Department of Microelectronics and Computer Science
Technical University of Lodz

Cooling Process

M.Sc. Krzysztof Slusarczyk
kms@dmcs.p.lodz.pl

Lodz, August 2002

Problem description: the cooling process is used widely in modern electronic (both power and microelectronics) to decrease the temperature of electronic devices. The increase of the temperature is caused mostly by current flow in semiconductor structure. Because of high thermal resistance of the silicon as a material, there is a strong need to reduce the temperature in the structure. What is more important, the whole structure never heats uniformly, there are local fluctuations of the temperature, which can cause the damage of the whole device (in fact, such thermal fluctuations are far more dangerous and in most cases cause permanent damage of the device – especially bipolar devices are endangered – positive thermal feedback). That's why it is so important to design adequate cooling system to each semiconductor device or system.

In this tutorial we will be analyzing a very simple model of a microprocessor "Pentium 2" class. In Fig. 1 an example of such processor used in this tutorial is presented. All models will have the same geometry and dimensions, your task will be to change radiators and cooling conditions and also to find the most optimal solution to this problem.
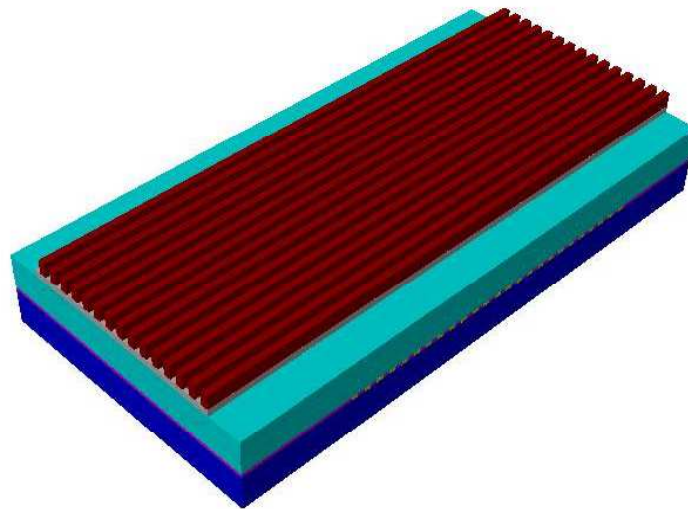

Fig. 1. One of the structure used in simulations

Part One

Pre-Processing

1. Setting the Layout Parameters

Start a new project in CFD – Micromesh.
Firstly set *Display Unit* as millimeters (from *View* in top toolbar select *Display Units*). Then click on *Change Boundary* button in toolbar, and set *Dimensions and Resolution* as it is shown in Fig. 2.
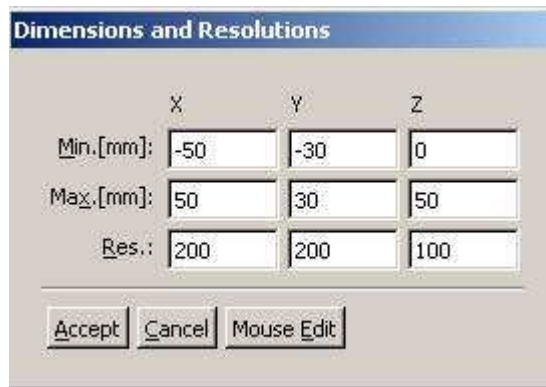
Fig. 2. Dimensions and resolutions of a new project

At the end, click on the *Change Grid* and set *Grid* to 1mm.

2. Creating a Layout

Click on *New Layer* button in the top toolbar to create a new layer. In the list of layers, click RB on the *NEW: Air (deposit)* layer, in the *Layer Properties* window which appears, click *Edit Material* button. Click *Add* button in *Materials* window. In the *New Material* window type names of new layers to be added: **package1**, **package2**, **package3**, **silicon**, **metal**. Three different "package" layers are needed to draw a whole chip containing silicon structure inside. Close the *Material* window.

