7-22-2019

Application of a Diverging-Converging Duct to a FSAE Radiator System

Bryce Thomas
Washington University in St. Louis

Ramesh K. Agarwal
Washington University in St. Louis

Application of a Diverging-Converging Duct to a FSAE Radiator System

Bryce Thomas[1] and Ramesh K. Agarwal[2]
Washington University in St. Louis, St. Louis, MO 63130

The focus of this paper is the design process for the radiator ducting on the WashU Formula Society of Automotive Engineers racing team vehicle. The side-pods for the WUFR-19 vehicle are designed as a diverging-converging duct around the radiator in order to provide a sufficient mass flow rate through the heat exchanger to cool the engine, while minimizing drag on the vehicle. Internal computational fluid dynamics simulations are run in ANSYS Fluent on an individual channel of the radiator core. A parametric setup is used to iterate through multiple inlet conditions. The goal of these simulations is to determine the relationship between the ducting inlet size, ram pressure and mass flow rate through the radiator core, while also accounting for heat transfer from the radiator in airflow conditions. The usage of internal simulations with varying inlet conditions reduces simulation time when compared to external side-pod simulations that have higher computational requirements and would require geometry modification between simulations. Results show that reducing the inlet size increases the pressure drop across and mass flow rate through the radiator for inlet sizes above 80% of the cross-sectional area of the radiator. Smaller inlet sizes begin to restrict airflow causing the mass flow rate to decrease. An inlet size of 70% of the cross-sectional area of the radiator and an outlet size of 80% of the inlet size were implemented in order to fully cool the engine and minimize drag on the vehicle.

I. Introduction

A. Wash U Racing

The Washington University in St. Louis Formula Society of Automotive Engineers (FSAE) racing team is a student run organization that designs and manufactures an open wheel formula style racecar on a yearly design cycle. With increased recruitment after the 2017-2018 season, there are over a hundred students coming from every discipline in the McKelvey School of Engineering along with students from the College of Arts and Sciences, the Olin Business School and the Sam Fox School of Design and Visual Arts. The rapid growth and development in the Wash U Racing team has allowed for more in-depth designs and optimization that were not implemented in the Bears Formula Racing 2018 car. The 2018-2019 design cycle marks the beginning of a new design platform designated as the Wash U Formula Racing 2019 (WUFR-19). The team competes at local St. Louis Sports Car of America (SCCA) autocross events and the FSAE Michigan international competition.

B. FSAE Competition

The FSAE Michigan competition is held annually at the Michigan International Speedway against approximately 120 other teams from around the world. It is scored based on the results of three static events and five dynamic events. The static events include presentation, design and cost analysis. The dynamics events are comprised of acceleration, skid-pad, autocross, fuel efficiency and endurance. The scoring breakdown for the eight events is shown in fig. 1.

THE FOLLOWING POINTS ARE POSSIBLE:		
Static Events	75	Presentation
(Thursday)	150	Design
	100	Cost Analysis
Dynamic Events	100	Acceleration
(Friday & Saturday)	75	Skid-Pad
	125	Autocross
	100	Fuel Efficiency
	275	Endurance
TOTAL POINTS:	1000	

Figure 1. FSAE scoring breakdown [1]

Although the application of a diverging-converging duct to a radiator system influences numerous scores, it has the most potential to improve the team's performance in the design, endurance and fuel efficiency events. During the design event, each of the system leads for the WashU Racing team presents to an industry expert on the design process for their respective systems. The judge's goal is to gauge understanding and depth of engineering concepts. The endurance event is a 22-kilometer race designed to test the performance, quality and durability of each vehicle. An overhead view of the course is shown in Fig. 2.

Figure 2. Overhead view of the FSAE Michigan endurance race course

The event may only be attempted once, and no work or maintenance may be done on a vehicle once the event is started. The race is divided into two runs with a 3 min driver switch period at the halfway mark. During the endurance event, engine overheating causes a high risk of not finishing the event resulting in a DNF (did not finish). The score for a DNF is only one point per completed lap. Additionally, the efficiency event scoring is based on the fuel usage during the endurance event. If the endurance event is not completed, a score of zero is given for fuel efficiency.

II. WUFR-19 Cooling System

The WU Racing 2019 car is powered by a 2008 Yamaha YZF-R6 4-cylinder engine that peaks at nearly 83 horsepower. The high-power output for a four-cylinder compared to one-cylinder engines employed by many competitors, results in a higher cooling requirement. The cooling of the system is managed by the radiator. Although some teams run radiators at the rear of the vehicle to reduce coolant line weight or at the outlets of undertray diffusers, the most common design is a single side mounted radiator. Two parallel side mounted radiators on either side is less frequently used due to the added weight of coolant lines and tanks. The design of the radiator for the WU Racing 2019 car was completed by the powertrain system using a genetic algorithm that employs the NTU (Number of Transfer Units) effectiveness method for analyzing heat exchanger performance. Turbulent conditions are assumed for the calculations in the NTU code. The code optimizes for cooling capacity and weight by adjusting the tank dimensions, core dimensions and fin density [2].

