2023 NDIA MICHIGAN CHAPTER
GROUND VEHICLE SYSTEMS ENGINEERING
AND TECHNOLOGY SYMPOSIUM
MODELING SIMULATION AND SOFTWARE TECHNICAL SESSION
AUGUST 15-17, 2023 - NOVI, MICHIGAN

AUGMENTING THERMAL AND SIGNATURE MODELS USING A FAST 3D FLUID DYNAMICS SIMULATION FOR IMPROVED CONVECTION FIDELITY

Joshua Pryor[1], Duncan Karnitz[1], Warren Powers[1], Douglas Banyai[1], Pete Rynes[1], Nathan Tison[2], Vamshi Korivi[2], Yeefeng Ruan[2]

[1]ThermoAnalytics Inc., Calumet, MI and Novi, MI
[2]US Army DEVCOM Ground Vehicle Systems Center, Warren, MI

ABSTRACT

Accurate thermal simulations for the purpose of thermal or infrared signature management require accurate representation of all modes of heat transfer. For scenarios with complex fluid dynamics and convective heat transfer, traditional options have included very simple 0D methods or very computationally expensive 3D CFD simulations. Motivated by adding options between these extremes and tuning the method to a heat transfer focus, a 3D fluid dynamics solver is developed that is tightly integrated and automatically coupled with the MuSES thermal and EO/IR simulation software. Key applications of interest include wind flow around ground vehicles for the purpose of infrared signature management and HVAC air flow within cabins for the purpose of thermal management. The flow solver uses novel numerical techniques to simplify the standard Navier-Stokes equations and avoid calculations which may not be necessary for thermal simulations. Several domain meshing strategies, physics models, numerical approaches, and test models are developed with the goal of ease of use and minimal additional time investment for a thermal simulation.

Citation: J. Pryor, D. Karnitz, W. Powers, D. Banyai, P. Rynes, N. Tison, V. Korivi, Y. Ruan, "Augmenting Thermal And Signature Models Using A Fast 3D Fluid Dynamics Simulation For Improved Convection Fidelity," In *Proceedings of the Ground Vehicle Systems Engineering and Technology Symposium* (GVSETS), NDIA, Novi, MI, Aug. 15-17, 2023.

1. INTRODUCTION

The efficient design and evaluation of Army vehicles requires accurate simulation of the factors that affect vehicle signature and thermal conditions. For design efforts, the simulations must be rapid, and easy to set up and modify. The models must account for all the transient physics that determine heat transfer including engine dynamics, engine exhaust flow, air flow within engine compartment, ventilation inside the cabin, and wind flow over the vehicle exterior. The simulation tool must also provide all the relevant design information including the effects of the thermal environment on conditions within the vehicle, transfer of heat between vehicle components, and the EO-IR (Electro-optical-infrared) signature of the vehicle.

To model convection, fast thermal simulation tools or calculations often rely on a library of simple convection relations. Such modeling fails to capture important details of the flow around vehicles or other targets, especially for targets in complex scenes. Modeling wind wakes, flow channeling, acceleration around obstacles, and the advection of heat to downstream objects is necessary for accurate temperature predictions. Thermal simulation tools can also be coupled with commercial CFD (Computational Fluid Dynamics) simulations to compute convection, but CFD is computationally costly and requires expertise in CFD modeling. Our study of convection modeling options revealed that there was no available solution that was compatible with the large spatial domains and long transient times associated with the electro-optical infrared (EO/IR) problems that are necessary for the Army to solve. To address this technology gap, a thermal-specific fluid dynamic convection solver is developed that can model these major flow features without the burdens associated with traditional CFD. This new approach to convective heat transfer calculations is being integrated into ThermoAnalytics' MuSES and TAITherm simulation software, which can provide precise modeling of thermal and radiative environments. The flow solver technology is commercialized under the name RapidFlow. This integration results in a more accurate analysis of remote sensing imagery.

2. BACKGROUND, APPLICATIONS, AND SUMMARY OF SOLVER APPROACH

RapidFlow was developed with the support of USG SBIR awards and significant IRAD investment. In 2015, ThermoAnalytics received Phase I and II SBIRs from the Department of Energy (DOE) National Nuclear Safety Administration (NNSA). This project supported the early development of RapidFlow to simulate landscape-size domains in the EO/IR. In 2019, the US Army GVSC awarded TAI an additional Phase I and II SBIR contract to refine RapidFlow's application as a single-solver solution for EO/IR simulation of ground vehicle platforms.