Back in the *Layer Properties* window of the first created layer:
– in the *Material* pull-down list, change the name from **Air** to **package1**
– change *Thickness [mm]* to 5
– accept changes

Draw the bottom part of the package:
– Pos.X –40; Pos.Y –20; SizeX 80; SizeY 40

Add a new layer, in the *Layer Properties* window of the second created layer:
– in the *Material* pull-down list, change the name from **package1** to **silicon**
– change *Thickness [mm]* to 0.5
– accept changes

Draw the silicon part of the chip:
– Pos.X –25; Pos.Y –15; SizeX 51; SizeY 30

Add a **metal** layer (thickness 0.5mm) and draw 25 block of dimensions: 1 by 5 mm. The distance between two metal blocks should be 1 mm. Your layout should be similar to the one presented in Fig. 3.
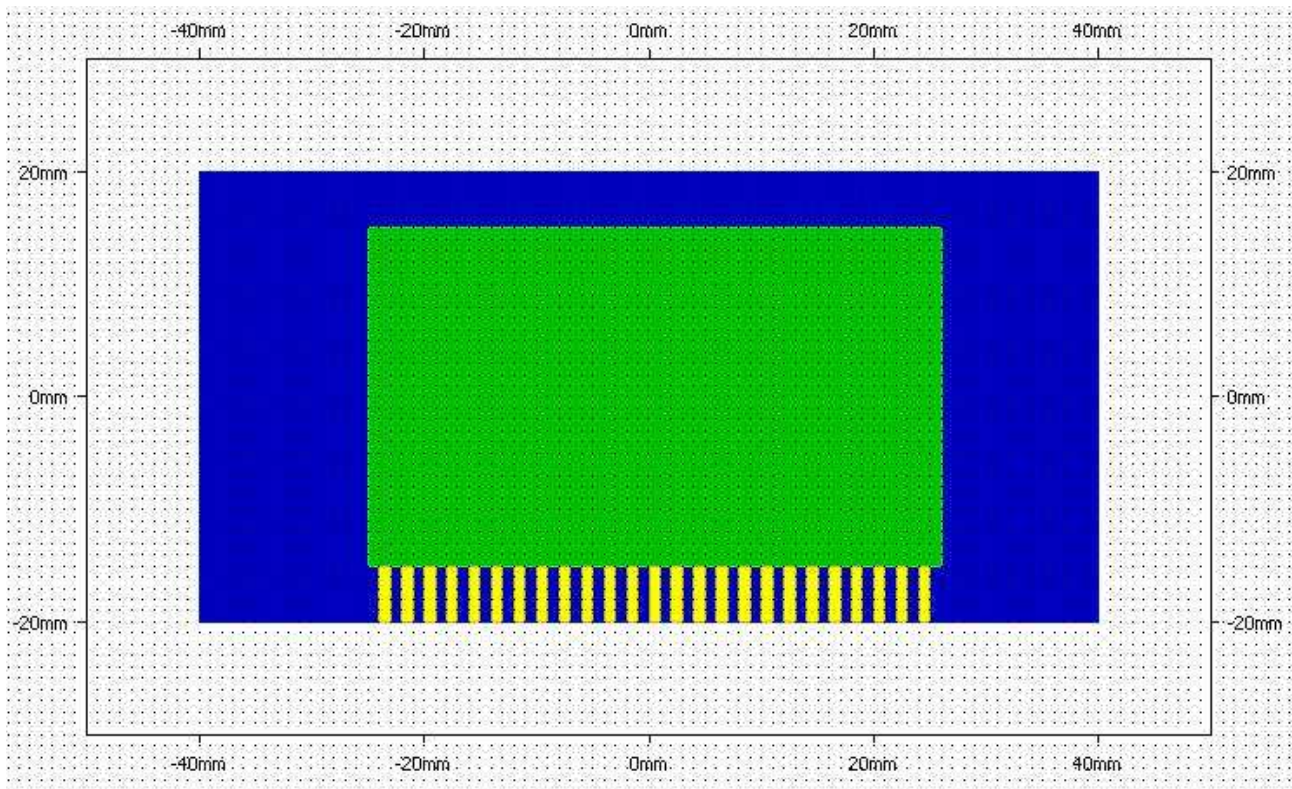
Fig. 3. The view of the layout after drawing metal blocks

The next step is creating a **package2** layer. This layer is specially added to fulfill the space not occupied by **silicon** or **metal** layers. The thickness of this layer is also 0.5mm. Do not forget to fill space among **metal** blocks.
*Tip: start from filling space among **metal** using copy / paste function, then draw blocks around **silicon** layer.*

The final step in this part of tutorial is to define **package3** layer, which covers all lower layers. The dimensions and thickness of this layer is identical as **package1**.
*Tip: after defining a layer, go to layer **package1**, copy the layer in the Editor window, then go to **package3** layer and paste it.*

3. Generating a Quick View

Click on the *Quick View* button in the top toolbar to visualize your work. This simple model visualization will be not quite interesting, but it will show you results of your work. During next parts of this tutorial you will add a different kinds of radiators to the structure, what will make the view much more attention-grabbing.

