Feasibility of COMSOL coupling to a reservoir simulator: electro-hydromechanical model

I.I. Bogdanov*, J.A. Torres, K. El Ganaoui and A.M. Kamp

Open and Experimental Center for Heavy Oil (CHLOE)
affiliated to the University of Pau, France,

*Corresponding author: CHLOE/ADERA - Bâtiment UFR Sciences et Techniques – Av. de l'Université – BP1155 - 64013 Pau Cedex France; *e-mail*: igor.bogdanov@univ-pau.fr

Abstract: Multi-physical environment of new technologies requires frequently a "multi-simulator" framework of its modeling. It means simply that one simulator is not always capable to model the necessary sequence of physical processes. Thus simulator coupling becomes more and more actual from the view-point of both the theory and the practice of numerical computations, in particular, for heavy oil recovery applications. Our current work is aimed at loose coupling of COMSOL *Multiphysics* to reservoir simulator governed by COMSOL *Script*. One particular example of heavy oil recovery method based on electrical (Joule) heating of the reservoir is presented in this paper. Main principles, modeling results, the coupling methodology advantages and drawbacks are discussed and completed by hopeful conclusions.

Keywords: low-frequency electric field, Joule effect, multiphase flow, thermal recovery method, reservoir simulations.

1. Introduction

Exploitation of non-conventional hydrocarbon resources becomes more and more challenging activity for energy producing companies throughout the world. Already known and novel approaches should be explored and proposed to provide reasonable technology for such an oil recovery.

Recently the COMSOL *Multiphysics* has been used for modeling the thermal multiphase flow through porous media in the different frameworks but using the same general approach [1-3]. This first experience demonstrates that some problems related to oil recovery applications can be solved successfully by fully integrated COMSOL models (FIM). However the modeling of the flow with phase transitions and/or under conditions of phase equilibrium for a multi-component fluid (*eg* water evaporation/condensation or so-called compositional models of oil recovery) in the "real" reservoir environment is not straightforward and requires considerable efforts. Reservoir characterization and fluid physical (and chemical) properties description are normally the important build-in blocks in any modern reservoir simulator. At the same time the flexibility and real multi-physical environment of COMSOL (together with other known advantages) make attractive to couple a dedicated simulator to a COMSOL model of physical phenomenon of interest like for example electric/acoustic/vibration fields *etc* (cf. [2-4]).

The main purpose of our work is to develop an efficient methodology for such a coupling which can be a promising tool for petroleum recovery numerical applications. It is worth to note that the coupling can be performed on the different base that may vary from a physical phenomenon introduced via its COMSOL model to the important advantage of the implementation of non-structured grid (and other offered functionalities) which takes part of built-in COMSOL *Multiphysics* environment.

As a successful example of application for such a coupling model (called EMIR in our team), the electric heating based method for heavy oil recovery is considered here (cf. the tentative model description in [2]). Another promising problem for coupling of such a type is the Darcy scale flow modeling with dynamic up-scaling of transport properties so that coupling takes place on multi-scale rather than multi-phenomenon base. For example, in figure 1 the unstable and stable two-phase flow modeling at Darcy and at pore scale is presented. The axial displacement in the hollow finite cylinder is characterized by unfavorable viscosity ratio (M=2000). The pore scale counterpart of the displacement is shown in the figure for the case of monodisperse circular grain pack (white circles). The COMSOL fully integrated model (FIM) used for pore scale flow is based on Navier-Stoks equations modified with Cahn-Hilliard model of two fluids interaction.

The fully integrated COMSOL models (FIMs) are useful and their development is feasible in or even beyond the framework of simulators coupling. Indeed, the coupling models (CMs) validation can't be done effectively without comparison to analytical and semi-analytical solutions available but also to the results of fully integrated COMSOL models (FIMs). Developed sometimes under simplified assumptions as compared to more or less realistic problem description they demonstrate excellent numerical performance and are capable to generate reference solution for great variety of cases under consideration. The FIMs development and validation are also presented and discussed in our paper.