2.1. Army application

The RapidFlow solver capability -- developed over the course of the associated Army SBIR effort -- has the promise of providing significant benefit in regards to the flow, thermal, and infrared signature analysis of Army ground systems (and other DoD materiel). Characteristics of internal and external flow fields significantly impact the thermal and infrared (TI) signatures of the materiel. Therefore, it is important to model the flow within the context of the TI simulations, such that the flow's thermal effects are sufficiently captured; however, it is not necessary, or time-efficient, to fully resolve all details associated with the flow field, especially in flow regions not near materiel surfaces. Recent progress made toward achieving this modeling capability and integrating it into MuSES' thermal / infrared solver has contributed to the evolution of a time- and resource-efficient means for performing TI simulations.

The impetus application for the Army SBIR effort was vehicle infrared signature analysis. Such analysis generally needs to be performed in classified computing venues which require standalone software licensing for the CFD solvers, involving additional expense. Also, such analysis involves long timeframes (multiple days), and the typical co-simulation between a traditional CFD solver and MuSES would often be resource-prohibitive, given the large number of simulations which typically need to be performed to more fully characterize vehicle infrared signature. Therefore, having a time-efficient, sufficiently-accurate, combined flow / thermal modeling capability in a single solver will be very beneficial for infrared signature M&S problems.

Another application area involves vehicle heating, ventilation, and cooling (HVAC) system assessments. Though the required problem timeframes tend to be shorter than those for infrared signature (involving time durations of hours rather than days), having combined flow / thermal modeling capability will be beneficial for early design / performance assessments performed prior to more rigorous, detailed-design, CFD-based flow / thermal analyses. Assessments of vehicle and HVAC system "pull-down" performance (associated with providing cooling in hot environments) and "pull-up" performance (associated with providing heating in cold environments) could be made. Early insights into suitable HVAC flow return and supply locations and required heating / cooling capacities could be quickly provided. A related application area involves human thermal effectiveness (using a human physiology and comfort model, such as that built into MuSES) assessments in the context of severe thermal environments, leveraging already-existing capabilities. In general, this M&S capability can be used to assess compliance associated with other MIL-STD-810 requirements, such as limits regarding the spatial variation of air temperature near warfighters.

Flow and convection capabilities also augment touch temperature prediction, and thermally-sensitive materials simulation near heat sources where the nearby flows need to be adequately modeled in order to sufficiently predict the component temperature fields and assess the effectiveness of mitigation solutions, such as heat shields, air movers, etc.

As the flow solver capability continues to expand, vehicle underhood cooling (which involves complicated flow fields due to the effects of cooling fans and vehicle motion) will be an application area for early design / performance assessment. Simulations of thermal exhaust plume impingement and EO/IR plume radiance will be integrated into MuSES via RapidFlow. Toxic fume concentration analyses could be performed for weapon firing, battery thermal runaway, and off-gassing associated with chemically contaminated components. Additionally, a total capability to model electric and hybrid-electric vehicles, which have unique thermal and EO/IR modeling challenges, will be enabled in MuSES with RapidFlow and a new Program of Record.

Other application areas could invariably be cited. However, initially it is believed that the strength of MuSES' combined flow / thermal modeling capability lies primarily in its ability to efficiently simulate the phenomenology associated with the afore mentioned applications with reasonable accuracy early in the design or performance evaluation process, prior to the availability of detailed information and when it is necessary to quickly explore the range of the design / analysis space with reasonable accuracy.

2.2. Solver approach

RapidFlow places its computational emphasis on the details that matter most to convection and IR signature prediction, especially EO/IR prediction accuracy and computation speed. RapidFlow solves many of the same Navier-Stokes equations that typical CFD software solves for, albeit with different assumptions and simplifications.

All CFD codes use assumptions, mainly in the form of empirical correlations. RapidFlow uses correlations that directly calculate convection coefficients based on local bulk flow conditions of temperature, velocity, and pressure. These correlations were tailored specifically for RapidFlow; they produce accurate values for convection when applied to the type of mesh that RapidFlow uses. Conventional CFD, in contrast, uses empirical correlations such as the “law of the wall” to compute the momentum gradient through the boundary layer, from which, using Reynolds’ analogy, it can then compute a temperature gradient through the boundary layer, from which in turn it can determine the heat rate of convection.