4. Build a 3D Computational Mesh

From the top toolbar select the *Prism-Hex Mesh* button to build a 3D computational mesh. The *Grid Parameters* window appears, set parameters as it is shown in Fig. 4.
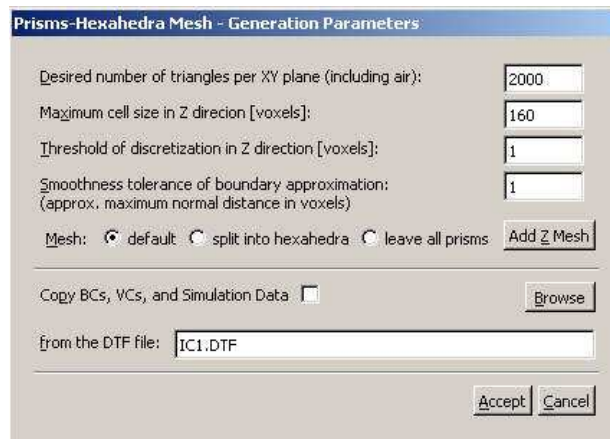
Fig. 4. Parameters of the Prism-Hex Mesh

By setting the numbers in this window, you may control the quality and the resolution of the generated 3D mesh, which will be finally stored in a DTF file.

Because we want to see the temperature distribution inside the model, we have to define also mesh in Z-direction. To do this click on *Add Z-Mesh* button. The *Additional Mesh Distribution in Vertical Direction* window appears. Our structure is 10.5 mm high (two layers of package – each 5 mm and one silicon layer – 0.5 mm), we need a grid point every 0.5 mm that makes 22 grid points in the Z-direction. Your window should resemble the one shown in fig. 5.
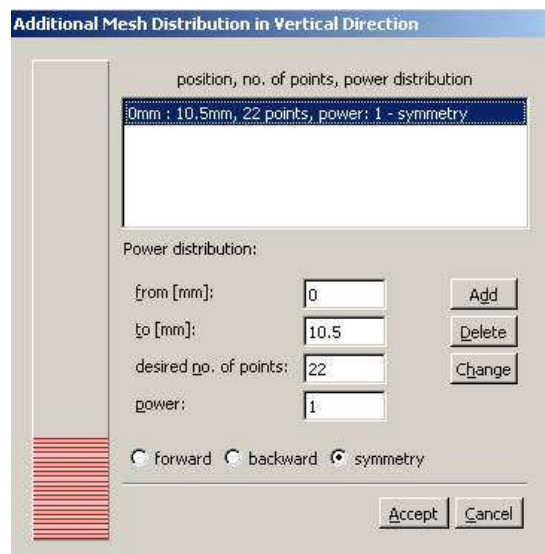

Fig. 5. Parameters of the Additional Distribution in Vertical Direction

Before mesh generation you will be asked to save your file.

5. Viewing the Generated 3D Mesh (DTF File)

Select the CFD – VIEW button from the toolbar. This button invokes the CFD – VIEW program installed on your machine in the CFDRC package. The DTF file just generated by CFD – Micromesh is opened automatically in CFD – VIEW.

In the *Object* window in the bottom – right corner, select *Geom.* From the toolbar select the following buttons (hover the mouse cursor over the buttons to see their description):

– *Grid on*
– *Surface on*
– you may select also *Flat Shading* from the top toolbar

Use LB to rotate the 3D mesh, MB to zoom in or out, RB to shift it. Your meshed 3D model in CFD – VIEW should look similar to the one shown in Fig. 6.
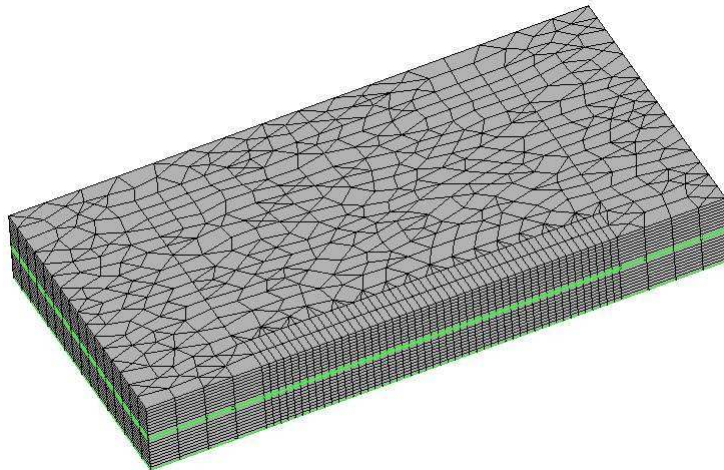

Fig. 6. Generated mesh for a chip

6. Preparing to Thermal Simulation of a Model

Select the CFD – GUI button from the top toolbar in CFD – Micromesh. This button invokes the CGF – GUI program installed on your machine in CFDRC package. Within CFD – GUI you can apply boundary conditions appropriate for your problem, set up analysis parameters, and start simulations.