2. The coupling paradigm

New ideas arise and their development for novel technologies needs new modeling tools. However, suitable simulators exist not for each new idea and at least at the tentative exploration period of the technology development it requires non-standard approaches and means.

Frequently the leading idea comes from a non-trivial combination or sequence of known phenomena. In these particular cases the natural choice of the modeling means is the coupling between existing software and programs. For the research work based on numerical modeling the huge public domain of informational resources, the open sources (including code ones) and open libraries (for one good example see [6]) offer a great potential for the coupling activity. On the contrary, if the entire model can be found then the problem has been already solved and certainly, closed. So it is not surprising that the coupling software development has already made its little history [5].

It seems evident, however, that the *Multiphysics* "spouse" should naturally come from a family of dedicated simulators.

One more positive reason for coupling methodology choice is that the time constant is different for applied research work so that, normally, the first pragmatic conclusions should be done urgently. Obviously, different type of the methodology for discretized models and of the grid used by simulators never facilitates the task of EMIR. Nevertheless, the advantage of coupling may come also from completely separate computations (i.e. separate models) for the equations of different type. For example, the electric potential (elliptic) equation is preferably to be computed with the FE method on unstructured grid due to near singular solution in the vicinity of the electrodes, and so do at least the pressure and even the temperature equations. Of course, care should be taken for this may cause some problems of stability and accuracy (cf. [7]) and accordingly some preliminary work has to be done.

In any case, a good contemporary practice would be to envisage the coupling possibility from the very beginning of a simulator development.

3. EMIR by COMSOL *Multiphysics*

As it has been stated above the recent development of the CM presented previously in [2] for low-frequency electric heating (LFH) process is considered here. Basically, in this particular case two distinct but interrelated physics are resolved: the thermal multiphase flow through porous media, and the electric current distribution. Two options were handled to solve the flow equations: using a COMSOL model based on the PDE application mode (FIM) or using commercial reservoir simulator STARS by CMG (CM).

3.1. Problem specification

The mathematical description of the multiphase flow problem solved by the COMSOL and the reservoir simulator is based on generalized Darcy's law and can be found elsewhere (*eg* [1]). On the other hand, the non-linear equations for electrical complex potential were modeled using the PDE application mode (cf. [2]). The coupling term between the two physics is the Joule (resistive) heating source; which is defined as

$$J = \sigma \left| \nabla \upsilon \right|^2 , \qquad (1)$$

where J is the heating power density [W/m^3]; σ is the bulk electric conductivity [S/m] and υ is the (complex) electric potential [V].

Thus the calculation of the bulk electrical conductivity σ is important point in this study since it explicitly couples the electrical model (COMSOL) and the thermal flow model (STARS) variables. The modified formulation of Archie's law which takes into account the

