

Coupled CFD-Thermal Analysis

Jing Pang, Ph.D., Senior Engineering Specialist
General Dynamics Land System, Sterling Heights, MI 48301

ABSTRACT

The both CFD (Computational Fluid Dynamics) and thermal analyses are used to predict a vehicle system thermal performance during the design development. The vehicle wall temperatures and compartments temperatures under various climatic conditions are predicted in MuSES thermal analyses. The temperature and air flow distributions inside the vehicle compartment are predicted in Star CCM+ CFD analyses.

Recently, GDLS, Thermal Analytics, and CD-adapco jointly developed a CFD thermal analysis panel. This panel can be used to apply all boundary conditions to MuSES model and StarCCM+ CFD model by a few button clicks. It can map convection coefficients predicted in CFD analysis to the MuSES model boundaries; and vice versa, map wall temperatures and heat rates predicted in MuSES models to the boundaries in StarCCM+ models. Using this panel, the MuSES analysis and StarCCM+ analysis can be coupled to predict vehicle thermal performance with higher accuracy. Besides, most model inputs can be combined into an Excel data file and reviewed by numerous colleagues. The tedious model set up procedure is simplified. Human data entry errors can be avoided and the design updates can be easily accommodated to the analysis models.

This paper is to introduce the concept of this developed CFD-Thermal Analysis Panel, as well as the discussions about how the mapped convection coefficients and mapped temperature boundary conditions affect the analysis results.

1. Introduction

The both CFD (Computational Fluid Dynamics) and thermal analyses are used to predict a vehicle system thermal performance during the design development. The vehicle wall temperatures and compartments temperatures under various climatic conditions are predicted in MuSES thermal analyses. The temperature and air flow distributions inside the vehicle compartment are predicted in Star CCM+ CFD analyses.

MuSES is full-featured heat transfer analysis software. The solar radiations through clouds in sky are modeled in MuSES according to the geographic locations and dates. Usually constant heat convection coefficients are assigned to the wall boundaries when setting up a MuSES model. The convection coefficient represents the thermal resistance of a fluid boundary layer between a heat transfer surface and a fluid medium. In most analysis situations, it is a fluid flow dependent variable.

StarCCM+ is a CFD analysis software. It is used to predict air flow and temperature distributions with assigned boundary conditions, such as wall temperatures, wall heat flux, and air flow conditions. There are no "readily-applicable" solar-sky radiation models in Star CCM+. The wall temperatures and heat fluxes associate to the solar radiations should be estimated using other methods for boundary conditions.

Recently, GDLS, Thermal Analytics, and CD-adapco jointly developed a CFD thermal analysis panel. This panel can be used to apply all boundary conditions to MuSES model and StarCCM+ CFD model by a few button clicks. It can map convection coefficients predicted in CFD analysis to the MuSES model boundaries; and vice versa, map wall temperatures and heat rates predicted in MuSES models to the boundaries in StarCCM+ models. Using this panel, the MuSES analysis and StarCCM+ analysis can be coupled to predict vehicle thermal performance with higher accuracy. Besides, most model inputs can be combined into an Excel data file and reviewed by numerous colleagues. The tedious model set up procedure is simplified. Human data entry errors can be avoided and the design updates can be easily accommodated to the analysis models.

This paper is to introduce the concept of this developed CFD-Thermal Analysis Panel, as well as the discussions about how the mapped convection coefficients and mapped temperature boundary conditions affect the analysis results.

2. Vehicle Thermal Performance Analysis Processes

The vehicle thermal performance analyses produce the predictions of the vehicle compartment temperatures, temperature distributions, and the vehicle wall temperatures for specified weather conditions such as weathers defined in AR 70-38. In general, the vehicle geometry design, the armor construction, the cooling and heating subsystem design specifications, vehicle operation conditions, vehicle thermal architecture heat load list, as well as the weather conditions are the inputs for the analyses. The vehicle thermal performance is the output from analyses. Figure 1 shows a typical vehicle thermal performance analysis process.