7. Setting up a Thermal Simulation

PT (Problem Type)
Tick modules:
– *Heat Transfer (Heat)*

MO (Module Options)
Leave defaults settings.

VC (Volume conditions)
Select all package parts (**package1**, **package2**, **package3**):
– in Shared panel
   • set *Rho* to 3000 kg/m³
– in Heat panel
   • set *Specific Heat Cp* to 1000 J/kg*K
   • set *Thermal Conductivity K* to 50 W/m*K

Select **silicon** part. In Shared panel, at the left from *Material Name*, there is a *Import Material Data* button. Import **silicon** data from *Material Database*; note,

that silicon is in *Metals*.
If you do not have connection to Material Database, set numbers given below:

– in *Shared* panel
  • in *Material Name* type Silicon
  • set *Density Rho* to 2329 kg/m^3
– in Heat panel
  • set *Specific Heat Cp* to 710 J/kg*K
  • Set *Thermal Conductivity K* to 163 W/m*K

Select metal parts. In Shared panel, at the left from *Material Name*, there is a *Import Material Data* button. Import **aluminium** data from *Material Database*; note, that aluminium is in *Metals*.
If you do not have connection to Material Database, set numbers given below:

– in *Shared* panel
  • in *Material Name* type Aluminium
  • set *Density Rho* to 2698.9 kg/m
– in *Heat* panel
  • set *Specific Heat Cp* to 900 J/kg*K
  • set *Thermal Conductivity K* to 210 W/m*K

BC (Boundary Conditions)
Select **package1** and all **AIR_...** boundaries
– group them
– in *Heat* panel
  • set *SubType* to *Adiabatic*

Select all silicon_... interfaces (package1_silicon, silicon_metal, silicon_package2, silicon_package3):
– group them
– in *Heat* panel
  • tick *Wall Heat Source*
  • set *Wall Heat Source Value* to 500000 W/m^2

SC (Solver Control)
– in *Iteration* panel
  • set *Max. Iterations* to 30

Out
– in *Graphic* panel
  • select all files

Run
Before you run the simulation, save your file.
Click *Submit to Solver* button to start the simulation process.

You can press View Residuals or View Output buttons to see real-time displays of the residual history and output of the file contents.

8. Simulation Post Processing in CFD – VIEW

Go to CFD – VIEW window and open the DTF file.

In the Objects window in the bottom-right corner select Geom. From the toolbar select following buttons:
– *Grid on*
– *Surface on*

Select the variable you want to display in Primary Variable pull-down menu, e.g. *T* for temperature or *H0* for enthalpy.

In the *Annotation* panel on the right-side (click *A*), select the *Legend* to see values of the displayed variable, change the *Size* of the legend to 12. After your legend is set, click again on *Geom.*

Use LB to rotate your model, MB to zoom in or out the figure, RB to shift it.

You may choose to turn on the Flat Shading or Perspective option in the top toolbar for better view of the device. The grid may be turned on or off. Your results in CFD – VIEW should be similar to the one presented in Fig. 7.
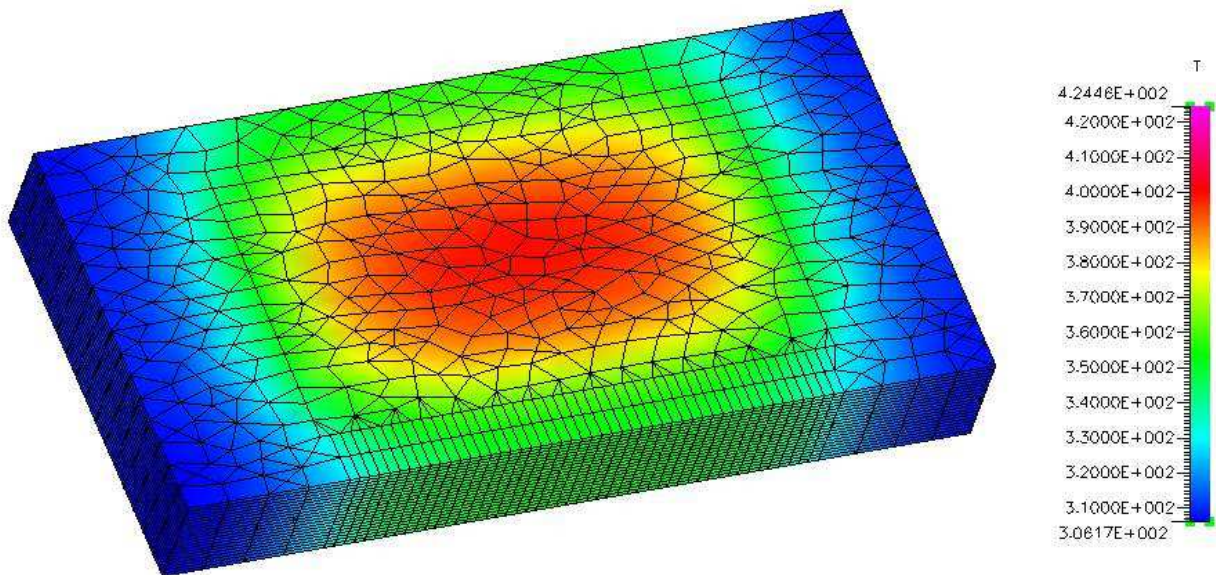

