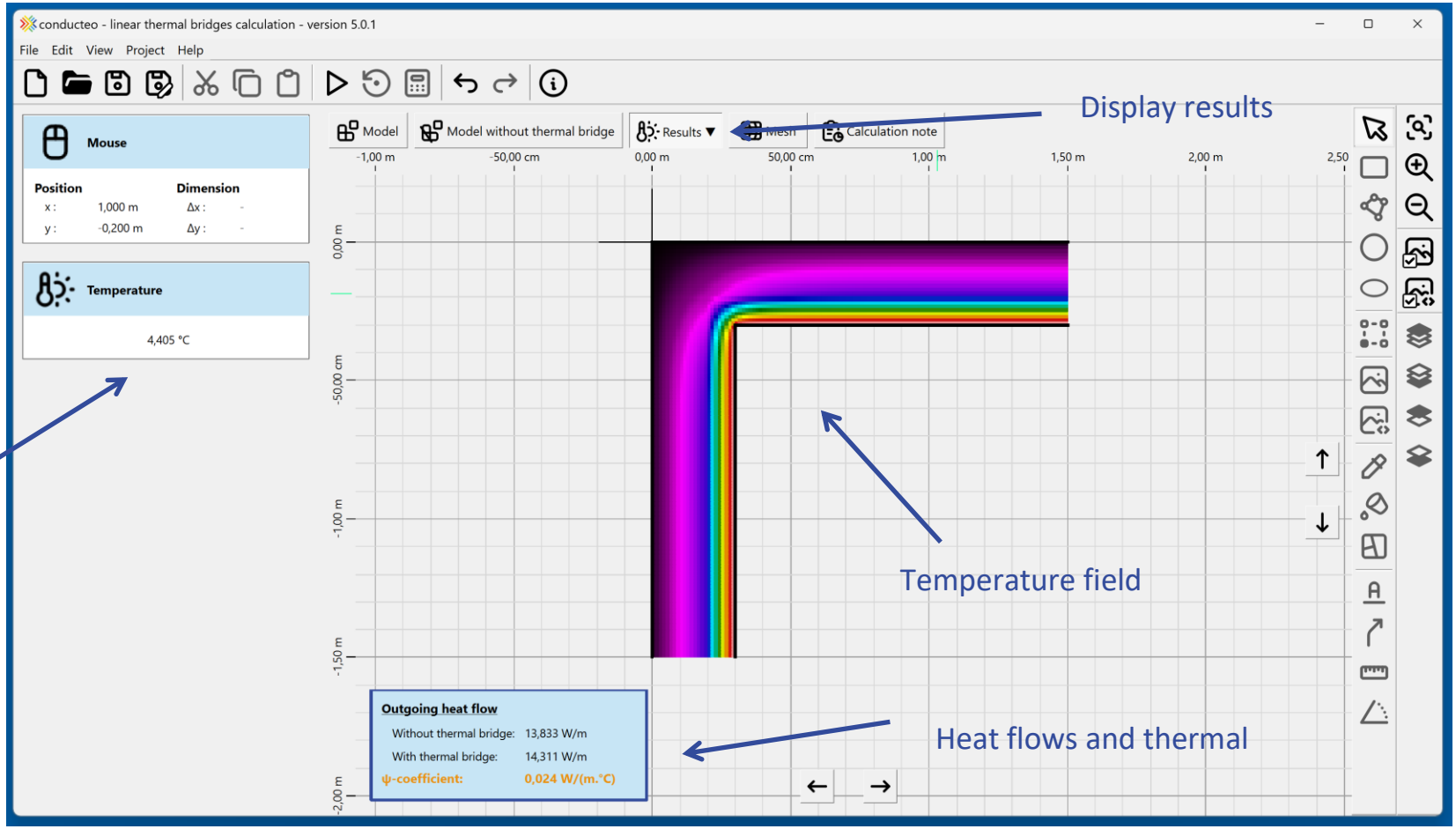


The junction between the two walls (creating the thermal bridge) is removed by extrusion: drawing a rectangular volume without material.