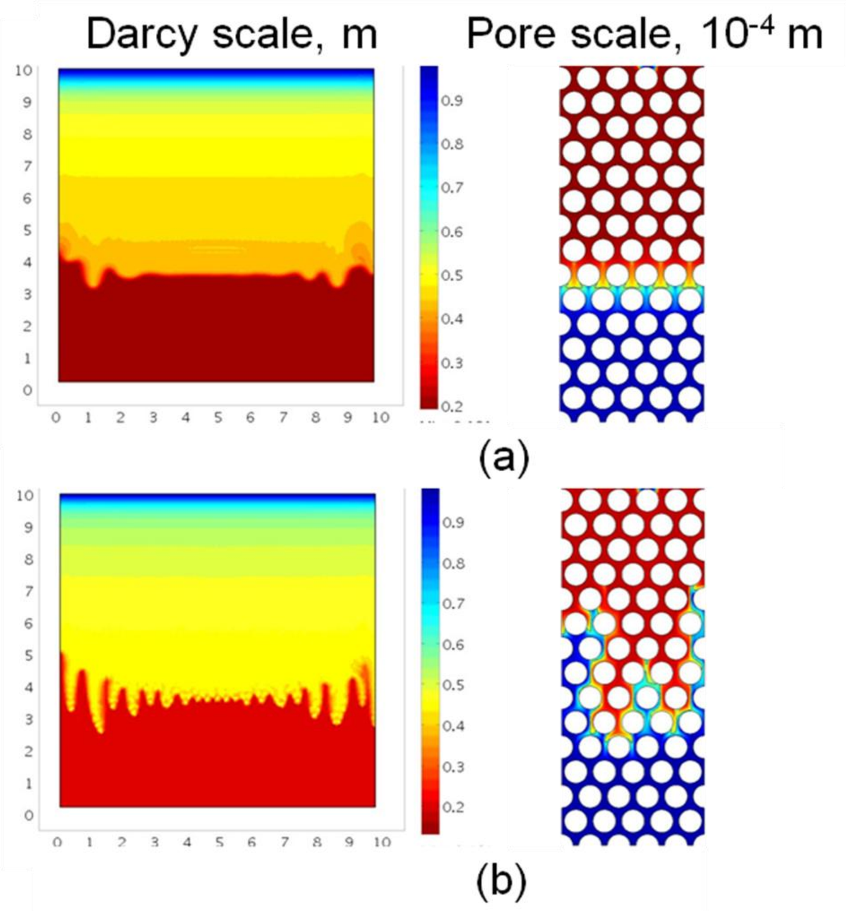

Figure 1. The FIMs application results: two-phase flow patterns at Darcy scale (a,b to left) and at pore scale (a,b to right; the diffuse interface computations on pore scale have been made by *S.Jardel*) at favorable for displacement front stability (a) and unfavorable (b) phase mobility ratio.

temperature dependency (factor *f(T)*) is used in current work:

$$\sigma = \sigma_w \varepsilon^m S_w^n f(T) . \qquad (2)$$

Here σ_W is pure water conductivity at reference temperature, ϵ medium porosity, S_W water local saturation, *m* and *n* (constant) parameters and the temperature dependency can be approximately presented by linear function which increases the conductivity by a factor of 3-4 for a temperature increase of 100°C [4].

3.2. The code main features

Technically, each CM run is governed via COMSOL *Script* facility and a special launcher has been written in *Phyton* which starts both simulators. Data exchange between COMSOL and STARS simulations is performed taking advantage of the COMSOL *Multiphysics*™ and COMSOL *Script* simulation environment facilities. For example, some external functions were written using COMSOL *Script* language in order to make compatible the format of data during the exchange between two simulators.

The sequential coupling used in EMIR is based on the loose coupling approach as described in [7]. Briefly, the loose coupling between the electrical heating and the thermal flow problems is performed explicitly after a certain number of time-steps done by STARS. The coupling time-step constitutes a fundamental numerical parameter which depends on the nature of tackled physical problem. The COMSOL *Script* provides two important features required for the coupling: the interpolation method to transform the exchange data for different grids, and the data verification using built-in mathematical operations (like the integration etc.) over the COMSOL mesh.

The EMIR code is written using programming language and functions of the COMSOL *Script*, the modular code structure managing not only input/output operations but allowing also fast adaptation of a COMSOL model geometry, grid generation, boundary conditions and its main physical parameters. Beside that the code controls the exchange data, including express-analysis of the numerical solution, and numerical tuning parameters of the simulators. Once made, it opens the way for coupling other dedicated simulators without any significant problems. So it should be emphasized that the successful EMIR application has been possible only due to making use of such a powerful mean like COMSOL *Script*. So thank you very much,

4. COMSOL *Script*

"Go back to Mississippi, go back to Alabama, go back to Georgia, go back to Louisiana, go back to the slums and ghettos of our northern cities, knowing that somehow this situation can and will be changed. Let us not wallow in the valley of despair.
I say to you today, my friends, that in spite of the difficulties and frustrations of the moment, I still have a dream." *Martin Luther King*