Fig. 7. The result of the simulation

What you are looking at is the external view of the device, as you can see the temperature at the top (or bottom) surface of the device reaches about 390K, but the legend represents the temperature range up to 422K. To see the full range of the temperature distribution, you need to see the cross-cut of the device.
To do this:
– click on *Geom*
– *Grid off*
– *Surface off*
– select **Off** as *Primary Variable*

Then select *Y-cut* from right-side *Surfaces* toolbar. Click on the following buttons from the toolbar:
– Grid on

– Surface on

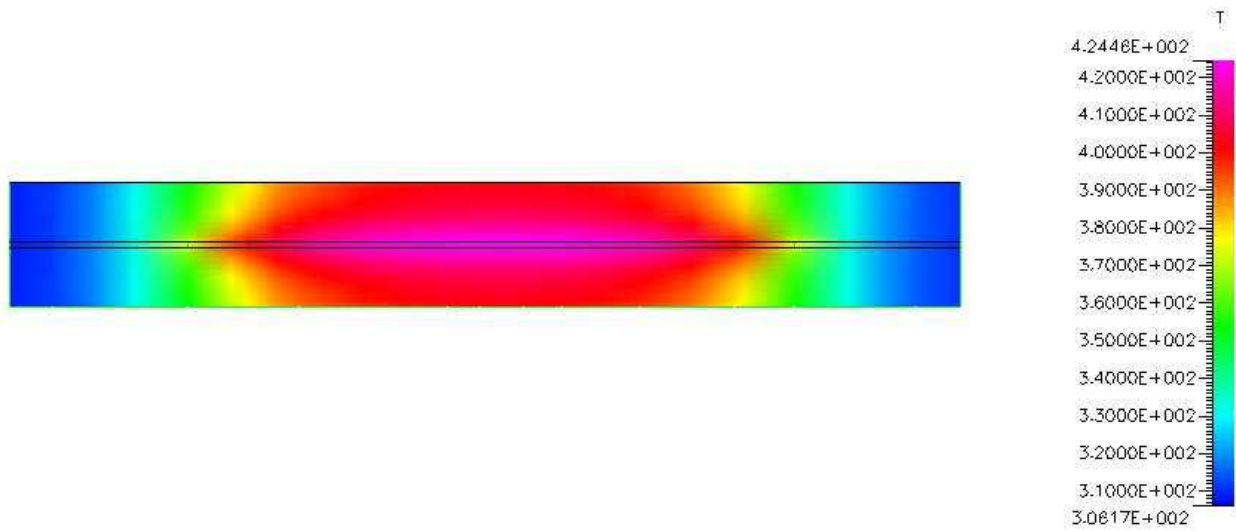Your results in CFD – VIEW should be similar to the one presented in Fig. 8 and 9.