The radiator designed for the WUFR 19 vehicle is angled outwards towards the rear of the car at 39° from perpendicular to the car centerline. This angle was determined to have the highest efficiency based on a series of scale wind tunnel tests. Heat is transferred to the coolant as it is rejected by the engine. This is because the engine does not act as an ideal Otto cycle in which all heat would be lost through the exhaust. Per FSAE rules, water is implemented as the coolant rather than anti-freeze. The water is pumped through the swirl pot and into the upper port of the radiator, passes through the radiator where it is cooled by air flowing through the radiator core and out the lower port back to the engine. This causes a vertical heat gradient in the radiator core. This gradient is largest at startup but will decrease as the car is in operation. As a result of minimal nodal temperature data on the radiator and inaccuracies in relative thermal camera values, the average radiator temperature values will be utilized in calculations. Additionally, there is a 1220 CFM fan with a shroud attached to the rear face of the radiator to prevent overheating during idling or stall. The operation of this fan should be limited during driving to reduce battery drain. This can be done by improving the radiator efficiency via a converging-diverging duct design. The fan and shroud system are neglected in this paper in order to reduce computational complexity and focus on optimizing the radiator ducting. A CAD of the radiator assembly was created by the powertrain system as shown in fig. 3.

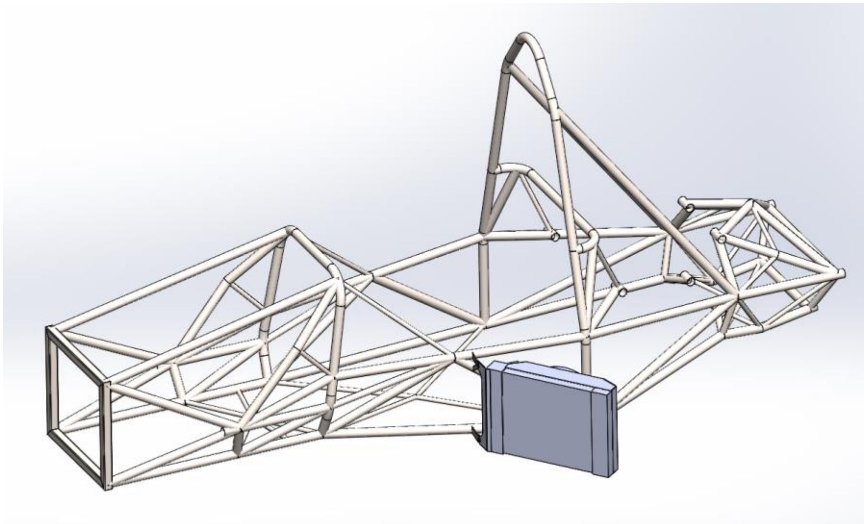

Figure 3. WUFR-19 Radiator CAD Model

The decision to angle the radiator outward instead of forward as in previous years presented a new challenge for the ducting. Due to the side skin curvature and the front radiator bracket used to mount the radiator, there is a sizeable gap between the radiator bracket and the side skin. This gap would allow a significant amount of air flowing over the body of the car to bypass the radiator and reduce the cooling efficiency. This large gap would also prevent any ducting from creating an even pressure distribution across the radiator to take full advantage of every channel in the core. In order to account for the gap, a flat carbon panel was manufactured that extends from the front face of the radiator bracket to the side skin parallel to the front face of the radiator. Another concern when designing ducting for a radiator is the closeout around the edges. At the FSAE competition speeds, full closeout has the potential to create backflow and thus drag; however too large of gaps can reduce cooling efficiency.

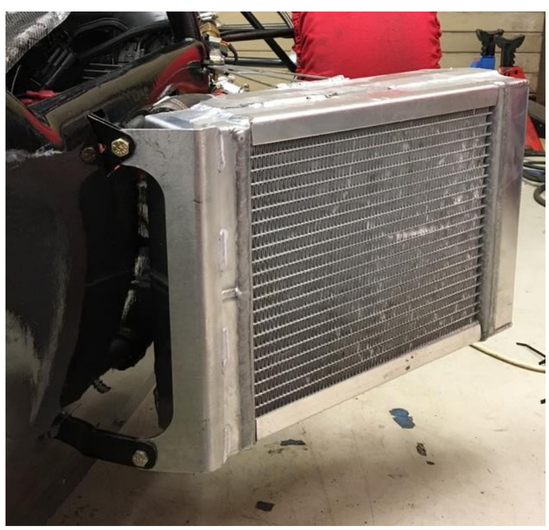

Figure 4. WUFR-19 radiator-body gap

As a result of minimal sidepod design and validation in previous years, the core density of the radiator was kept at 12 fins per inch for the previous several iterations. The custom ordered aluminum crossflow radiator for the WUFR 2019 car, shown in fig 4. has a 10.25 by 11 by 2-inch core and a fin density of 15 fins per inch. The overall dimensions for the radiator are 14.75 by 10.25 by 2. These dimensions do not account for the addition of the radiator fan shroud.