5. Model equations: the FIM of the thermal two-phase flow

The thermal flow model equations include the fluid component mass and the total thermal energy conservation equations (cf. [2]). The mass conservation equations are written for the water (index "*w*") and for the oil which is assumed to be uniform non-volatile hydrocarbon liquid (index "*h*"). Then the equations read as

$$\partial_t\left(\varepsilon\rho_w S_w\right)+\nabla\cdot\left(\rho_w u_w\right)=0 \qquad (3)$$

$$\partial_t\left(\varepsilon\rho_h S_h\right)+\nabla\cdot\left(\rho_h u_h\right)=0 \qquad (4)$$

where phase flows, u_p, $p=w,h$, are described by generalized Darcy' law

$$u_p=-K\eta_p\cdot\left(\nabla P_p+\rho g e_z\right) \qquad (5)$$

Here K is absolute permeability, ρ and η phase density and relative mobility, S phase saturation. As the temperature is not uniform in the reservoir, the total thermal energy conservation equation which includes solid (index "*s*") and fluid phases (index "*f*") contributions under assumption of local thermal equilibrium (one-temperature approach), complements the model

$$\partial_t\left(E_s+E_f\right)+\nabla\cdot\left(U_f-\lambda\nabla T\right)=J \qquad (6)$$

where E is volumetric internal energy, U_f total volumetric flow of thermal energy, λ reservoir (bulk) thermal conductivity coefficient, T temperature. The total flow U_f comprises fluid phase flows, $U_p= \rho_p h_p u_p$, where h is phase specific enthalpy.

Finally, pore volume conservation constraints phase saturations in usual manner

$$S_w+S_h=1\ . \qquad (7)$$

The FIM results of oil displacement by hot water around single injection well are presented in figure 2.

6. Model equations: the Joule resistive heating

In the case of complex electric potential, υ (see above), the electric charge conservation law may be written as a system of two stationary equations for real and imaginary potential parts:

$$\nabla\cdot\left(\sigma\nabla\varphi\right)=0 \qquad (8)\text{a}$$

$$\nabla\cdot\left(\sigma\nabla\psi\right)=0 \qquad (8)\text{b}$$

where

$$\upsilon=\varphi+i\psi\ .$$

The Joule heat release is defined by the equation (1) where

$$\left|\nabla\upsilon\right|^2=\left(\nabla\varphi\right)^2+\left(\nabla\psi\right)^2 \qquad (9)$$

and the bulk medium electric conductivity σ is temperature and water saturation dependent variable according to Archie's law (2) This means that the heating source J is solution dependent and there is a strong coupling between electrical and thermal flow phenomena. In particular, this non-linear coupling leads to the temperature rise acceleration during LFH.

The equations above either complement the FIM of the thermal flow (subsection 5) for the COMSOL computations of the LFH application or offer the COMSOL model for the EMIR computations.

7. Results and discussion

The results can be classified into two main groups. The first one includes those obtained for testing and validation purposes while the second one presents few particular applications related to field scale process modeling. Few test cases are presented first.

7.1. Code validation methodology

For the sake of simplicity the initial problem has been reduced to heat transfer with single phase stationary flow under conditions of the EH in a uniform medium. It can be shown that the dimensionless problem for 1D radial case is described by following equation

$$\theta_\tau + \frac{\Pi}{\rho}\nabla\theta - \Delta\theta = \frac{1}{\rho^2} \qquad (10)$$

which applies for dimensionless radial distance $\rho_0 \le \rho \le 1$. Here the dimensionless variables are introduced as follows: θ is temperature, τ time, Π the thermal Peclet number. Analytical solutions have been developed for the conduction and convection/conduction heat transfer in uniform or composite medium using stationary boundary conditions for applied potential, temperature and/or no conduction heat flow at the boundaries $\rho = \rho_{0,1}$.