Fig. 8. The temperature distribution at the cross-cut of the structure
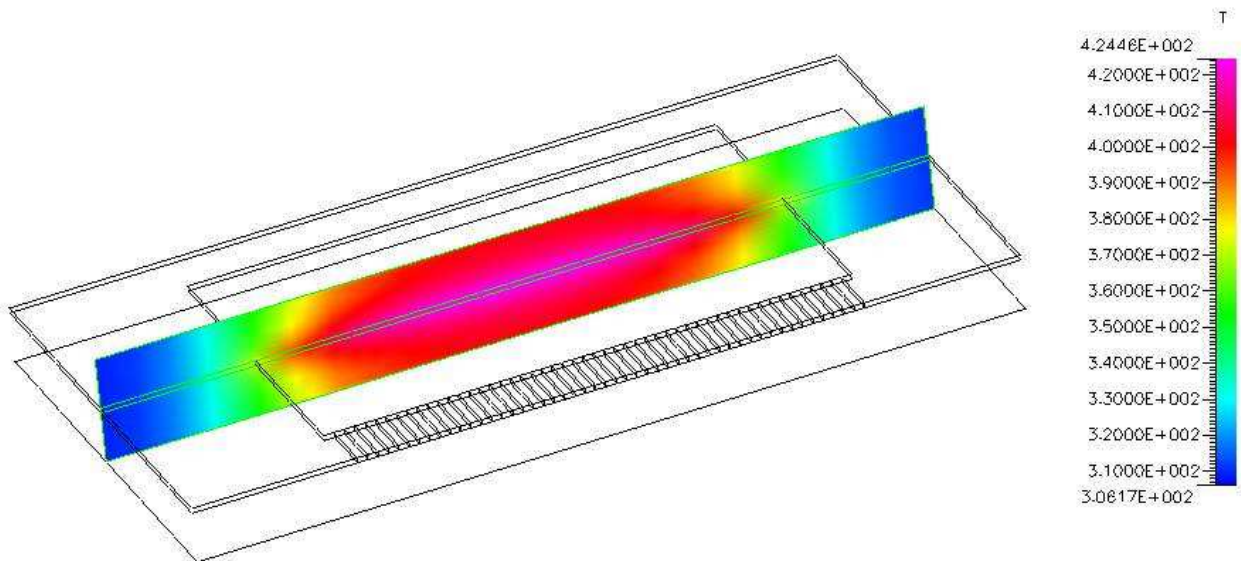

Fig. 9. The temperature distribution at the cross-cut of the structure

Use slider to see temperature distribution in different cuts, not only in central position (Fig. 10 and 11).
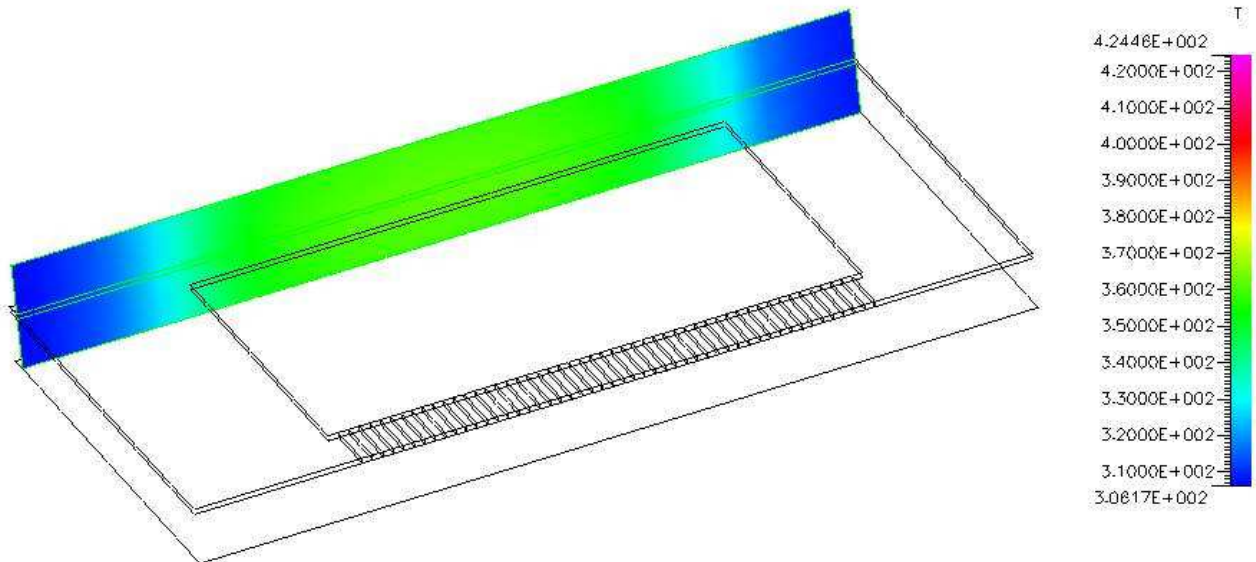
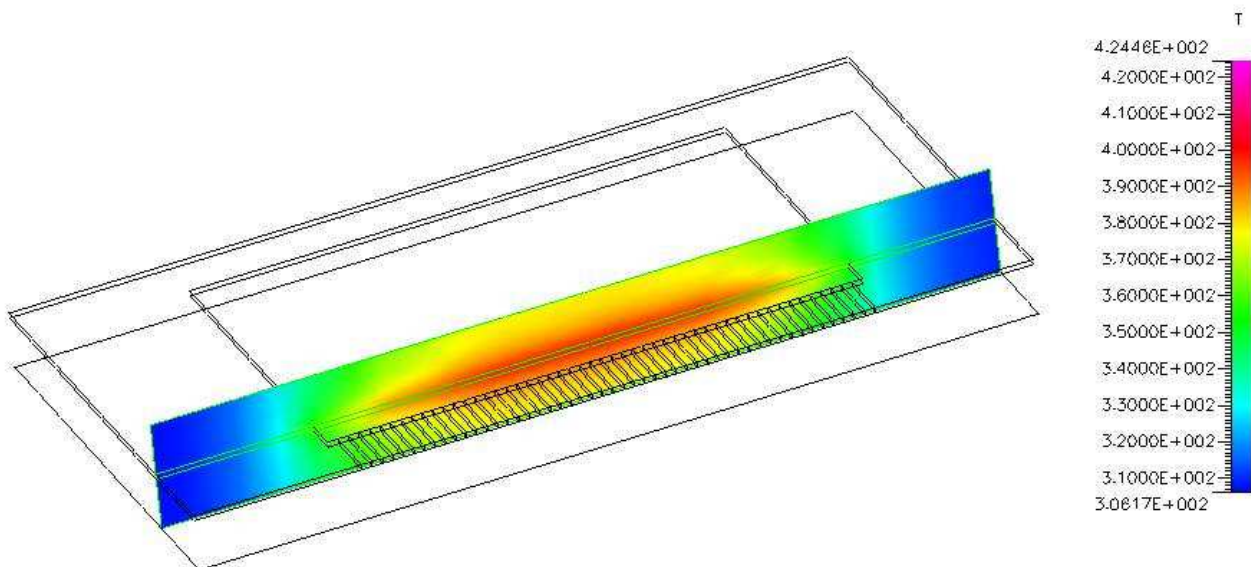*Fig. 10. The temperature distribution at the cross-cut of the structure*

