AERODYNAMICS OF COOLING FOR RX8 TURBO CONVERSION

INTRODUCTION

This is a short article written during the conversion of a Mazda RX8 with twice the power using a 13B REW engine from a Mazda RX7 FD and a single turbo (the FD has twin sequential small turbos). It covers some relevant theory as background for another article covering its practical application and makes some recommendations where appropriate.

Why write this article? The RX8 cooling system is inadequate, being designed for half of the power of my conversion with no provision for an intercooler. I could just copy what someone else has done, but I want to design a system from scratch using the knowledge and experience I have in aircraft and missile design.

Following this short introduction I will cover some preliminaries, aerodynamic theory and practice from car research, some applicable aircraft research, flow in ducting and flow in intercooler header tanks.

SOME PRELIMINARIES

An engine generates large amounts of heat and an RX8 gets rid of it by 2 oil coolers exhausting into the front wheel wells and a radiator exhausting predominantly over the engine. Some of the radiator exhaust evacuates through the wheel wells and side vents, the latter being of little practical value as it has a narrow and convoluted path (ie the side vents are purely a styling feature). This arrangement is pretty typical of modern cars, combining as it does simplicity, cheapness and ease of manufacture.

The relatively small size and low positioning of the rotary engine means that there is much more space for cooling systems in the RX8 and RX7 than in other cars and considerable scope for innovative solutions.

We shouldn't just assume that manufacturers have got things right. From cars through to early models of iconic aircraft such as the German JU87 Stuka dive bomber and Bf 109 fighter, designers have got things horribly wrong. The cooling of many cars can be improved at minimal cost by sealing gaps around radiators.

We need to cool not only the engine but also air conditioning where fitted, the turbo, gearbox and other items such as the brake fluid reservoir and ignition system that are affected by heat from the engine.

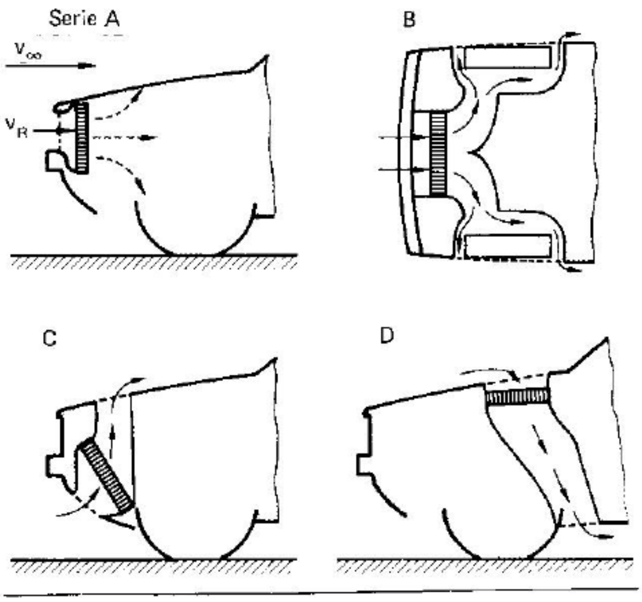
AERODYNAMIC THEORY AND PRACTICE FROM CAR RESEARCH

Put a radiator in an air flow and the majority of the air will spill over the sides of the radiator creating large amounts of drag, a result of the radiator structure being 10-50% of the frontal area, turbulence in the air channels slowing down the cooling air flow and drag from the cooling surfaces. Ducting is needed in front and behind the radiator to take only the air required and introduce it back into the air flow. Air will only flow through a radiator if there is a pressure difference across it and efficient cooling needs the air going through the radiator to be turbulent.

Advances in computing have led to Computational Flow Dynamics (CFD) becoming available to those without big pockets. There are now several free CFD programmes available to enthusiastic amateurs and a number of support groups have sprung up to help someone model a car in a weekend or 2. However, radiators are very hard to model effectively and so there is no open source literature on them and their ducting

The 4 Main Ways of Exhausting Radiator Cooling Air

As shown in the diagram below, there are effectively 4 main ways of exhausting radiator cooling air: to the engine bay; to the front wheel wells; through the bonnet; and to the underside.



Each way will have different effects on the drag of the car and lift at the front wheel. An often-quoted study of cooling arrangements gives the following analysis, where it is important to note that:

	Arrangement	Speed through Radiator As Propn of Car Speed	Contribution to Cd[1] (Drag Coefficient)
A	Front mount venting into engine bay	27%	0.025
B	Front mount ducted to wheel well	14% (52% of A)	0.020 (80% of A)
C	Ducted to bonnet/hood	25% (93% of A)	0.010 (40% of A)
D	Top-mount inlet ducted to underside	10% (37% of A)	0.020 (80% of A)

However, these figures must be taken with care. Option A assumes free exit for the air from the engine bay when this is often not the case in practice and the others involve other variables as, for example, ducting exhaust air from arrangement C along rather than perpendicular to the bonnet will reduce the increase in Cd.

Radiator Position

In general, the radiator should be mounted as low down as possible in order to lower the car's centre of gravity and may be angled by up to 30 degrees without appreciably reducing cooling efficiency. As far as is practical with other considerations, the radiator should be near the engine in order to reduce weight, cost and coolant volume.