Another approaches used for the code validation were:

(1) direct comparison to the FIM solution;
(2) juxtaposition of EMIR and STARS computation results;
(3) energy balance computations;
(4) analysis of grid global refinement and choice of FE influence.

In figure 3 the results of the preheating of the reservoir by Joule power during 1 year are shown. Comparison between the EMIR, the FIM and the analytical solution of (10) shows very good agreement. Similar problem of single electrode heating but for case of its partial penetration and with water circulation around it to prevent the connate water evaporation are illustrated in figure 4. The simulations have been done in 3D. The results of STARS to EMIR comparison demonstrate good agreement; note that STARS used radial structured grid that is perfectly adapted to the problem geometry unlike the non-structured tetrahedral COMSOL grid. Finally, the Joule power density field around the electrode is illustrated in figure 5.

7.2. Field applications

At present, the EMIR contributes mainly into estimation and validation of different aspects of the process under consideration and in part of its feasibility. In particular, as the results obtained with the CM in the 2D cases are encouraging enough then a 3D EMIR version is under development and few applications are prepared for the nearest future.

7. Conclusions

- The coupling numerical models based on *COMSOL Multiphysics*™ remain a valuable numerical tool for petroleum applications;
- Currently for coupling purposes between COMSOL and a reservoir simulator the *COMSOL Script*™ is the indispensible mean which can't be either avoided or substituted within the framework of COMSOL *Multiphysics*™;
- Along with coupling models the COMSOL based fully integrated models play important role in the coupling methodology.

8. References

1. I.I. Bogdanov, K. El Ganaoui, A.M. Kamp, COMSOL 2D Simulation of Heavy Oil Recovery Process by Steam Assisted Gravity Drainage, *European COMSOL Conference 2007, Grenoble, France* (2007)

2. I.I. Bogdanov, K. El Ganaoui, A.M. Kamp, Study of Electric Heating Application for Heavy Oil Recovery, *European COMSOL Conference 2008, Hannover, Germany* (2008)

3. T. Freeman, R. Chalaturnyk, I. Bogdanov , Fully Coupled Thermo-Hydro-Mechanical Modeling by COMSOL Multiphysics, with Applications in Reservoir Geomechanical Characterization, *COMSOL Conference 2008, Boston, USA* (2008)

4. B.C.W. McGee, F.E. Vermeulen, *Journal of Canadian Petroleum Technology*, **Vol. 46**, No.1, pp.1-7 (2007).

5. Redler, R., Valcke, S., and Ritzdorf, H.: OASIS4 – a coupling software for next generation earth system modelling, *Geosci. Model Dev. Discuss*., 2, 797-843 (2009).

6. W. Bangerth, R. Hartmann and G. Kanschat, deal.II — a general-purpose object-oriented finite element library, *ACM Transactions on Mathematical Software*, vol. 33, no. 4, article 24, 27pp., (2007)

7. J. Kim, H.A. Tchelepi and R. Juanes, Stability, Accuracy and Efficiency of Sequential Methods for Coupled Flow and Geomechanics, *SPE 119084*, 20pp. (2009)

9. Acknowledgements

TOTAL company is gratefully acknowledged for having sponsored our research work.