Figure 5. WUFR-19 radiator front view

III. Side-pod Ducting

The primary purpose of sidepods for F1 racing is to reduce drag on the vehicle as a whole. At higher F1 speeds, higher core fin densities can be utilized that provide more surface area for heat transfer to occur. Higher fin densities also require a larger pressure drop to push air through the radiator core. However; lower track average speeds at FSAE competitions mean that a converging diverging duct can be used to increase the pressure drop across the radiator; therefore, allowing for the usage of higher fin densities and an increase in the efficiency of a cooling system. Another difference between FSAE cooling systems and F1 is the closeout of the radiator. F1 vehicles fully closeout the radiator to channel all of the air through it. Due to lower pressure differences at FSAE speeds, full closeout would result in flow reversal. This would decrease cooling efficiency and increase drag. Additionally, since the surface friction factor and heat rejection are dependent on the Reynolds number, the radiator ducting cannot be optimized for the entire velocity range of the vehicle. The duct is optimized for the vehicles average track speed of 30 mph (13.411 m/s). The implementation of a curved lip on the inlet of a sidepod duct is common in order to minimize pressure losses at the inlet and improve efficiency in off angle flow conditions. The curved lip was not implemented in this design as a result of manufacturing difficulties.

Another consideration is the necessity for a guard built into the sidepod. The guard is made out of a mesh material that prevents rocks and other debris from entering the sidepod and damaging the fins of the radiator. Fins hit by rocks and other track debris can be bent backwards which prevents air from entering the channel and reduces the cooling efficiency of the radiator. Bent fins can be seen in the right image of figure 5 after several round of testing without a sidepod guard. The radiator guard is neglected in this report because the pressure drop across the mesh is minimal compared to the pressure change both in the sidepod and across the radiator.

IV. Design Constraints

A. FSAE Constraints

The primary physical constraints of the FSAE side pod for the WUFR-19 are the side impact structure of the frame and the tire locations. Clearance requirements must be met with respect to the wheels in order to pass rule T.1.1.2 that covers open wheel vehicle criteria.

> *"No part of the vehicle may enter a keep out zone defined by two lines extending vertically from 75 mm in front of and 75 mm behind, the outer diameter of the front a rear tires in the side view elevation of the vehicle, with tires steered straight ahead. This keep out zone will extend laterally from the outside plane of the wheel/tire to the inboard plane of the wheel/tire." [1]*

These keep out zones, with respect to the WUFR-19 wheelbase of 1537 mm and wheel diameter of 10 inches, limit the length of the portion of the sidepod extending past the inner plane of the wheels to 922 mm.

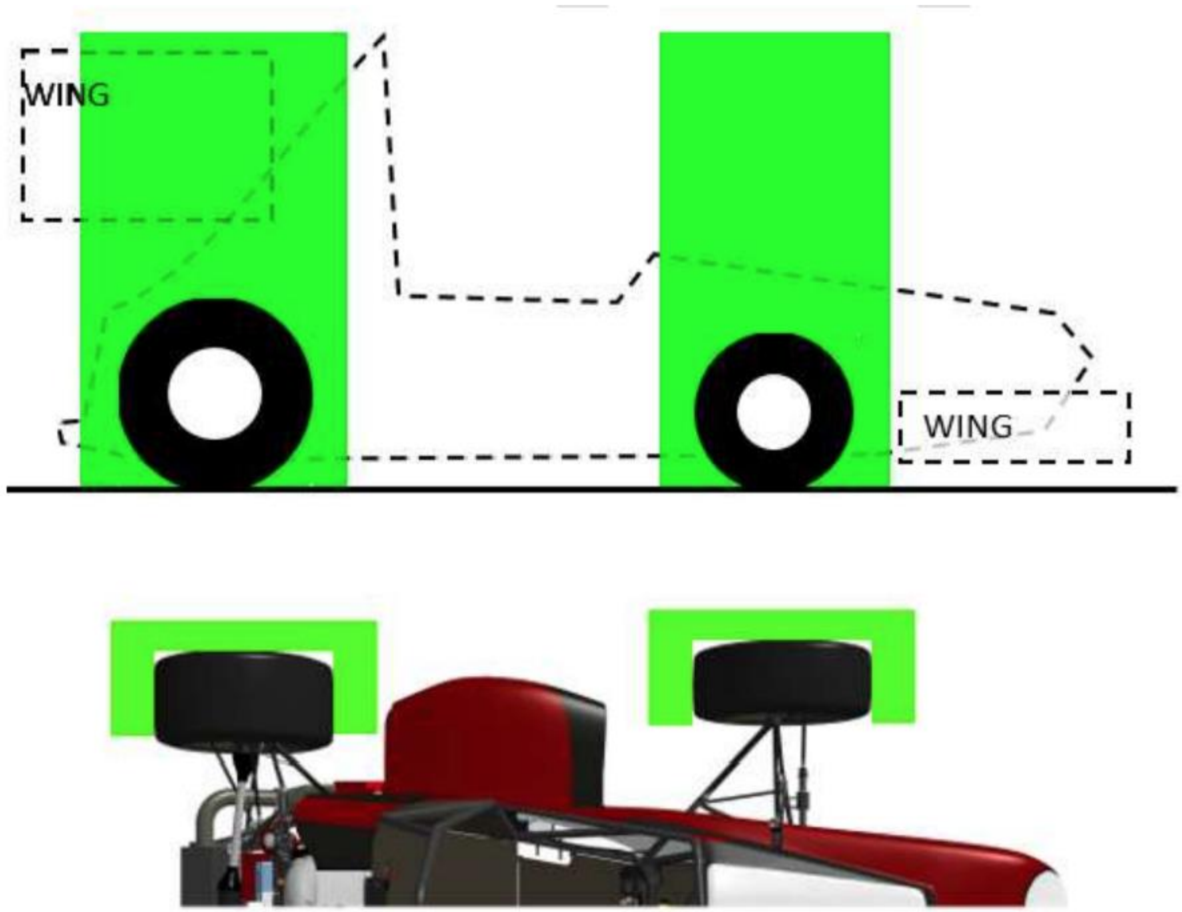

Figure 6. FSAE rule T.1.1.2 regarding tire keep out zones [1]