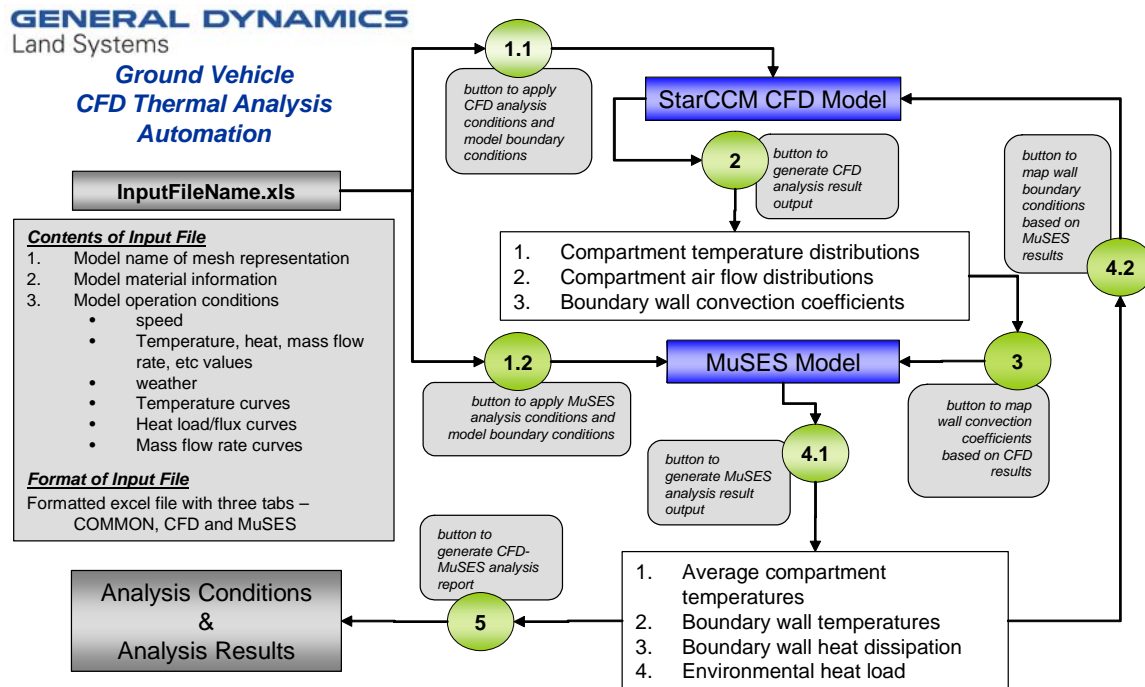


Figure 1. Vehicle Thermal Performance Analysis Process

The buttons in Figure 1 represent the function buttons developed in our CFD-Thermal Analysis Panel. All the input information are composed in a excel file. When the Panel buttons clicked, the conditions are applied. This significantly reduces the model set up time. In addition, all the model conditions are documented in a well organized manner. Any design update or condition changes can be applied and traced easily.

Figure 2 is the representation of this CFD Thermal Analysis Panel with all function buttons. The functions of each button are following.

Initialize – create data files for vehicle material information, and vehicle boundary conditions from DATA.xls (excel file) which will be used to set up both Star CCM+ and MuSES models.

Set up STAR-CCM+ Model - transfer boundary conditions from data files (generated by **Initialize**) to the StarCCM+ model.

Set up MuSES Model - transfer vehicle structured material information and vehicle boundary conditions from data files (generated by **Initialize**) to the MuSES model.

Run MuSES Model - run MuSES model in a batch mode

Run STAR-CCM+ Model – run STAR-CCM+ model

Generate Convection Coefficient Map Data – Generate convection coefficient map data from STAR-CCM+ model results for later applying boundary conditions in MuSES model.