The use of these convection correlations, rather than boundary layer momentum and temperature gradients, to compute convection allows RapidFlow to use a coarser mesh than what CFD needs to model flow within the boundary layer. RapidFlow needs only local bulk flow conditions - thus it does not need a fine mesh to compute conditions near or in the boundary layer. The fine flow details that RapidFlow ignores are rarely important drivers in the thermal behavior of objects at the scale viewed by EO-IR sensors.

Needing to compute only bulk flow on a coarse grid allows RapidFlow to simplify the Navier-Stokes to the level of time-split Euler equations. This simplification of the flow equations allows RapidFlow to be fast and robust and to require very minimal user effort and expertise compared to CFD. This equation simplification, coupled with the "invisible-to-the-user" integration of RapidFlow into MuSES, allows for the complete automation of flow modeling for heat transfer and EO-IR signature prediction within MuSES. MuSES with RapidFlow is a single solver that can compute thermal performance and fluid flow with little more user effort than is required now to run a MuSES thermal simulation without fluid flow prediction.

The simplicity of the RapidFlow grid allows MuSES to automatically generate the fluid mesh without significant user effort. If the geometry for the thermal solver is altered, such as to represent the relocation of vehicles or personnel in a scene, MuSES will automatically generate a revised fluid mesh to model the fluid flow given the new geometry.

RapidFlow uses a staggered grid in which scalar quantities (pressure, gas species, and humidity) are calculated at cell centers while vector quantities (velocity components) are stored at the centers of cell faces or at vertices. The staggered grid simplifies the solution for the Poisson equation for pressure, and allows the divergence of the velocity field to be represented with second-order accuracy.

RapidFlow uses a fractional step method to compute advancements in the flow field via an advancement of the pure advection equation. Additional models for turbulent mixing or buoyancy may also be included in this prediction. The predicted velocity field is then corrected by a correction method that computes a pseudo-pressure and uses the pressure gradients to compute the correction factors. This results in an incompressible flow field. This method is inherently transient and explicit in nature; depending on the method by which advection is modelled and whether additional closure models are included in the prediction. RapidFlow currently uses an un-conditionally stable Semi-Lagrangian method to model the advection; combined with an implicit integration of turbulent viscosity to provide closure to the RANS equations if configured. This method is stable for any size time step and accurate for long durations due to usage of a 3rd order Runge-Kutta integration scheme.

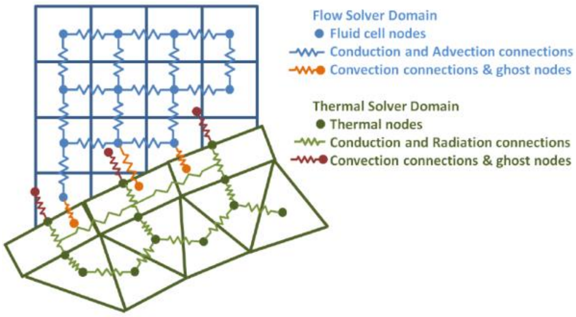

Figure 1: Domains of the flow and thermal solvers.

RapidFlow is then coupled directly to the thermal solver via the calculation of convection conductors between the bounding fluid region and the nearby surfaces [Figure 1]. In steady state this conjugate problem is fully coupled with iterations that include additional fluid motion if the buoyancy model is enabled. In transient coupling this method is reduced to just time-step coupling in the interest of long transient performance. Additionally in transient coupling the flow solver can operate in a pseudo-transient mode where the computation will terminate at a specified tolerance if a steady state is reached for the given time step conditions. The result is a fully complete conjugate heat transfer simulation between the fluid volume domain and all interacting surfaces.