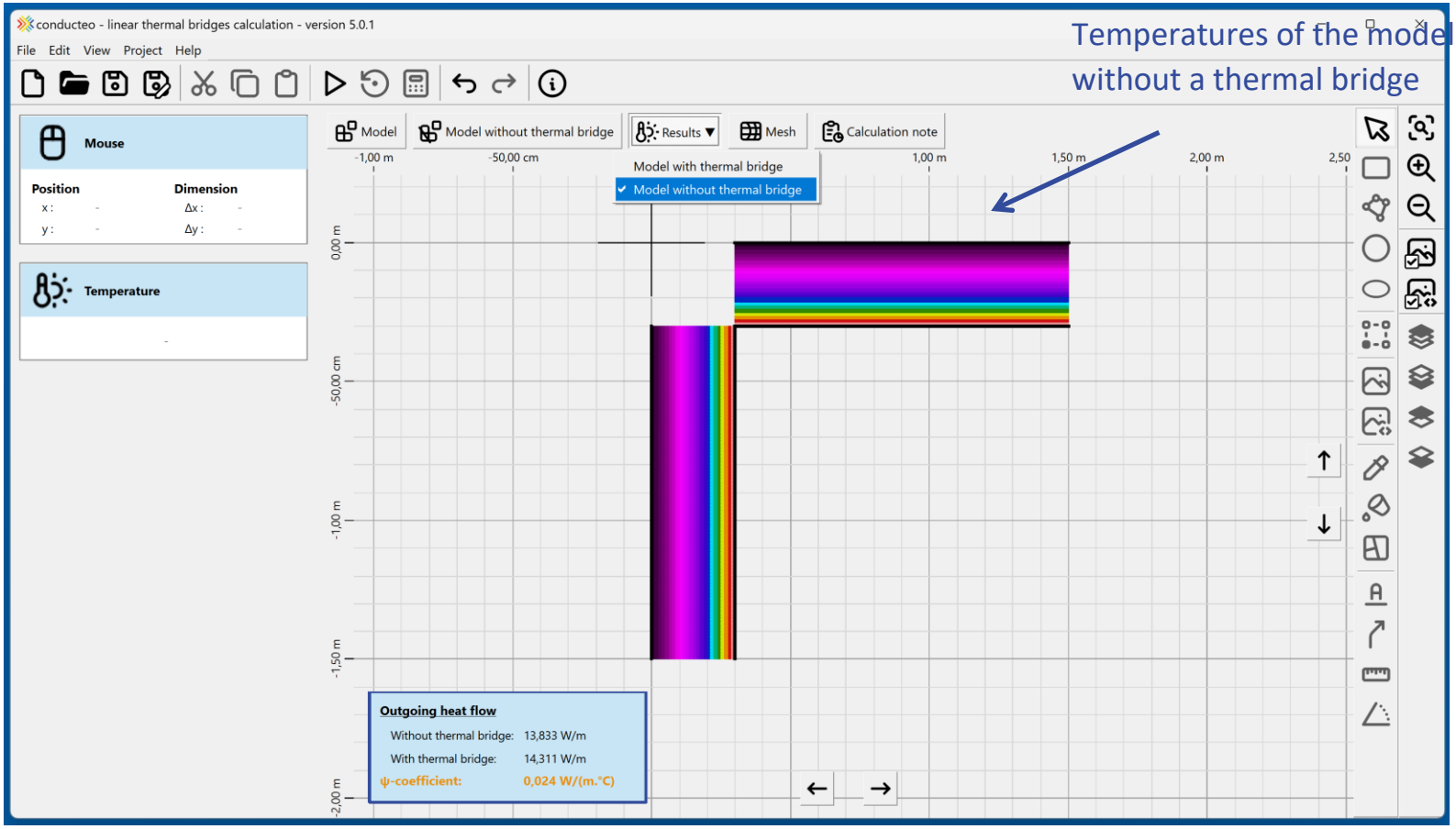


The simulation can now be started.