The location of the radiator is also dependent of the side impact geometry of the frame that the radiator must be fastened to with welded tabs. Due to difficulties with tab orientation and welding near the main node between the front and rear roll hoops, the radiator was mounted just behind the node. Placing the radiator further forward in front of the node would result in a minimal inlet length with a sharp curvature. This would increase the risk of flow separation within the duct that would increase vortices and drag on the vehicle.

B. Exhaust Constraints

Along with the other WUFR-19 platform changes, the exhaust was redesigned to extended forward in front of the engine and curve back rather than only extending rearwards. The new exhaust configuration is shown in figure 7.

Figure 7. WUFR-19 exhaust system

This redesign had two primary benefits. It improved the vehicle's weight balance from the front to the rear and contributed to the lowering of the vehicle's center of gravity, both of which play a large factor in drivability and cornering during dynamic events. Per FSAE rule IC.7.2.3,

> *"Any exhaust components (headers, mufflers, etc.) that protrude from the side of the body in front of the Main Hoop must be shielded to prevent contact by persons approaching the vehicle or a driver exiting the vehicle" [1]*

Thus, the side pod on the driver's right-hand side of the vehicle is required to adequately protect the driver from both convection and radiation. A secondary concern was the heat transfer from the exhaust to the carbon fiber ducting. Higher temperatures are required for delamination of the composite material; however, the glass transition temperature of the epoxy would be reached making the duct more brittle. Since minimal loading occurs on the side pods, the structural integrity would not be compromised as a result of high temperatures. These issues were solved through the use of heat sealant tape. The main impact of the exhaust on the side pod design is that it dictates the length since the side pod must be long enough to fully cover the exhaust.

V. Radiator Simulation Methods

General design practices for radiator ducting utilize pitot tubes or pressure taps in order to measure the pressure drop across a radiator core. This is because creating a mesh/grid of a dense radiator core for computational fluid dynamics software would require many millions of cells in order to accurately simulate flow. One way to model a radiator is as a porous jump boundary using a known pressure drop. The porous medium has a pressure change of a finite thickness defined as a combination of Darcy's Law and an inertial loss term as follows [3]:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \quad (1)$$

A porous jump boundary condition was implemented instead of a porous region in order to reduce computation time for full car simulations. Although a porous jump region can account for heat transfer through the medium, it cannot properly solve for the heat transfer between multiple materials. The usage of a porous jump boundary condition requires a known relationship between the velocity and pressure drop through the radiator core in order to calculate the pressure jump coefficient C_2. The porous medium thickness (Δm) and face permeability (α) are also required inputs in order to simulate a porous jump boundary condition. Without readily available sensor data, an internal radiator core simulation method was implemented in order to computationally solve for these values.

VI. Radiator Core Computational Model

In order to create the geometry for the preliminary radiator core simulation, digital calipers were used to precisely measure the dimensions of an individual radiator core channel. A single channel was modeled in SolidWorks CAD and imported into the ANSYS fluent software package as shown in figure 8.