Map Convection Coeff on MuSES Model – Map the CFD predicted convection coefficients to the MuSES model corresponding boundaries

Generate Temperature Map Data – create a data file with wall temperatures predicted in MuSES model which will be used for CFD model boundary conditions

Map Temperature on STAR-CCM+ Model – Map the wall temperatures using MuSES resulted data

Post Processing – Post processing STAR-CCM+ results

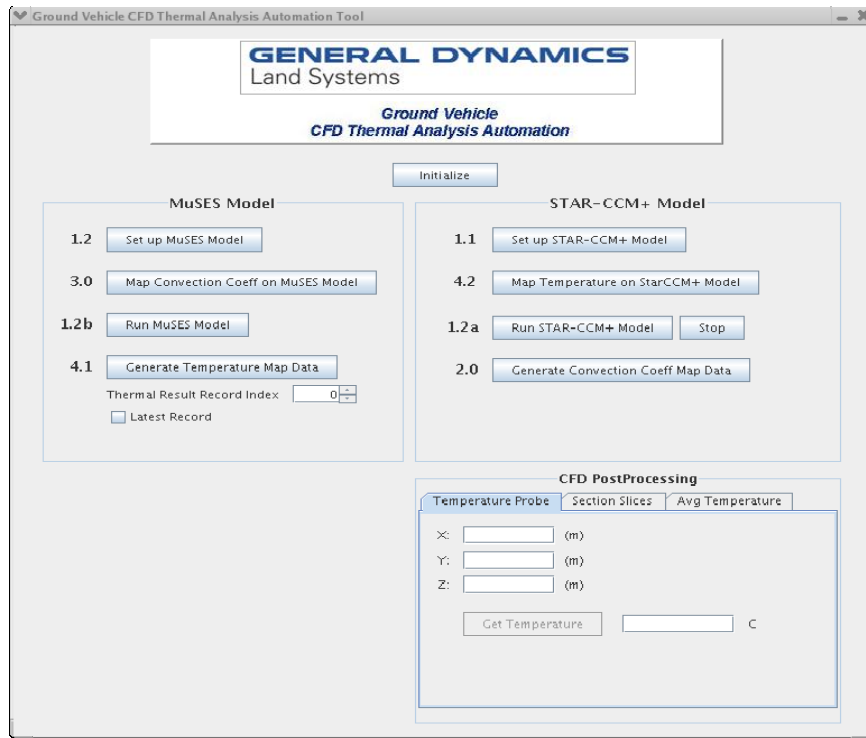


Figure 2. The CFD Thermal Analysis Panel

3. External Wall Temperature Effect of Convection Coefficient Mapping

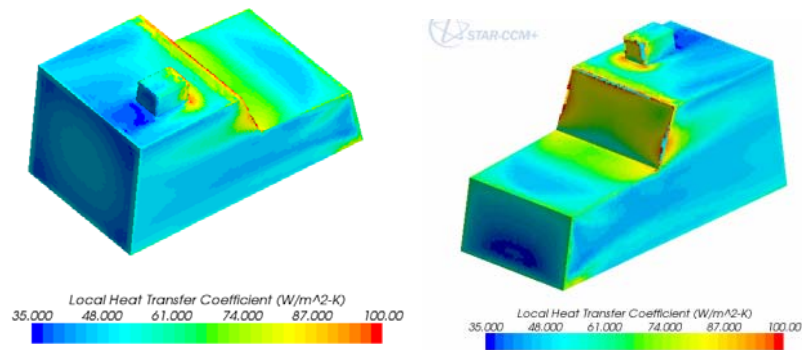


Figure 3. Vehicle External Wall Convection Coefficients

Figure 3 shows the external wall convection coefficients of a vehicle moving 20 MPH, which are the STAR-CCM+ model results. Using CFD-Thermal-Analysis Panel, the data are mapped to the MuSES model. Figure 4 shows the wall temperature comparisons between the models with mapping and without mapping.

