Simulation of Multi-Physical Fields in a Fault Arc

Xiaohui Li[a], Can Bao and Huaren Wu

School of Electrical and Automation Engineering, Nanjing Normal University, 210042 Nanjing, China

[a]61011@njnu.edu.cn

Keywords: ANSYS Fluent, fault arc, short-circuit, multi-physical fields.

Abstract. A fault arc may destroy electrical equipment and threaten human life. Research on the multi-physical fields in a fault arc helps reduce this danger. This paper describes a theory for the multi-physical fields in the arc. These multi-physical fault arc fields were simulated using ANSYS Fluent. A case was then simulated. The simulation results are consistent with the theoretical analysis.

Introduction

The short-circuit current in power systems may be tens of thousands of amperes. The temperature of an arc at the fault location could be over 20000 K. The fault arc may destroy electrical equipment and threaten human life. Fault arcs involve electromagnetic fields, temperature fields and fluid flow. The multi-physical fields from the arc should be simulated to reduce the associated damage.

Reference [1] depicts fault arc voltages and introduces an arc fault model. Reference [2] used the business software ANSYS Fluent to simulate the multi-physical fields from a cutting arc. Reference [3] studied the arc motion along the arc runners and arc commutation during contact opening. Reference [4] built a 3-D simulation model for a miniature circuit breaker and calculated the multi-physical fields from an arc in a circuit breaker using Fluent.

The fault arc may move significantly under the action of an electromagnetic field. Simulating the multi-physical fields for a fault arc is difficult. No simulation method for a fault arc has been achieved before now. This paper introduces a simulation method for the multi-physical fields of a fault arc. The equations for mass conservation, momentum conservation, energy conservation and the electromagnetic fields of the arc are described. The geometry and mesh creation were interpreted. The UDF creation and simulation setup in ANSYS Fluent are depicted. The simulation results for a sample case are then shown.

Theory Describing the Multi-physical Fields for a Fault Arc

ANSYS Fluent is a computer program for modeling fluid flow, heat transfer, and chemical reactions with complex geometries. The fluid flow conserves mass, momentum and energy. The conservation equations for mass, momentum and energy are solved in ANSYS Fluent for a fluid flow.

The mass conservation equation can be written as follows [5-7]:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho v) = 0 \tag{1}$$

where ρ is the density, and v is the velocity.

The momentum conservation is described by

$$\frac{\partial}{\partial t}(\rho v) + \nabla \bullet (\rho v v) = -\nabla p + \rho g + F \tag{2}$$

where p is the pressure, and F is the external body forces.

The equation for energy conservation is given by

$$\frac{\partial}{\partial t}(\rho E) + \nabla \bullet (v(\rho E + p)) = -\nabla \bullet (\sum_j h_j J_j) + S_h \tag{3}$$

where S_h is the volumetric heat source.

Equations (1)-(3) are built into ANSYS Fluent. The user specifies the source terms for (2) and (3) to simulate the fluid flow.

ANSYS Fluent must calculate the electromagnetic fields from the multi-physical fields in a fault arc. The vector potential equation is described by

$$\nabla^2 A = -\mu j_s + \mu_0 \sigma \nabla V = -\mu_0 j \tag{4}$$

where A is the vector potential, j is the electric current density vector, and V is the scalar potential.

For 3D geometries, (4) can be written as follows:

$$0 = \nabla \bullet \nabla (A_x) + \mu_0 j_x - \mu_0 \sigma \frac{\partial V}{\partial x} \tag{5}$$

$$0 = \nabla \bullet \nabla (A_y) + \mu_0 j_y - \mu_0 \sigma \frac{\partial V}{\partial y} \tag{6}$$

$$0 = \nabla \bullet \nabla (A_z) + \mu_0 j_z - \mu_0 \sigma \frac{\partial V}{\partial z} \tag{7}$$

The current continuity equation is expressed as:

$$0 = \nabla \bullet (\sigma \nabla V) \tag{8}$$

ANSYS Fluent does not include (5)-(8). Users can setup user-defined scalar (UDS) transport equations in ANSYS Fluent according to the general scalar transport equations given by ANSYS Fluent.