Fig. 11. The temperature distribution at the cross-cut of the structure

9. Model Modifications 1 – Adding a Radiator

The temperature received in the simulation (424K – that is 151°C) is far too high for the microprocessor to work. We try to reduce it by adding a simple radiator to the structure.

Go back to CFD – Micromesh and save your project under a different name. Then add a new layer, name it radiator1. Our radiator will be a simple block of copper placed at the top of the structure. In the properties of the new layer set thickness to 1 mm. Draw a rectangle of the dimensions shown in Fig. 12.
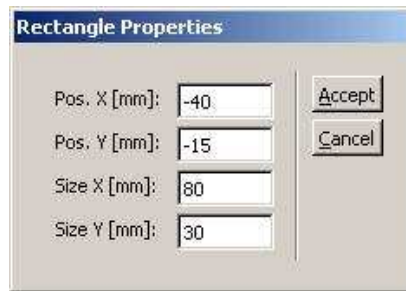
Fig. 12. Dimensions of the radiator

Save your file and repeat Step 4 through Step 8.

Caution: your modified structure is higher than previous one, so you must increase the Z-direction to 11.5 mm and number of grid points to 24 in *Additional Mesh Distribution in Vertical Direction* window.

In the CFD – GUI define material in VC as copper. If you do not have such material in Material Database, set the following numbers:
– in *Shared* panel
  • in *Material Name* type Copper
  • set *Density Rho* to 8960 kg/m
– in *Heat* panel
  • set *Specific Heat Cp* to 385 J/kg*K
  • set *Thermal Conductivity K* to 385 W/m*K

Can you see any differences between results obtained in this and previous model? Can you explain those differences?

10. Model Modifications 2 – Modifying a Radiator

Our next step is to modify a simple radiator designed in Step 9.

