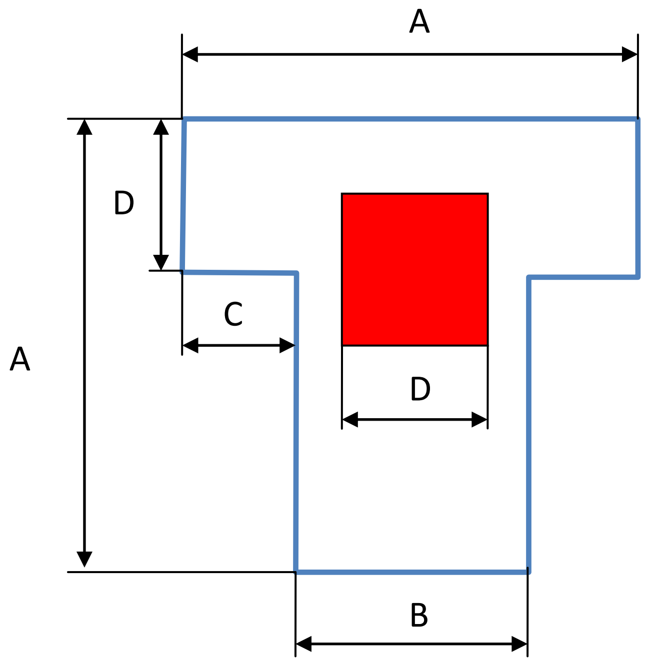
Laboratory exercise no. 2: Heat transfer simulation

Aim: The aim of the laboratory is to create a numerical simulation of heat transfer in a specified object.

Laboratory programme:

1. Defining of the physical and mathematical model describing heat transfer phenomena.
2. Applying the discretization of the equations and implementation of initial and boundary conditions.
3. Writing the code.
4. Numerical stability testing.
5. Calculation of the spatial and temporal temperature distribution In the plate for boundary condition of type 1.
6. Calculation of the spatial and temporal temperature distribution In the plate for boundary condition of type 2.
7. Calculation of the temperature increment In the plate for the boundary condition of type 2 and comparison with the theory.

Physical object:



The modelled object is metal plate made of alumina, cooper or stainless steel presented on the above figure. In the middle of the plate there is a heater attached (red square)

The values of parameters A-h are specified by a tutor during the laboratory.

The equation describing the modelled physical phenomena to be solved (a constant and isotropic thermal conductivity coefficient is assumed):

$$\frac{\partial T}{\partial t} = \frac{K}{c_w \rho}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)$$

Boundary condition type 1:

The area having contact with heater has a constant temperature equal to 80°C, while the edge of plate has a temperature equal to 10°C during the whole simulation. Such condition is obtained in mathematical model by application of Dirichlet condition (force constant temperature value at the simulation boundaries)

$$T(r_1,t)=80°C$$
$$T(r_2,t)=10°C$$

where:

r_1 – part of the plate belonging to red area
r_2 – part of the plate belonging to blue edge

Boundary condition type 2:

We assume a constant power of the heater equal to 100W during the first 10s of the simulation. The heater is switched off during the later times. The edges of the plate are thermally isolated from the environment.

Therefore for each computational node having contact with the heater in each time step the increase of temperature caused by a heater is equal to:

$$\Delta T=\frac{P\cdot dt}{c_w\cdot B^2\cdot h\cdot \rho}$$

The thermal insulation of the plate edges is obtained by von Neumann boundary condition (derivative of the temperature function is equal to zero)

$$\frac{\partial T(r_2,t)}{\partial r}=0$$

where:

r_1 – part of the plate belonging to red area
r_2 – part of the plate belonging to blue edge
P – power of the heater
K- thermal conductivity coefficient of the plate material
B^2 – heater area
c_w – specific heat of the plate material
h – plate thickness

Initial condition:

$$T(r,0)=20°C$$

Derivative approximation:

$$\frac{\partial T}{\partial t}=\frac{T_{i,j}^{n+1}-T_{i,j}^{n}}{\Delta t} \qquad \frac{\partial^2 T}{\partial x^2}=\frac{T_{i+1,j}^{n}-2T_{i,j}^{n}+T_{i-1,j}^{n}}{(\Delta x)^2}$$

Equation discretization:

$$\frac{T_{i,j}^{n+1}-T_{i,j}^{n}}{\Delta t}=\frac{K}{c_w\rho}\left(\frac{T_{i+1,j}^{n}-2T_{i,j}^{n}+T_{i-1,j}^{n}}{(\Delta x)^2}+\frac{T_{i,j+1}^{n}-2T_{i,j}^{n}+T_{i,j-1}^{n}}{(\Delta y)^2}\right)$$

$$T_{i,j}^{n+1}=T_{i,j}^{n}+\frac{K\Delta t}{c_w\rho(\Delta x)^2}\left[T_{i+1,j}^{n}-2T_{i,j}^{n}+T_{i-1,j}^{n}\right]+\frac{K\Delta t}{c_w\rho(\Delta y)^2}\left[T_{i,j+1}^{n}-2T_{i,j}^{n}+T_{i,j-1}^{n}\right]$$

Input data:

dx – distance between computational nodes in x direction
dy – distance between computational nodes in y direction
dt – time step
K – thermal conductivity coefficient
Nt – number of time steps
NX – number of computational nodes in x direction
NY – number of computational nodes in y direction

Approximated values of physical parameters: *(after www.wikipedia.pl)*

alumina:

Density	ρ=2700 kg/m^3
Specific heat	c_w=900 J/kgK
Thermal conductivity	K=237 W/mK

cooper:

Density	ρ=8920 kg/m^3
Specific heat	c_w=380 J/kgK
Thermal conductivity	K=401 W/mK

stainless steel:

Density	ρ=7860 kg/m^3
Specific heat	c_w=450 J/kgK
Thermal conductivity	K=58 W/mK

Laboratory outline:

1. Writing the computer programme simulating heat transfer using the method described above
2. Calculation of the time evolution of temperature distribution in a metal plate made of selected material for specified initial condition and boundary condition of type 1
3. Testing of the numerical stability for different values of time steps and spatial resolution.
4. The calculation of temporal evolution of spatial temperature distribution and final temperature (simulation is running until steady state condition (to be defined by a student), while the heater is switched off after 10s of the simulation) in the plate for

defined initial condition and the 2-nd boundary condition for alumina and cooper (comparison of the results obtained for two metals).

5. The comparison of final temperature increment for 2-nd type of boundary condition with theoretical heat balance calculation.
6. Optional introduction to the model the heat dissipation mechanism for the whole surface of the plate (heat exchange with the environment).
7. Computer programme can be written in any programming language or software environment. Recommended environment is MATLAB.
8. Programme code supplemented with appropriate comments should be included as a part of a report prepared in pdf format.