The general scalar transport equation is shown below.

$$\frac{\partial \rho \phi_k}{\partial t} + \nabla \bullet (\rho u \phi_k) = \nabla \bullet (\Gamma_k \nabla \phi_k) + s_{\phi_k} \qquad k=1,2,\ldots,N \tag{9}$$

A UDS transport equation has four terms that the user can customize, unsteady, convection, diffusivity and source. (5)-(8) were compared to (9), and the parameters for the four terms in (9) were set to add (5)-(8) to ANSYS Fluent.

The source term in the fluid momentum equation is the Lorentz force given by:

$$F = j \times B \tag{10}$$

where the magnetic field is $B = \nabla \times A$ (11)

The source term, S_h, includes the Joule heating rate given by:

$$Q = \frac{j^2}{\sigma} \tag{12}$$

Simulating the Multi-physical Fields within the ANSYS Workbench

The ANSYS Workbench combines several ANSYS simulation tools. Users can work with the ANSYS Workbench to perform various structural, thermal, fluid, and electromagnetic analyses. Simulation procedures within the ANSYS Workbench are described as follows.

Launch ANSYS Workbench and then double-click Fluid Flow (Fluent) in the Toolbox on the left side of the Project tab. The fluid flow (Fluent) system is shown in Fig. 1 from the project schematic view, and a Fluent fluid flow analysis system is created using the ANSYS Workbench.

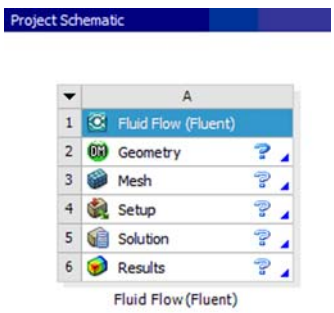

Fig. 1 Systems A

Create a Geometry Using the ANSYS DesignModeler. The ANSYS DesignModeler application is a geometry tool within the ANSYS Workbench. The ANSYS DesignModeler may be used as a geometry editor for existing CAD models. The ANSYS DesignModeler is a parametric feature-based solid modeler designed so the user can intuitively and quickly begin drawing 2D sketches and modeling 3D parts. The ANSYS DesignModeler features two basic operating modes: DesignModeler 2D sketching and DesignModeler 3D Modeling.

Double-click the second cell “Geometry” in system A shown in Fig. 1 to launch the ANSYS DesignModeler application in a separate window. The user can draw 2D sketches or model 3D parts. After a geometry is created, save it as myfile.agdb. Exit DesignModeler and return to the ANSYS Workbench window.

Create a Computational Mesh for the Geometry Using ANSYS Meshing. ANSYS ICEM CFD is a meshing tool within the ANSYS Workbench. ANSYS ICEM CFD can parametrically create meshes from a geometry and simplifies the mesh generation process.

Double-click the third cell, “Mesh”, in system A to launch the ANSYS Meshing application from the ANSYS Workbench. The file myfile.agdb is opened in the “Fluid Flow (Fluent) – Meshing [ANSYS ICEM CFD]” window. The user can generate meshes from a geometry in the ANSYS Meshing. Save the mesh as the file myfile.msh. Close the meshing tool.

Write the User-Defined Functions. User-defined functions (UDFs) allow the user to customize ANSYS Fluent. A UDF can enhance the standard code features. Users can use a UDF to define their own boundary conditions, material properties, and source terms for the flow; initialize a solution; or enhance post-processing. UDFs are written in the C programming language, and the source code is saved with a .c extension.

The source code file myudf.c was written to simulate the multi-physical fields for a fault arc. The file myudf.c contains the udf.h header file at its beginning using #include "udf.h".

The density, viscosity, thermal conductivity, specific heat capacity, enthalpy, sonic velocity and electrical conductivity of the air are temperature-dependent [8]. These values are saved in arrays or data files.

Users can use DEFINE_PROPERTY to specify a custom material property. That is, the predefined DEFINE macros are used to determine the density, viscosity, thermal conductivity, enthalpy, sonic velocity and electrical conductivity of the air.