3. KEY SOLVER CAPABILITIES

3.1. Modeling of turbulent mixing

The initial RapidFlow solver only considered inviscid flows. Thus the question remains: how the inviscid assumptions interact with turbulent simulations. By definition the Reynolds number scales inversely with the viscosity of the fluid; thus if we are to assume that the viscosity is zero we would numerically be assuming that the Reynolds number is infinite. In practice however we see turbulent behavior in for example the simulation of flow over a cylinder. The explanation for this behavior is the numerical diffusion that exists due to the spatial discretization of the fluid. In first order numerical schemes it can be shown that the numerical diffusion scales linearly with the cell spacing. This means that for our target of modeling fluids with generally coarse meshes we need to consider both reducing this source of numerical error as well as potentially augmenting the turbulent behavior with additional models.

The issue of attempting to model sub-grid turbulence below the resolution of the simulation grid can fall into various categories. The first of which is similar to Large-Eddy simulation approaches where a sub-grid scale model of the isotropic turbulence is introduced. We will not be delving into that work here; but it is potentially of great interest for future endeavors. An alternative method attempts to borrow from the world of fluid animation where lively turbulent flows are of interest for visual appearance. The most straightforward of these methods is the injection of vorticity confinement. In this method the existing curl forces in the fluid are computed on the grid, and amplified by a small scale factor in order to induce additional rotational kinetic energy to offset the losses due to numerical diffusion. Though this method does result in more lively and turbulent flows it requires careful tuning of the scaling factor to prevent the flow from blowing up with too much energy. Thus this method is not used in practice and we will forgo future investigation into these methods until their need can be identified and demonstrated.

A third method, which we now implement in the RapidFlow solver, is turbulence modeling via Reynolds Averaged Navier-Stokes (RANS) equations [1]. The explicit simulation of fluids generally must occur at time scales that are significantly shorter than the time scales of conduction and radiation. For this reason we are primarily interested in the time averaged behavior of the fluid.

The RANS equations are by definition steady state equations. However as a consequence of the averaging method they contain a set of terms that cannot be directly computed by a time averaged simulation. Thus additional equations must be introduced to model these additional terms and provide a closed system of equations to be solved. The first introduced equation is the Boussinesq approximation which postulates a linear relationship between the magnitude of the strain rate tensor of the time averaged flow and the to-be-modeled Reynolds stresses. From this approximation is introduced the turbulent viscosity as a spatially varying viscosity. Numerous additional equations are possibly introduced to model the newly defined turbulent viscosity and turbulent kinetic energy. For our initial model we will consider the simple zero equation mixing length model for turbulent viscosity which does not include a model of the turbulent kinetic energy.

$$\vec{\boldsymbol{R}} = \boldsymbol{v_t}\left(\frac{\partial \vec{\boldsymbol{U}}_i}{\partial x_j} + \frac{\partial \vec{\boldsymbol{U}}_j}{\partial x_i}\right) \tag{1}$$

$$\boldsymbol{v_t} = \boldsymbol{\rho} \boldsymbol{L}^2 \sqrt{\frac{1}{2}\boldsymbol{\Sigma}\left(\frac{\partial \boldsymbol{U}_i}{\partial x_j} + \frac{\partial \boldsymbol{U}_j}{\partial x_i}\right)^2} \tag{2}$$

The zero equation mixing length model defines the turbulent viscosity as a function of the local gradients in the time averaged values. Thus we incorporate a calculation of the turbulent viscosity throughout the fluid and use that in the solution of a spatially varying diffusion equation for all three components of velocity. The solution of these equations is done using a basic Jacobi smoother to quickly approximate an unconditionally stable diffusion of the momentum to model the additional turbulent mixing.

3.2. Modeling of buoyant-driven flow

For modeling thermally induced flow we look to determine efficient ways to approximate the natural forces from changes in fluid density at different temperatures. A thorough modelling of this effect would require a robust model of fluid density as a function of temperature and handling of the physics of the density based flows. Numerical methods for such compressible or pseudo-compressible models exist and are of interest for future endeavors. For the simplest implementation, we instead turn to the Boussinesq approximation [2] where density is only treated as a varying property in the modeling of the gravitational effects on density.

In this model a reference temperature is chosen. From this a reference density and the material’s thermal expansion coefficient are determined. The gravitational term is formed using a density defined from a first order Taylor series approximation of the temperature dependent density curve centered at the reference temperature. The resulting additional forces are integrated into the fluid simulation at each time step explicitly.