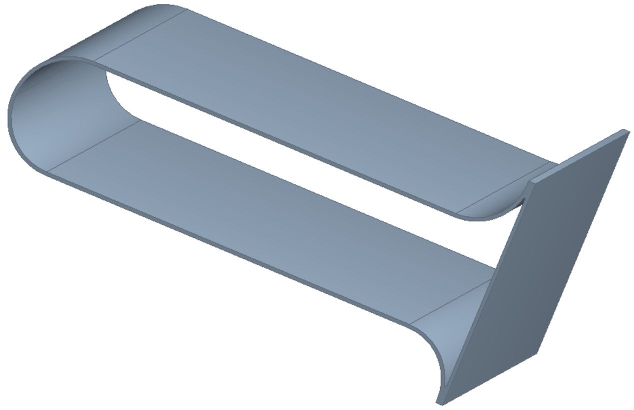

Figure 8. Radiator core channel model

Since a well-designed diverging-converging duct creates an even pressure distribution across the radiator, the radiator core simulation operates on the assumption that each channel has similar flow conditions. Another simplification is the usage of the average operating temperature of the radiator, 200° Fahrenheit as measured during vehicle operation, for heat transfer calculations. The heat transfer is simulated by setting thermal boundary conditions at the walls of the channel assuming thermal equilibrium in the radiator. The heat transfer is important to simulate because it affects the viscosity of the air and conversely the Reynolds number of the flow. Major pipe pressure losses are dependent on the Reynolds number, as well as the relative roughness of aluminum of 0.0015 mm, to calculate the friction factor. Therefore; the pressure drop across the radiator is affected by the heat transfer. The radiator core simulation was set up as a parametric study of 11 different cases. A MATLAB code was created in order to calculate the inputs for each case: velocity magnitude, velocity z-component, velocity x-component, turbulent intensity, turbulent length scale and the ram pressure. Each case corresponds to a different area ratio of the diverging-converging duct inlet to the cross-sectional area of the radiator ranging from 1.0 to 0.5. The velocity components are necessary to account for the angle of the radiator. This allows for the simulation to show the effect of duct inlet sizing on mass flow rate within the radiator core. The ambient temperature and air density were set based on the design criteria that the sidepod be able to provide a high enough mass flow rate through the radiator to cool the engine on an 80° F day. The air density is simulated as an ideal gas with viscosity varying with temperature according to the three-coefficient form of Sutherland's law as follows [3]:

$$\mu = \mu_0 \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0+S}{T+S} \tag{2}$$

The turbulent intensity at the core of the fully developed duct flow is estimated using the following equation derived from an empirical correlation for pipe flows [3]:

$$I = \frac{u'}{u_{avg}} = 0.16 * (Re)^{-1/8} \tag{3}$$

Since turbulent eddies cannot be larger than the duct itself, an approximate equation can be utilized to calculate the turbulent length scale in which a factor based on the mixing length in fully-developed turbulent pipe flow is implemented as follows [3]:

$$l = 0.07 * L \tag{4}$$

The near wall approach was implemented in the radiator core simulations in order to accurately simulate turbulent flow with heat transfer. The near-wall model approach requires a mesh resolved down to a $y^+ \leq 1$. This is achieved by calculating the first layer thickness, Δs, of the initial element layer near the wall and creating a logarithmic inflation layer in the mesh. The mesh outside the logarithmic inflation layer is comprised of unstructured tetrahedrons with high orthogonal quality and low skewness values. The final mesh/grid with ~1,750,000 elements is shown in figure 9.

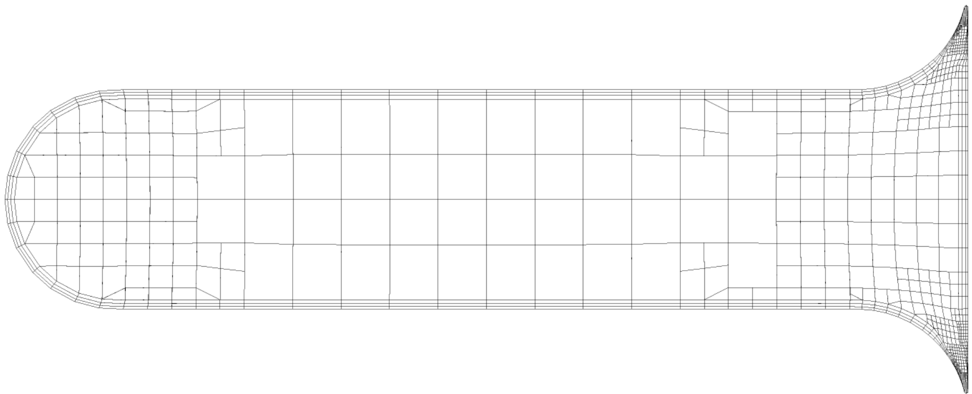

Figure 9. Radiator core channel mesh

The k-omega SST (shear-stress transport) model with two transport equations representing the turbulent kinetic energy and the specific dissipation was used for the radiator core parametric study. This turbulence model was used because of its ability to accurately simulate internal flow and heat transfer. The k-omega SST model differs from the baseline (BSL) k-omega model by a modification of the eddy viscosity definition to account for the turbulent shear stress. The Menter k-omega shear stress transport equations can be written as [4]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{6}$$

VII. Radiator Core CFD Results

Table 1 shows the channel and total radiator core mass flow rate for each inlet area ratio simulated. The radiator sizing genetic algorithm estimated a required mass flow rate of 0.75 kg/s through the radiator in order to cool the engine. However, the actual required mass flow rate is closer to 0.60 since approximately 5.8% of heat rejected by the engine is released to engine oil and 14.0% is release to ambient air [5]. Since the WUFR-19 has minimal covering around the engine compared to a standard car, the heat released to ambient air is likely even higher. The total radiator core mass flow rate was calculated by scaling the channel mass flow rate by the number of total channels in the 15 FPI radiator core.