Use DEFINE_SPECIFIC_HEAT to define the specific heat of the air.

Use DEFINE_SOURCE to calculate and specify the volumetric sources of momentum and energy. C_UDMI allows access to or stores the user-defined memory (UDM).

Use DEFINE_DIFFUSIVITY to specify the diffusivity for the UDS transport equations.

Use DEFINE_INIT to specify a set of initial values.

The P1 radiation model was implemented as a UDF, utilizing a user-defined scalar transport equation. An example is given in “8.2.5.2. Implementing ANSYS Fluent’s P-1 Radiation Model Using User-Defined Scalars” in [8].

Set up the multi-physical field simulation in ANSYS Fluent. Users may use the ANSYS Fluent fluid flow systems to set up and solve a 3D multi-physical field problem. The setup steps are described as follows:

Right-click the third cell “Mesh” in system A shown in Fig. 1 to display the applicable context menu. From the context menu, select “Update” to update the data. Right-click the third cell, “Mesh”, in system A to display the applicable context menu. From this context menu, select “Transfer Data To New” and “Fluent” so that system B appears as shown in Fig. 2 for the Project Schematic. Right-click the second cell, “Setup”, in system B and select “Edit” from the context menu to launch ANSYS Fluent. The file myfile.msh is opened in ANSYS Fluent.

Alternatively, the user may use the “File/Read/Mesh...” menu item of ANSYS Fluent to read the file myfile.msh.

Highlight “General” in the navigation pane by clicking it, and click the “Check” button from the “General” task page to check the file myfile.msh. Select “Pressure-Based”, “Absolute” and “Transient” for the solver from the “General” task page.

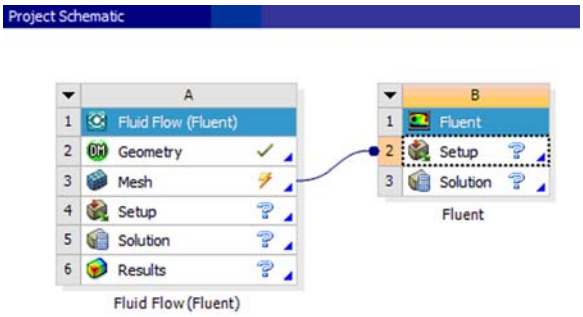

Fig. 2 Systems A and B

Enable the “Energy Equation” option in the “Energy” dialog box from the “Models” task page to calculate the heat transfer.

Select the “Define→User-Defined→Functions→Compiled…” menu item. Click the “Add…” button to add the myudf.c file and click the “Build” and “Load” buttons in the “Compiled UDFs” dialog box.

Select the “Define→User-Defined→Function Hooks…” menu item to open the “User-Defined Function Hooks” dialog box and add the “Initialization”, “Adjust” and “Wall Heat Flux” functions to the myudf.c file.

Select the “Define→User-Defined→Memory…” menu item and input the number of UDM locations into the UDM dialog box.

Select the “Define→User-Defined→Scalars…” menu item and input the number of UDS into the UDS dialog box. Select “Solution Zones”, “Flux Function” and “Unsteady Function”.

Select “air” and click the “Create/Edit…” button from the “Materials” task page to open the “Create/Edit Materials” dialog box. Select “Density”, “Specific Heat”, “Thermal Conductivity”, “Viscosity”, and “UDS Diffusivity” Functions from the myudf.c file.

Select a zone name and click the “Edit…” button from the “Boundary Conditions” task page; click the “Momentum”, “Thermal”, and “UDS” tabs to specify their respective boundary conditions.

Specify the reference values in the “Reference Values” task page.

Input parameters for the “Under-Relaxation Factors” from the “Solution Controls” task page. Click the “Limit…” button to change the solution limits.

Initialize from the “Solution Initialization” task page.

Input the “Time Step Size” and “Number of Time steps” parameters from the “Run Calculation” task page.

Input “solve”, “set” and “Expert” in the “console” window. Answer “yes” to “keep temporary solver memory from being freed”.

Click the “Calculate” button to start the simulation of the multi-physical fields.