3.3. Accurate mapping between thermal geometry and fluid domain

Our initial mapping algorithm from surface facets to fluid cells was designed primarily to provide lookup of fluid temperature close to the facet centroid. For these reasons a rudimentary ray casting algorithm was implemented as a way to search for the nearest fluid cell from the facet centroid. This algorithm is insufficient in several contexts. One, when the resolution of the faceted surface contains much larger surfaces than the nearby fluid cell size this algorithm does not appropriately sample the fluid temperature variation across the surface facet. Two, under the same scenario we would like to use the mapping algorithm to map surface defined boundary conditions such as inlet and outlet planes to the fluid domain. In the described scenario the mapping can only map the large facet to a single fluid cell; resulting in only a single fluid cell being marked as an inlet boundary when the true boundary may have a much larger cross-sectional area. Finally, as alluded to by the limitation of mapping one facet to one fluid cell situations can arise when convective heat exchange must occur over a large swatch of facets and fluid cells each facet can only exchange data with a single fluid cell resulting in a non-smooth convective exchange.

In the interest of resolving all of these limitations we develop a new mapping algorithm that leverages intersection testing data generated by the automated meshing algorithm. With this information we can assume that any facet directly maps to any intersecting fluid cells and that through this mechanism we can efficiently map a single facet to multiple fluid cells.

In the previously described contexts this method resolves most issues. In the context of mapping inlet and outlet faceted boundaries to the fluid domain we simply define all fluid cells intersecting the inlet or outlet faceted region as inlet or outlet boundary cells respectively. This provides a spatially contiguous representation of the boundary over which the respective boundary conditions for momentum and continuity can be enforced. In the context of coupling the conjugate heat transfer problem between the volume-discretized fluid domain and faceted surface this method gives us the additional spatial overlapping data needed to more smoothly couple the heat transfer. We choose initially a rudimentary algorithm in which convection based heat fluxes are evenly divided between all intersecting fluid cells for a given facet. Those fluxes are integrated through the solution of the fluid thermal equations to provide a smooth spatial heating or cooling of the fluid. In the reverse coupling, the many fluid cells for each surface provide a spatially averaged convection correlation and fluid temperature as boundary conditions in the MuSES thermal solve.

4. SIMULATION RESULTS

4.1. Ground vehicle wind-flow test case

We have created a test case for the flow solver technology using a ground vehicle in wind scenario based on a standard MuSES model of a BMP-2 tank. This test case is meant to demonstrate the value of our RapidFlow approach for long diurnal wind simulations which are common for EO/IR analysis of ground vehicles. These diurnal simulations can often involve frequent changes in wind conditions which combined with the long simulated time can make coupling with conventional CFD impractical. However, simple 0-D wind models cannot predict key fluid dynamics effects of impingement, advection of heat, and variation of convection due to wind shadowing and wake regions. Since these two conventional approaches are both problematic in terms of practicality or fidelity, this case clearly demonstrates the benefit of our integrated RapidFlow solver approach.

The demonstration model is created from an existing BMP-2 model built for EO/IR analysis including engine heat sources. The weather scenario is defined using measured weather data taking during a field test (summer 2017 in Upper Michigan). The wind speed and direction over the simulated day are shown in Figure 2. The geometry of the thermal model is shown in Figure 3:

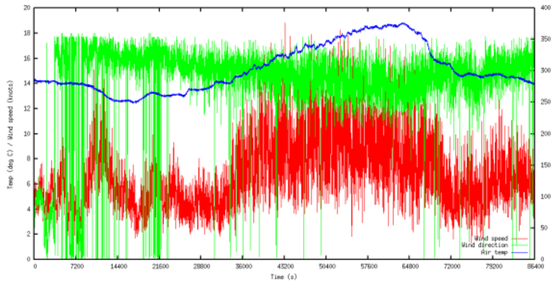

Figure 2: Wind scenario conditions for BMP-2 test case

Figure 3: Thermal model geometry for BMP-2

In order to maintain fast solve time for simulations on the order of 1-2 days of simulated time, a coarse mesh performs well for this case and still captures wind flow phenomena.

When determining flow solver meshing parameters for this case, our goal was to maintain at least real-time simulation performance using a 5-minute thermal solver time step size. The key inputs are the base meshing size, refinement levels, CFL number used to determine flow solver time step size, and the maximum transient duration used to reach a steady flow field (for each thermal time step).