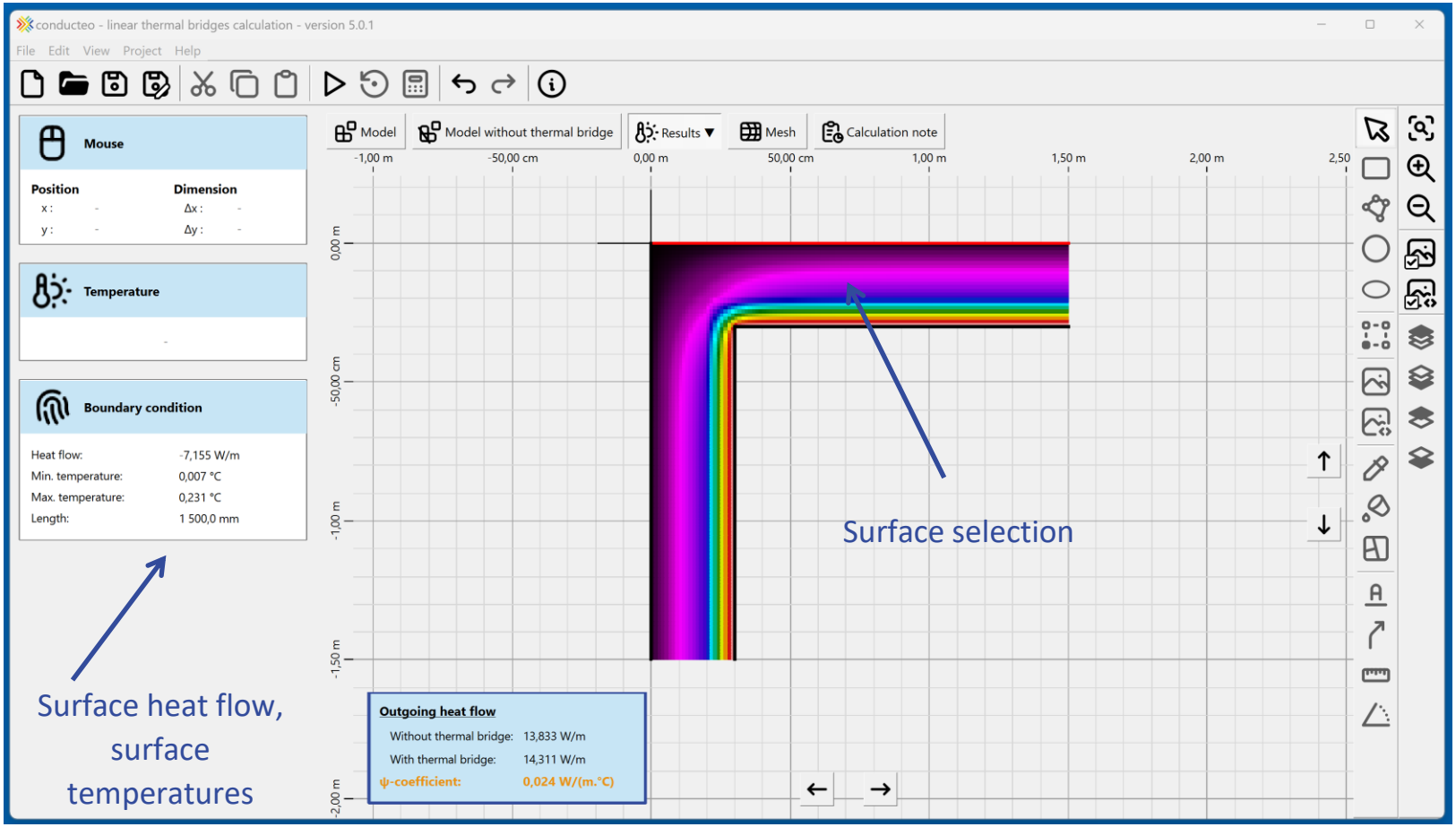
The “Results” tab allows you to display the temperature field.
The temperature under the cursor is shown on the left.
The heat flows for the models with and without a thermal bridge, as well as the thermal bridge value, are displayed.