Table 1 Radiator Core Mass Flow Rates

Area Ratio	Turbulent Intensity	Duct Ram Pressure [kPa]	Channel Mass Flow Rate [kg/s]	Total Radiator Core Mass Flow Rate [kg/s]
1	4.1847%	0	0.00015361	0.5576043
0.95	4.2116%	11.36	0.00016221	0.5888223
0.9	4.2402%	22.137	0.0001683	0.610929
0.85	4.2706%	32.331	0.00017198	0.6242874
0.8	4.3030%	41.943	0.00017341	0.6294783
0.75	4.3379%	50.973	0.00017269	0.6268647
0.7	4.3755%	59.42	0.00016998	0.6170274
0.65	4.4162%	67.284	0.00016541	0.6004383
0.6	4.4606%	74.566	0.00015909	0.5774967
0.55	4.5094%	81.265	0.00015116	0.5487108
0.5	4.5634%	87.382	0.00014175	0.5145525

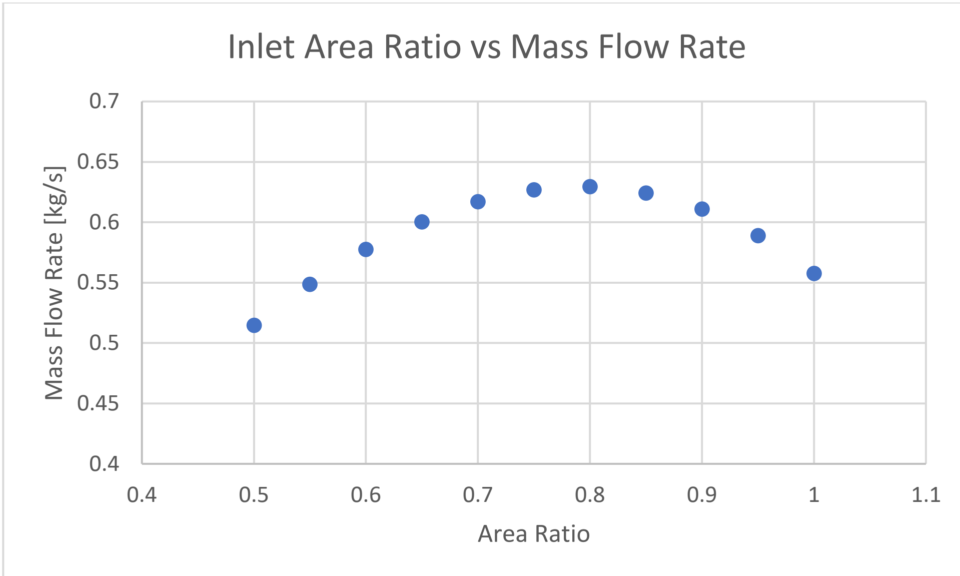

Figure 10. Plot of the total mass flow rate of the radiator with respect to the diverging-converging duct inlet sizing

It can be seen from figure 10 that reducing the inlet ratio from 1 to 0.8 can increase the pressure drop across the radiator and cause an increase in the mass flow rate through the radiator. An inlet size less than 80% of the cross-sectional area of the radiator, begins to restrict flow through the inlet and into the radiator core thus reducing the mass flow rate through the radiator core. Higher inlet areas also result in minor backflow near the radiator core channel outlet as a result of lower pressure drops hence reducing the mass flow rate. The inlet for the WUFR-19 diverging-converging radiator duct was set to be 70% of the radiator cross sectional area. This inlet size yields a high enough mass flow rate to sufficiently cool the engine, with a degree of safety accounting for off angle flow conditions, and has the potential to result in a lower drag coefficient in comparison to higher area ratios simulated. Based on the pressure drop across the radiator computed by the 0.7 inlet ratio simulation, the required outlet size was calculated to be 80% of the inlet. The converging portion of the radiator duct increases the velocity of the air at the outlet so that it is equal to the velocity of the freestream air moving around the exterior of the sidepod. This minimizes drag caused by vortices in the wake region of the side-pod.

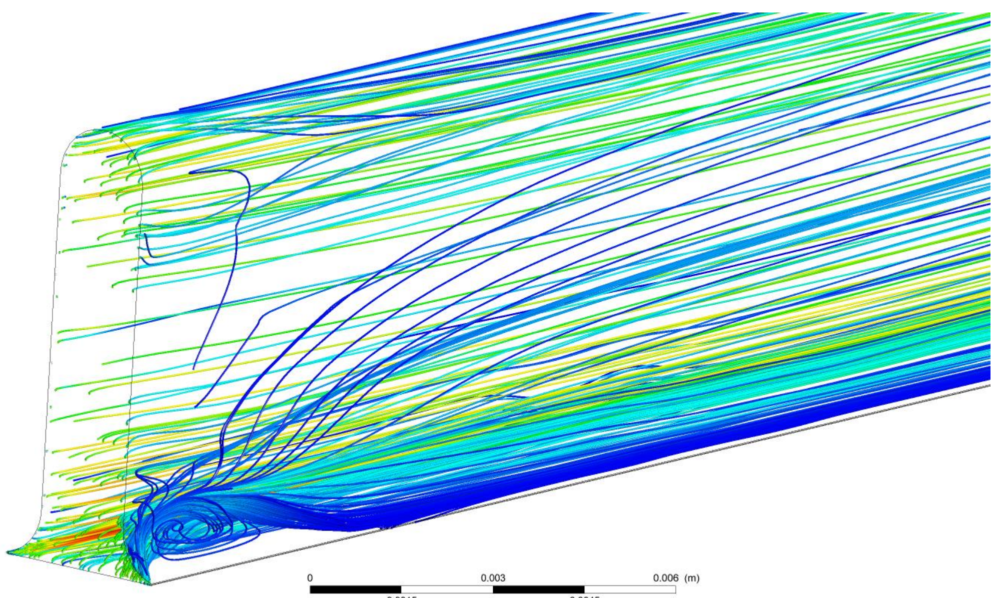

Figure 11. Velocity streamlines of air entering a radiator core channel at a 39° angle

A plot of the velocity streamlines through the radiator core channel is shown in figure 11. The flow entrance angle of 39° causes vortices to form at the lower right corner of the inlet. This increases the distance necessary for the flow to fully develop. The act of angling the radiator has benefits for improving effective surface area for the core, but also causes more drag due to vortices forming within the core.