Some statistics for typical runtime of the BMP-2 model are shown in Table 1. All cases are using 10 parallel threads and a thermal solver timestep size of 5 minutes. The rapid flow results used a CFL of 10 and a maximum transient duration of 30s for idle and 10s for exercised.

Table 1: Runtime performance for BMP-2 wind model

Vehicle condition	Runtime (0-D McAdams wind)	Runtime (RapidFlow, 0.35m base size, 1 refinement level)
24-hr idle	1732s (50x faster than real-time)	34077s (2.5x)
24-hr exercised	1406s (60x)	38715s (2.2x)

Sample results from the rapid flow simulation are shown below. Both the idle and exercised cases include the hot air (mix of engine cooling air & engine exhaust) venting from the engine compartment into the windflow domain. Snapshots are taken from the same timestep early in the diurnal simulation. At this time, there is a low wind speed of 2.5 m/s at a direction of 107.4 degrees (primarily from the minus-Y direction, angled slightly towards the plus-X direction). In the exercised case, the vehicle is driving at 18 m/s with a heading of 0 (equivalent to airflow from the plus-X direction) which dominates the windflow condition.

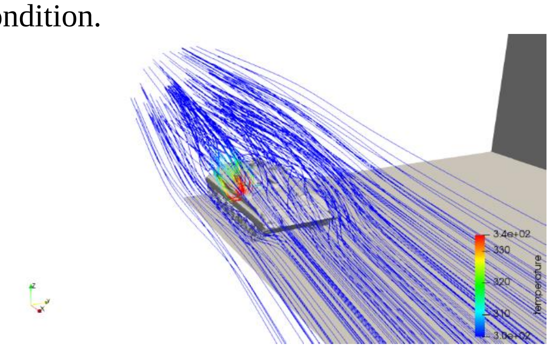

Figure 4: RapidFlow simulation streamlines colored by temperature, BMP-2 model, exercised

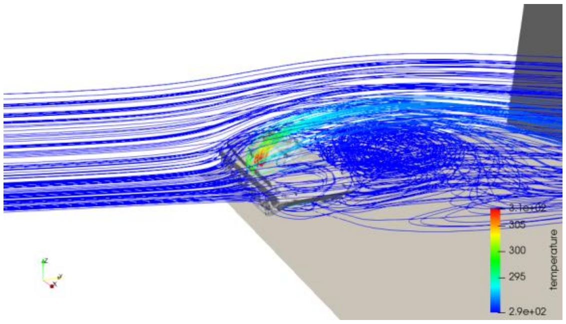

Figure 5: RapidFlow simulation streamlines colored by temperature, BMP-2 model, idle

The following images show a comparison of the convection boundary conditions predicted at one particular timestep within the exercised diurnal simulation. The 0-D McAdams wind model results show uniform convection conditions that do not account for local variations in flow speed or advection of heat sources around the vehicle.

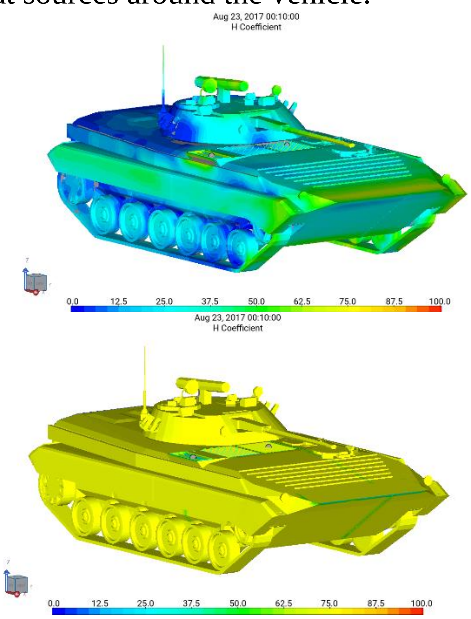

Figure 6: Comparison of heat transfer coefficients between RapidFlow (top) and McAdams 0-D wind (bottom), BMP-2 model, exercised