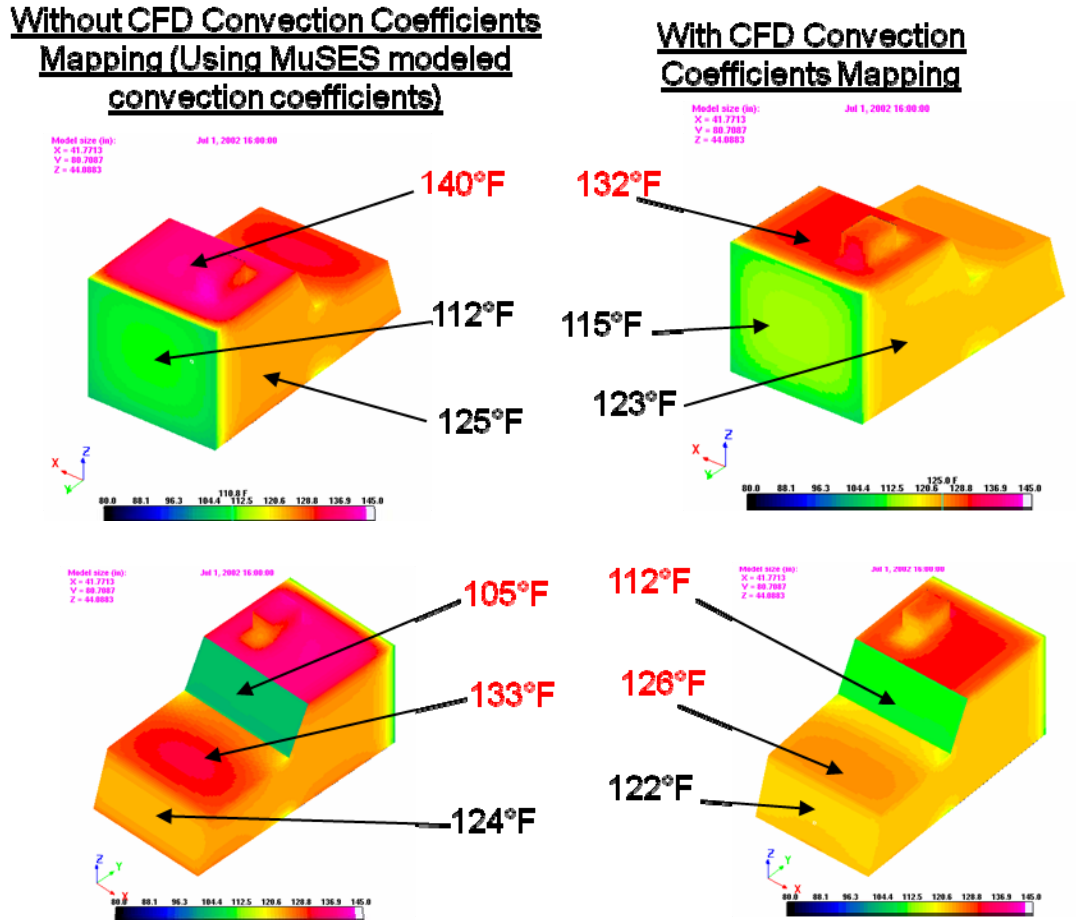


Figure 4. Vehicle External Wall Temperature Comparisons.

In the case of “without CFD mapping”, the convection coefficients are estimated using MuSES default turbulence model (“Wind” setting). The temperature numbers shown are area averaged results per specific boundary. The significant temperature differences (marked as RED) are observed from the places where the strong flow separations happening.

4. Internal Wall Temperature Effect of Convection Coefficient Mapping

A simple compartment is created for this study. The STAR-CCM+ model including an air flow unit with a mass flow and 59°F fixed temperature inlet and a pressure outlet boundary conditions and a few electronic units with assumed heat dissipations. To ensure the accuracy, three prism layers are created and adjusted thickness until the y^+ in the range of 30 to 100. The CFD predicted convection coefficients are shown in Figure 5.

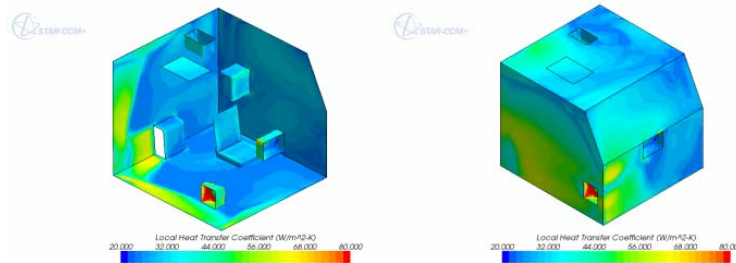


Figure 5. A Compartment Internal Wall Convection Coefficients

There are three ways to map the CFD predicted convection coefficients to the MuSES model – 1, mapping; 2, calculating the boundary area averaged convection coefficients, then applying to the corresponding boundaries in MuSES model; 3, calculating all

boundary area averaged convection coefficient and then applying to the MuSES model boundaries. These three methods are used and the MuSES results are shown in Figure 6.