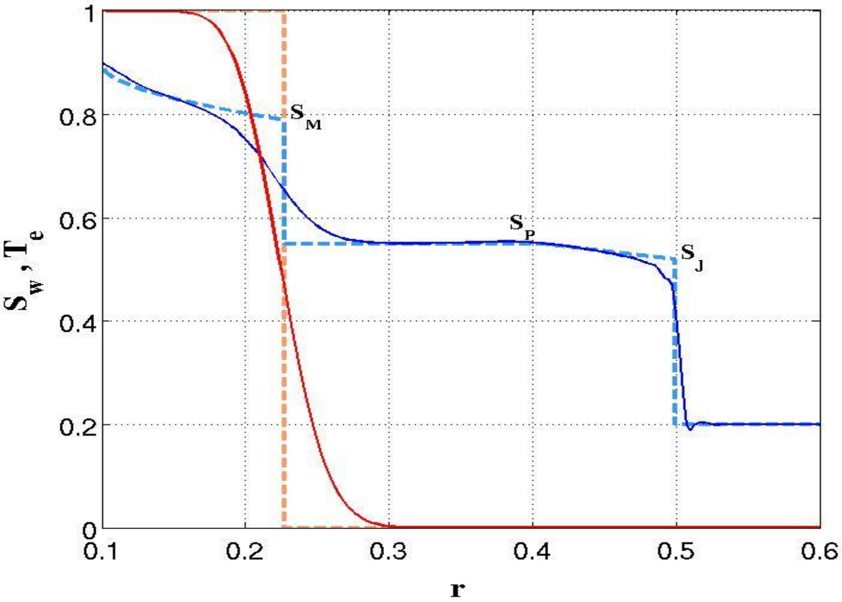

Figure 2. The thermal two-phase flow FIM results: the case of hot water injection. Comparison of exact (dashed lines) and numerical (solid lines) solutions in radial flow geometry. The temperature is normalized by injection water temperature value.

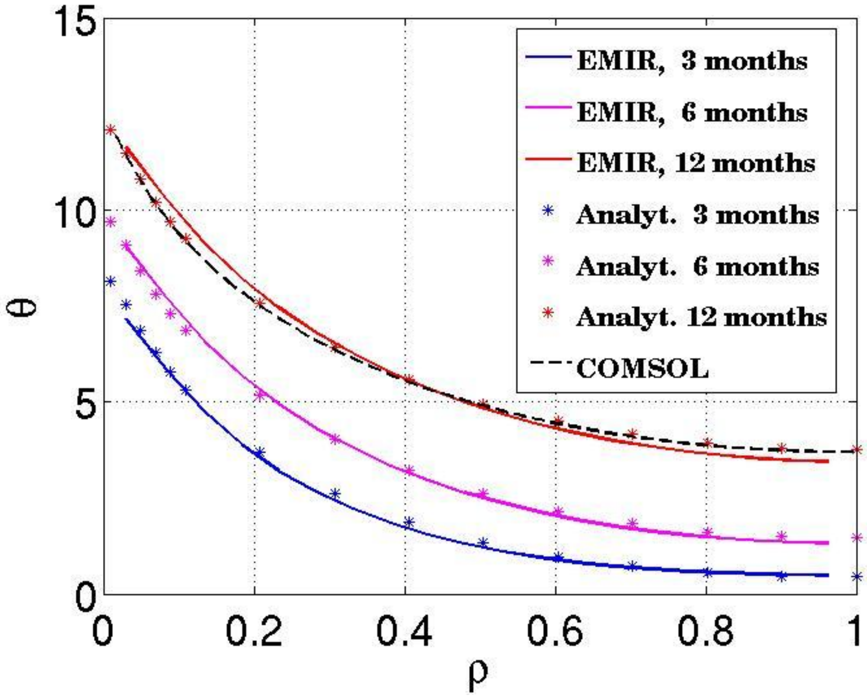

Figure 3. Preheating near single electrode with radial flow of the electric current: the dimensionless temperature profiles resulted from the EMIR computations (solid lines) are juxtaposed to exact solution (in asterisks) and the COMSOL FIM results (dashed line); computations were made at uniform bulk electric conductivity.