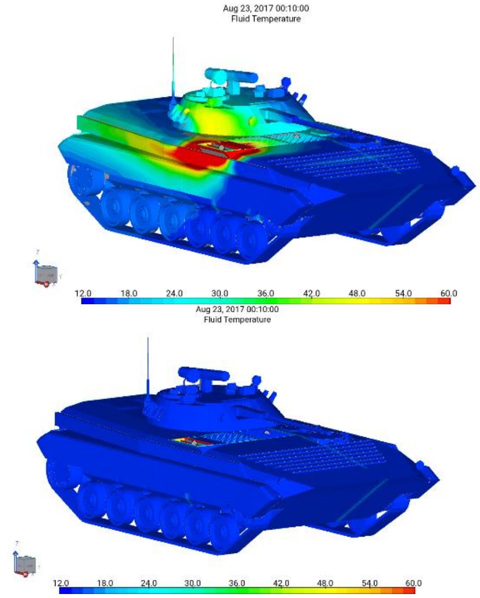

Figure 7: Comparison of fluid temperatures between RapidFlow (top) and McAdams 0-D wind (bottom), BMP-2 model, exercised

4.2. Comparison to conventional CFD simulations

We have compared the flow and thermal simulations using our RapidFlow solver approach to conventional CFD simulations for models including external cross flow over a cylinder. This case is defined by incoming airflow at 50 m/s and 35^o C while the cylinder surface (50mm in diameter) is fixed at 150^o C. This scenario is expected to result in convective heat transfer of 620 W from the cylinder surface [3]. In conventional CFD, this test case was found to be very sensitive to mesh size, particularly the near-wall mesh size and refinement in the wake area. CFD results from a better-performing mesh are shown below along with equivalent results from the RapidFlow solution.

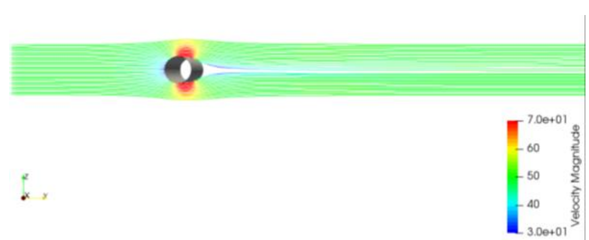

Figure 8: Conventional CFD results for cylinder in crossflow (computed heat rate = 637.4 W)

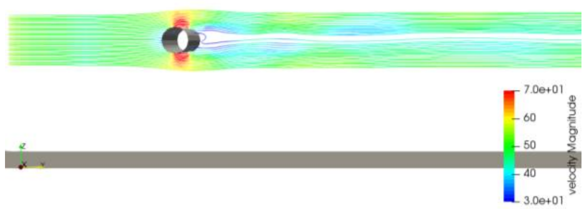

Figure 9: RapidFlow results for cylinder in crossflow (computed heat rate = 611.3 W)

Significant differences can be seen in the flow-field results, including differences in the peak acceleration above and below the cylinder and in the size and nature of the wake region behind the cylinder. Despite those differences, both simulations provide similar levels of error relative to the expected solution. This illustrates one of the primary design goals of RapidFlow: providing enough fluid dynamics fidelity to achieve reliable results when heat transfer is the primary simulation focus while requiring less model creation time and complexity.

5. CONCLUSION

Research, development, and testing of the RapidFlow solver technology has demonstrated practical value for solving the engineering problems of interest. Based on the current state of the code, scenarios similar to wind flow over ground vehicles or HVAC climate control inside cabins will be among those to which the technology is most applicable. The early results discussed here already demonstrate that this approach can fill a gap between conventional 3D CFD and simplified 0D/1D methods that provides value for heat transfer and EO/IR modeling.

Significant future work involving this technology is planned, including further development of the software and application testing and validation. Full parallelization using distributed CPUs and GPUs will allow faster turnaround time of simulations and higher resolution meshes. Further refinements of our physics models, such as additional buoyancy and turbulence models, may be necessary for accuracy in some flow situations. Improvements to the user experience will further make the approach accessible and easy to use.

6. REFERENCES

[1] Reynolds, O. "On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion." In Proceedings of the Royal Society-Mathematical and Physical Sciences, vol. 451, no. 1941, pp. 5-47, 1995.

[2] Tritton, David J. “Physical fluid dynamics.” Springer Science & Business Media, pages 127-128, 2012.

[3] Holman, Jack Philip. “Heat transfer”, McGraw-Hill Inc., Ex. 6-8, 1997.