VIII. Side-pod Modeling

The design of carbon fiber components is completed in the SolidWorks CAD package-using surface modeling techniques to model the minimal thickness and complex curvature of composite parts. Unlike traditional CAD practices, surfaces are lofted between sketches to create a complex contour that can be extruded to a chosen thickness. Furthermore, surface modeling allows for further accuracy when contouring to the frame of the car within an assembly. Designing within an assembly is crucial to ensure proper fitment of the diverging converging duct with respect to the body of the vehicle, the radiator and the exhaust. Flanges are created at the upper and lower edges where the side pod meets the body to facilitate the usage of Dzus Lion quick release fasteners.

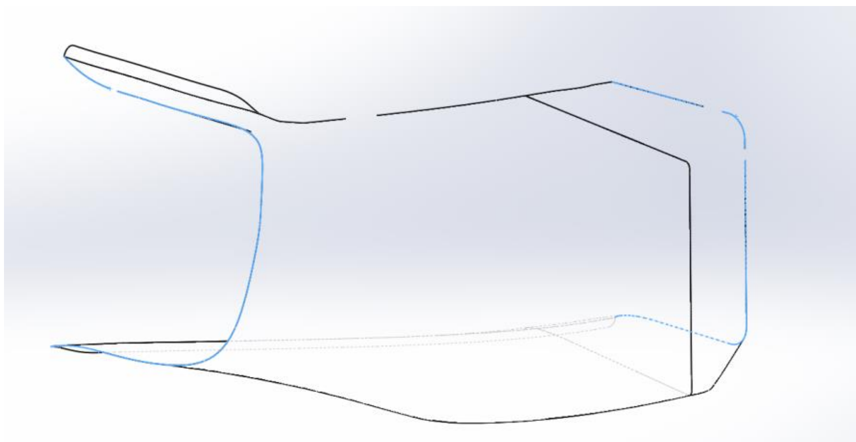

Figure 11. Wireframe view of the WUFR-19 finalized side-pod duct model

IX. Manufacturing

A. Mold Manufacturing

Once a component design is approved through the team design review process, a mold is then created in the SolidWorks CAD package. The desired surface finish of the part determines whether the mold will be concave or convex. This is because when the component is manufactured, the side of the part face down on the mold will have a higher surface quality. Once the mold is designed such that there is an adequate flange and low edge gradient for vacuum bagging, the mold needs to be manufactured through CNC operations. The mold is put into a computer-aided machining (CAM) software (FeatureWorks or Rhino) that is used to determine the path that the CNC mill cutting tool takes and the bit utilized in the machine. The mold is then manufactured using subtractive cuts from either medium density fiberboard (MDF) or high-density Elfoam (polyisocyanurate). Any cracks or holes are filled with automotive grade Bondo and sanded down. Next the surface finish of the mold is iteratively sanded down up to one thousand grit and primer is applied. Higher priority molds have a fiberglass layer applied by our sponsors at Accuwright Fiberglass Co. Once the surface finish of the mold is ideal, a 2-inch-thick line of blue tape is place around the flange edge of the mold. This prevents chemical agents from affecting the surface of the flange and allows for easier vacuum sealing of the mold. Lastly, a thin layer of release agent is sprayed onto the mold. The release agent dries as a thin film and prevents the carbon fiber layer from sticking to the mold. When the part is ready to be pulled of the mold, the water-soluble release agent can be washed away.

B. Duct Manufacturing

The core of the WashU Racing's composite manufacturing is done in house by the students. However, due to equipment limitations some large-scale computer numerical control (CNC) mill operations and mold fiberglass application is completed with the assistance of team sponsors. We use a wet layup process for manufacturing in which resin and hardener are mixed and applied to dry layers of carbon fiber. The other two most common types of layups are pre-preg and resin infusion. The first uses carbon fiber that already contains the optimal ratio of epoxy, and the other involves the use of a resin inlet and outlet to pull resin through the dry fiber during the vacuum bagging process.

The first is not used due to high cost and the necessity of storing pre-preg carbon fiber in a low temperature environment while the second requires investment in a different vacuum pump setup. This report will outline a general manufacturing procedure; however, the manufacturing process and complexity tends to vary as a result of component geometry and structural requirements.