Go back to CFD – Micromirror and save your project under a different name. Add a new layer and name it radiator2. We will add a few blocks of copper to increase the surface of the radiator. In radiator2 layer properties set 2 mm as a thickness of the layer. Draw 15 blocks of dimensions 80 by 1 mm, the space between two blocks is 1 mm. Your radiator in Quick View should be similar to the one presented in Fig. 12.
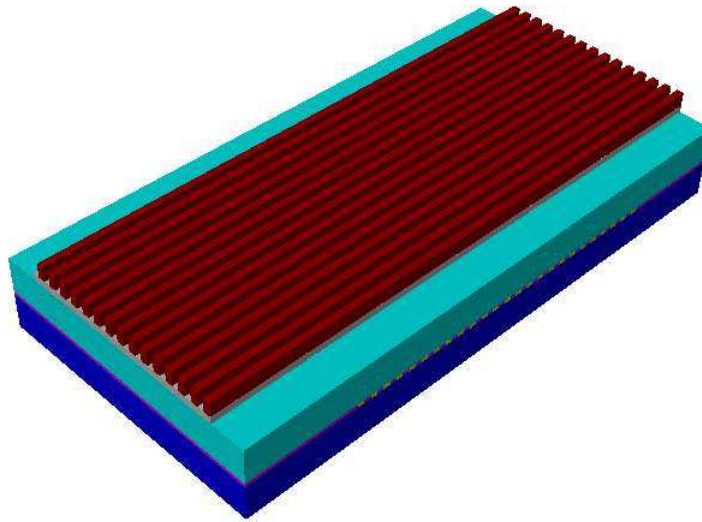
Fig. 12. The chip with modified radiator

Caution: your structure is higher than previous one – change its height and number of grid points in Z-direction.

Save your file and repeat Step 4 through Step 8. In the CFD – GUI define material in VC as copper. If you do not have such material in Material Database, set the same numbers as in Step 9.

Compare your results to previous ones.

11. Model Modifications 3 – Modifying a Radiator

In this step you will modify a radiator designed in Step 10.

Go back to CFD – Micromesh and save your project under a different name. Increase the thickness of the radiator2 layer to 5 mm.

Caution: your structure is higher than previous one – change its height and number of grid points in Z-direction.

Save your file and repeat Step 4 through Step 8.

12. Model Modifications 4 – Adding Air Cooling 1

In this step you will modify a model simulated in Step 7.

Go back to CFD – GUI and save your file under a different name. In this step you will only change one boundary condition. You will change wall sub type from adiabatic to isothermal, that will tell the simulator, that fixed temperature is set to this wall.
Go to **BC** (Boundary Conditions)
– select **AIR_package3** issue
  • change *Heat SubType* from **Adiabatic** to **Isothermal** and set the *Temperature T* to 300K
– apply changes

Finish Step 7 and repeat step 8.

Compare simulations results.

13. Model Modifications 5 – Adding Air Cooling 2

In this step you will modify a model simulated in Step 11.

Go back to CFD – GUI and save your file under a different name. Change boundary condition for walls named **AIR_package3** and **AIR_radiator1** in the way described in Step 12.

Compare simulation results.

14. Model Modifications 6 – Adding Air Cooling 3

In this step you will modify a model simulated in Step 12.

Go back to CFD – GUI and save your file under a different name. Change boundary condition for walls named **AIR_package3**, **AIR_radiator1**, **AIR_radiator2** in the way described in Step 12.

Compare simulation results.

15. Model Modifications 7 – Adding Air Cooling 4

In this step you will modify a model simulated in Step 13.

Go back to CFD – GUI and save your file under a different name. Change boundary condition for walls named **AIR_package3**, **AIR_radiator1**, **AIR_radiator2** in the way described in Step 12.

Compare simulation results.

Part Two

Project

16. Project 1

It is quite common to over-clock the microprocessor in PC-computer, the amount of the heat emitted by the microprocessor increase dramatically, what quite often leads to thermal breakdown of the structure and destruction of the processor itself.
Increase the Wall Heat Source twice and simulate all structured without and with the additional air-cooling. Remember, that both Intel® and AMD® processors work properly in the temperature up to 50°C. Suggest the most adequate cooling solution to this problem.

17. Project 2

All cases simulated above are very rare in common life. In real microprocessor a few heat sources can be found, and never the whole structure heats uniformly.

The idea of the project is to design a processor with one heat source. You are going to simulate two cases described below.

In this project you will design a structure very similar to the one tested above. The new structure should involve three new layers: **silicon2**, **silicon3**, and **package4**, all of them should have thickness of 0.5 mm. **Silicon2** will have dimensions of 20 by 20 mm and will be placed on **silicon1** (previously named **silicon**) in the center of the structure (but you can move it in the direction you want, to make the simulation not symmetrical one). This layer will be also the one and only heat source in the whole structure. The **silicon3** layer is the additional one, just to fulfill the space above **silicon1** and not occupied by **silicon2**. Remember to increase the thickness of **package2** to 1 mm. Add also **package4** layer to fulfill the space above the metal layer.

Run two simulations. In the first one, define the interface between **silicon2** and **silicon3** as the *Wall Heat Source*; in the second simulation define all **silicon2_...** interfaces as *Wall Heat Source*. In both simulations set the value of the *Wall Heat Source* to 5e+5 W/m².