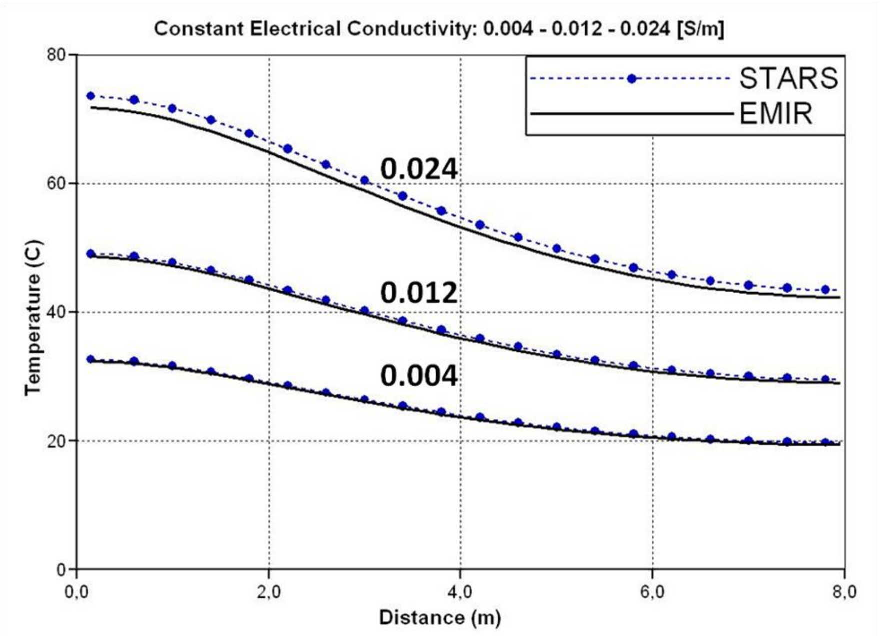

Figure 4. Preheating near single partially penetrated electrode with water circulation around it: the temperature profiles after 1 year at different bulk electric conductivity which is uniform (corresponding values are depicted over the curves) [S/m].

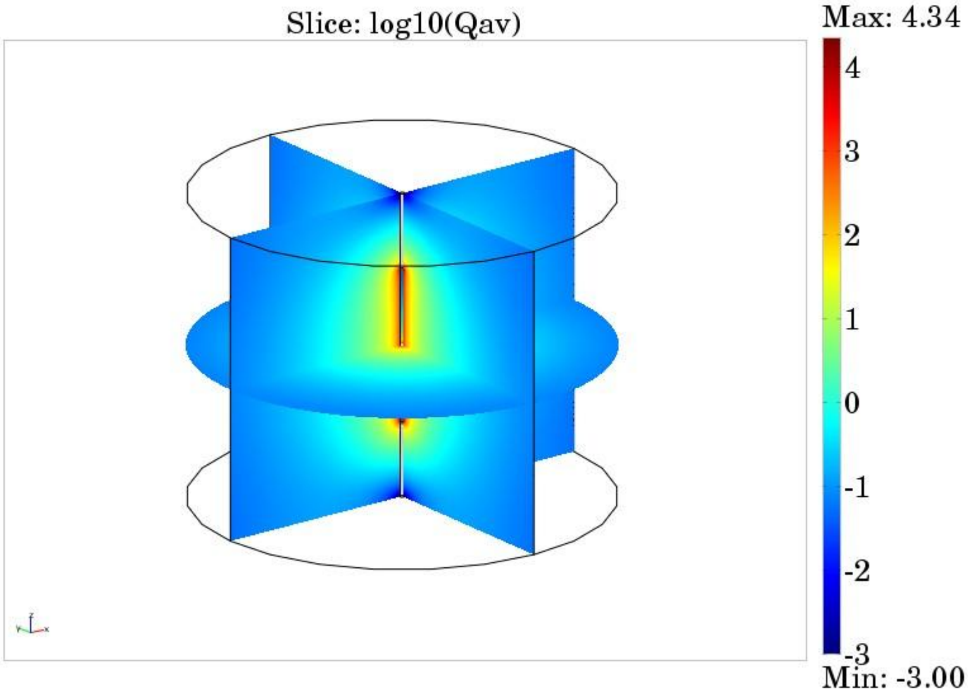

Figure 5. The Joule power density field around single partially penetrated electrode at σ=0.004 S/m.