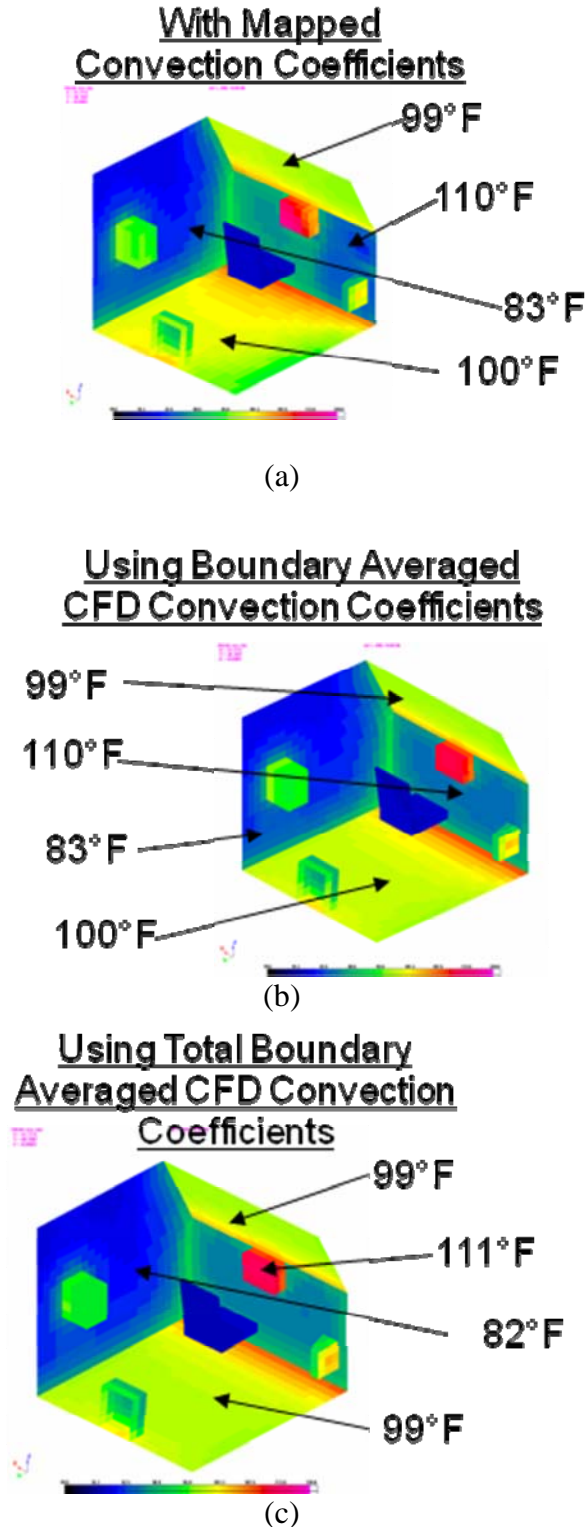


Figure 6. MuSES Model Wall Temperature Comparisons with Different CFD Convection Coefficient Mapping Methods

The boundary averaged wall temperatures don't have any significant difference between the three CFD convection coefficient mapping methods. However, there are certain variations in temperature distributions. With these three mapping methods, the average compartment temperature are resulted the same value, 75°F.

5. Conclusions

The CFD Thermal analysis automation panel is efficient in organization and documentation the model conditions. This panel reduces model construction time limit human errors during the model construction.

There are significant MuSES wall temperature differences between using CFD convection coefficient map and using MuSES default “Wind” model. There is no significant effect in compartment internal wall temperatures and compartment air temperatures when using averaged CFD convection coefficients compared to using high fidelity convection coefficient mapping.