Simulation Results

The 2D multi-physical fields for a fault arc were simulated according to the steps above. The geometry is shown in Fig. 3, where “A” is an electrode, and “B” is a metal plate in the air. There is an electric arc between A and B. The arc current is 10 kA. The temperature field is depicted in Fig. 3. Electrode A was not simulated. The highest temperature of the arc was over 20000 K. The pressure distribution for the fault arc is characterized in Fig. 4. The electric potential distribution for the fault arc is shown in Fig. 5. The electric potential is higher at plate B than electrode A.

Conclusions

The fault arc involves electromagnetic fields, temperature fields and fluid flow. ANSYS Fluent can simulate these coupled multi-physical fields. The built-in equations for mass, momentum and energy

conservations in ANSYS Fluent may be immediately used. Five UDS transport equations were added to simulate the 3D electromagnetic fields and P-1 Radiation Model. The simulated fault arc results are consistent with the theoretical analysis.

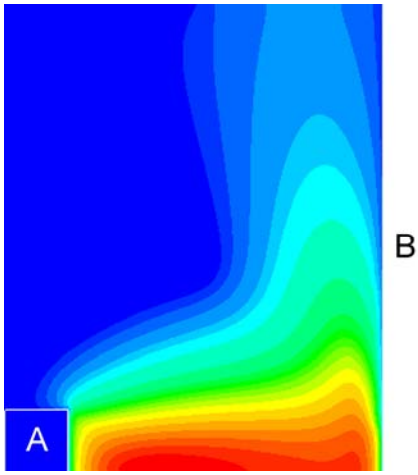

Fig. 3 The temperature field for a fault arc

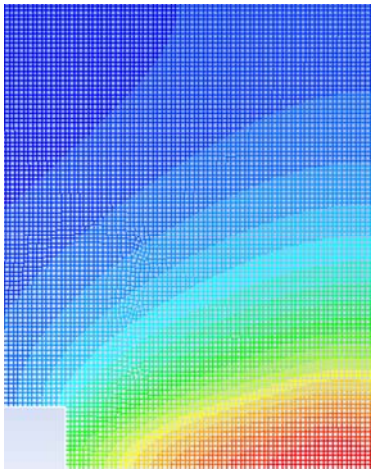

Fig. 4 The pressure distribution for a fault arc

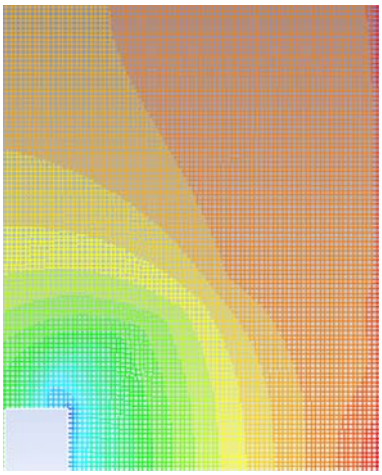

Fig. 5 The electric potential for a fault arc

Acknowledgements

This work was financially supported by the National Natural Science Foundation of China (51177074).

References

[1] H. Wu, X. Li, H. Schau and D. Stade: IEEE Trans. Power Delivery, Vol. 20, (2005) p. 1204-1205

[2] Q. Zhou, H. Yin, H. Li et al: J. Phys. D: Appl. Phys., Vol. 42, (2009) p. 95208-95216

[3] Y. Wu, M. Rhong, Z. Sun et al: J. Phys. D: Appl. Phys., Vol. 40, (2007) p. 795–802

[4] Y. Wu, M. Rong, F. Yang et al: IEEE Trans. Plasma Sci., Vol. 39, (2011) p. 2858-2859.

[5] ANSYS INC. ANSYS Fluent-User’s guide, (2015)

[6] P. A. Davidson: An Introduction to Magnetohydrodynamics: Part A, Cambridge Univ. Press (2001)

[7] P. M. Bellan: Fundamentals of Plasma Physics, Cambridge Univ. Press (2006)

[8] Y. Naghizadeh-Kashani,Y. Cressault, A Gleizes and Y Naghizadeh-Kashani: J. Phys. D: Appl. Phys. Vol35, (2002) p. 2925-2934