Abstracts of MMA2015, May 26–29, 2015, Sigulda, Latvia

THE ANALYSIS OF PARALLEL SOLVER FOR THE HEAT TRANSFER IN ELECTRICAL POWER CABLES.

ANDREJ BUGAJEV, RAIMONDAS ČIEGIS, VADIMAS STARIKOVIČIUS

Dept. of Mathematical modelling, Vilnius Gediminas Technical University

Sauletekio av. 11, 10223 Vilnius, Lithuania

E-mail: `andrej.bugajev@vgtu.lt, rc@vgtu.lt, vadimas.starikovicius@vgtu.lt`

In this work, we study the performance of parallel OpenFOAM-based solver for heat conduction in electrical power cables. For computational experiments, we use the following 2D benchmark problem:

$$\begin{cases} c\rho \dfrac{\partial T}{\partial t} = \nabla \cdot (\lambda \nabla T) + q, & t \in [0, t_{max}], x \in \Omega, \\ T(x,0) = T_b, & \text{when } x \in \Omega, \\ T(x,t) = T_b, & \text{when } x \in \partial\Omega, \\ [T] = 0, \quad [\lambda \nabla T] = 0 & \text{when } x \in \partial\Omega_D, \end{cases} \tag{1}$$

here $x = (x_1, x_2)$, $T(x,t)$ is temperature, $\lambda(x) > 0$ is heat conductivity coefficient, $q(x,t,T)$ is the source function, $\partial\Omega$ is the contour of domain Ω, $\rho(x) > 0$ defines mass density, $c(x) > 0$ is specific heat capacity, T_b, t_{max} are given constants. Operator $\nabla \cdot (\lambda \nabla T) = \sum_{j=1}^{2} \frac{\partial}{\partial x_j} \left(\lambda \frac{\partial T}{\partial x_j} \right)$ is the diffusion operator. The solution and flux continuity conditions are satisfied on boundaries of domains with different diffusion coefficients $\partial\Omega_D$.

When we need to deal with 2D and 3D mathematical models for the heat transfer in various media (metals, insulators, soil, water, air) and non-trivial geometries, only parallel computing technologies can allow us to get results in an adequate time. To solve numerically selected models, we develop our numerical solvers using the OpenFOAM package [1]. OpenFOAM is a free, open source CFD software package. It has an extensive set of standard solvers for popular CFD applications. It also allows us to implement new models, numerical schemes and algorithms, utilizing the rich set of OpenFOAM capabilities. The important consequence of this software development approach is that numerical solvers can automatically exploit the basic parallel computing capabilities already available in the OpenFOAM package.

In this work, we study and analyze the parallel performance of OpenFOAM-based solver for heat conduction in electrical power cables. The main goal is to consider the scalability and efficiency of the developed parallel solver in the case when the parallel system is not big, but it consists of non homogeneous multicore nodes. The mesh is adaptive and it is partitioned by using Scotch method. Then load balancing techniques must be used in order to optimize the parallel efficiency of the solver. The second aim is to investigate the sensitivity of parallel preconditioners with respect to the number of processes.

REFERENCES

[1] OpenFOAM. http://www.openfoam.org