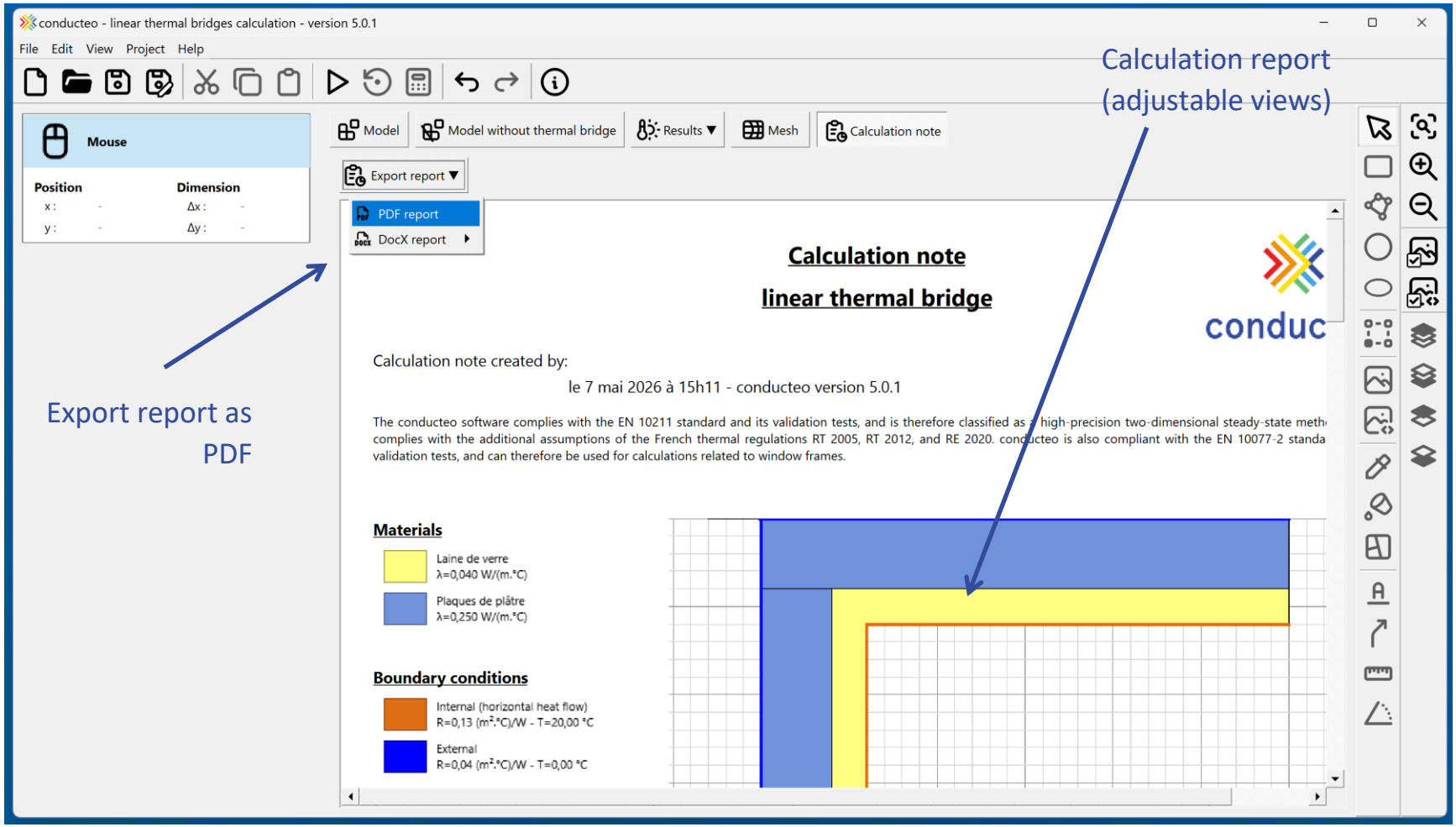
The “Results” tab allows you to display the temperature field for the model without a thermal bridge.



By selecting surfaces, the heat flow and surface temperatures are displayed on the left side of the software.



The “Calculation report” tab allows you to display an automatically generated report.
The model views can be adjusted with the mouse (wheel to zoom, middle-click to move the view).
The “Export report” button allows you to save it as a PDF or DocX file.



You are now ready to get started!