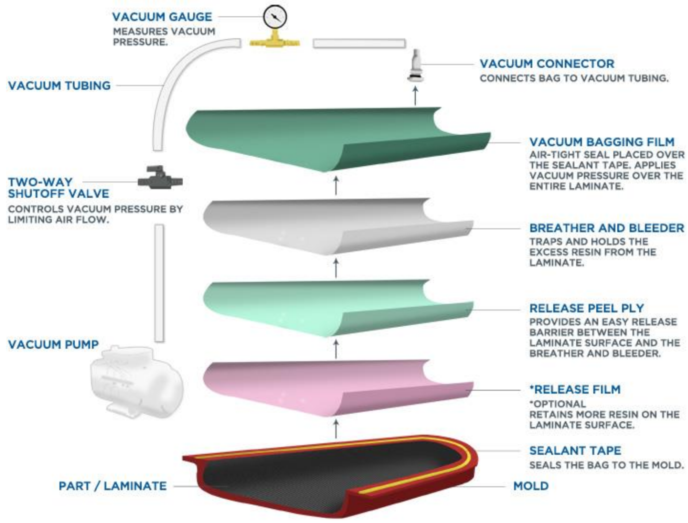

Figure 12. Carbon fiber wet layup process diagram [6]

Once all of the layers for the component are cut using templates, the wet lay-up process can begin. Our epoxy from ACP has a 1-hour pot life (working time), a chemical activation temperature of 75° F and a mixing ratio of 2 parts hardener to 1-part resin. The amount of epoxy mixed is calculated based on the surface area and density of the layers of carbon fiber. The layers of carbon fiber are then laid on the mold and wetted up until no dry spots are present. A core can be sandwich in between layers during this process. Then peel ply and barrier cloth are placed on top of the composite layers. The peel ply is a porous Teflon coated PTFE fiberglass that allows excess epoxy to seep through but prevents the barrier cloth from sticking to the final part. The cloth serves to absorb excess epoxy and approach the optimal ratio of epoxy to carbon fiber in the layup. Tacky tape and vacuum bag are then used to seal the mold as an E2M2 vacuum pump compresses the layers down. The epoxy takes 24 hours to fully cure and then can be removed from the mold.

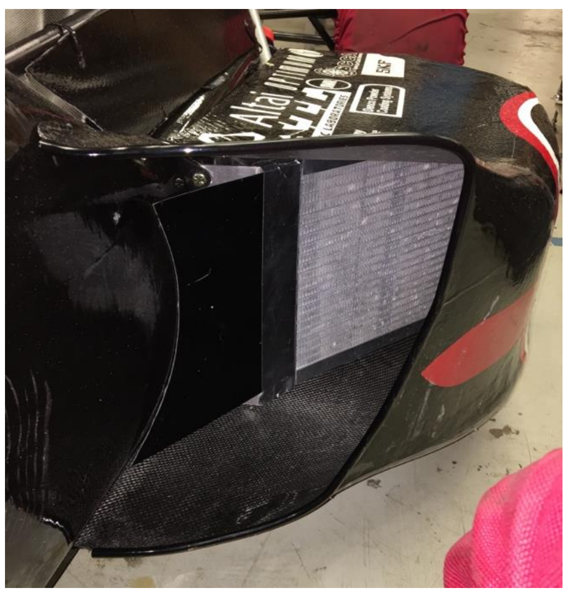

Figure 13. WUFR-19 carbon fiber side-pod ducting around the radiator

X. Conclusion

The mass flow rate results from radiator core channel simulations are viable values with respect to the required mass flow rate of 0.60 kg/s calculated from the heat rejection of the Yamaha R6 engine. Simulations show that the inlet size has a parabolic relationship with the mass flow rate through the radiator. This relationship can be used to properly size the inlet and outlet of a diverging-converging radiator duct for mass flow and minimal drag. Across three days of vehicle operation, there were no signs of engine overheating. This supports the implementation of internal radiator channel simulations to optimize a diverging-converging duct. This design process can be further validated through comparison with an external CFD simulation of the WUFR-19 side-pod duct with the radiator modeled as a porous jump-zone.

Figure 14. WUFR-19 at the 2019 FSAE Michigan competition

XI. Continuation

The length of the sidepod was set for the WUFR-19 car as a result of the exhaust length and the positioning of the radiator was determined by frame node mounting locations. If these constraints are adjusted by future designs, the length of the sidepod can be optimized to reduce the length of the sidepods and conversely the weight. A MATLAB code has been developed for future optimization that can streamline the CAD process of multiple diverging converging ducts for iterative CFD simulations. The code has inputs for the radiator dimensions and angle, the length of the duct and the area ratio of the sidepod inlet to the cross-sectional area of the radiator. The code creates a 3-D plot of the radiator as well as three sinusoidal curves to model the top, bottom and side curvature of the duct. The data points for the curves can be imported to SolidWorks and lofted together to minimize the CAD time for iterative geometry simulations. Lastly, entrance effects can be mitigated through the development of a frontal lip on the side pod to reduce pressure losses.

References

[1] "FSAE Rules 2019," Jul. 2018.

[2] Cecil, J., *Optimization of an FSAE Heat Exchanger using Mixed Integer Nonlinear Programming in MATLAB*.

[3] *Fluent Documentation* Available: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/ai_sinfo/flu_intro.html.

[4] Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, Vol. 32, No. 8, 1994, pp. 1598-1605. c

[5] Seider, G., and Bet, F., "Virtual Approach to Predict Heat Rejection of Combustion Engines," Mar. 2015

[6] "Oven vacuum bag processing-lay-up and bagging guidelines." Cytec Industrial Materials, Woodland Park, NJ, 2012.