Inlet Configurations

[1] Coefficient of drag, equal to drag divided $0.5\rho v^2 A$ where A is a representative area

The radiator requires such a large air mass flow that a forward-facing duct is needed, as low down as possible on the front of the car where pressures are highest. Fancy arrangements such as NACA ducts have inadequate air flows and pressure recovery. Bonnet/hood scoops are inefficient and are generally difficult to integrate with possible radiator locations.

Due to the 3 factors mentioned earlier that restrict air flow through the radiator, the inlet should be smaller than the radiator. I discuss this in greater detail later, but for the moment we can make a quick estimate: the turbulence before and in the narrower channels will reduce airflow speed by say 35-50% and the channels are between 50% and 85% of the frontal area, which combine to give an inlet area of 40-60% radiator area.

The inlet configuration must have a rounded lip and smooth transition along its length in order to avoid flow separation that will choke the duct and hence reduce pressure recovery and increase drag. The simplest duct shape, one with straight sides angled outwards, is known as a rectangular duct; it should have sides angled at no more than 11 degrees to the inlet air flow, which requires long duct lengths of over twice the radiator height.

Surprising to most people, theoretical and practical analyses show that the cooling drag is the same regardless of radiator height, provided the inlet walls avoid flow separation. Larger inlets produce more drag overall because the car's fixed-size radiator effectively limits the air passing through and so they slow and spill large quantities of air that create more drag than if deflected by a smaller inlet.

The optimum velocity for the air presented to the radiator is a balance between cooling efficiency, where (all other things equal) higher velocities give better cooling, and pressure drop across the radiator (and hence drag) where higher velocities give higher drops and hence higher drag.

Exit Configuration

The exit ducting converges to accelerate the exhaust air and therefore has a negative pressure gradient. This means that the rate of change of the duct area is much less important than for the inlet diffuser.

Ideally the exhaust air should be exhausted from an exit perpendicular to the freestream flow. However, we cannot do this. The typical production car method of exhausting through the engine bay underside is relatively inefficient as it creates drag from the exhaust air's bouncing around off the car's rough underside. Air could be ducted out of side vents or into the front wheel wells but, in general, the only reasonable alternative to the bottom of the engine bay is the bonnet/hood. The pressures across the bonnet change from very low at the bonnet lip to slightly positive at the windscreen base, so the earlier the exhaust air exits from the bonnet the better.

The diagram below shows 4 alternative designs of exit through a bonnet/hood showing differences in the measures of pressure and drag (Cp[2] and Cd respectively).

[2] Coefficient of pressure, equal to drag divided by $0.5\rho v^2 A$ where A is a representative area

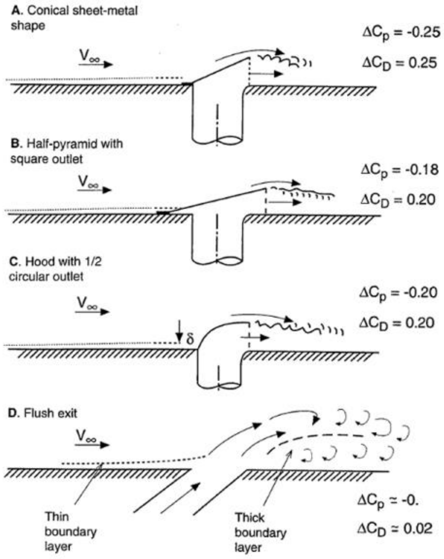


Fig. 6-58. Variety of cooling-flow exits, their drag increment, and typical pressure coefficient values at the exit plane (C_D is based on exposed frontal area).

In A, B and C the exhaust air flowing along the plate generates lift from its higher velocity (and hence lower static pressure), differential pressure on the shape creates drag and the freestream flow creates a turbulent boundary layer at the junction of the 2 flows, generating drag. In case D, however, the 2 flows mix violently, creating a thick turbulent boundary layer that raises the pressure on the plate after the lower pressure of the exit, creating a roughly zero change in Cp; the additional drag from the turbulent boundary layer is largely offset by the saving in drag from not having differential pressures across a cowl, resulting in only a small increase in drag.

A Gurney flap (basically a short vertical plate) ahead of a bonnet vent will help pull air out of a vent by lowering the pressure behind it at the expense of some extra drag. This is why you see a hump ahead at the front of vents in many carbon fibre bonnets/hoods.

If air is passed across the engine or attached systems such as a turbocharger, it should be exhausted through vents in, by order of preference, the bonnet, bodywork sides behind the front wheel, wheel wells and engine bay underside. Air ducted out of the bottom of the engine bay should be as far back as possible and if practical in line with the car underside.

SOME APPLICABLE AIRCRAFT RESEARCH

Introduction

Most theory and professional research relates to aircraft engines where there is little beyond the 1940s, when efforts turned to jet engines. . However, the abundance of home-build aircraft projects over the last 25 years has led to many amateur projects reviving the principles at speeds relevant to cars – a light aircraft climbing at 85 mph has power demands and air flows very much like a car at that same speed. In general,
both theory and practice hold up reasonably well when translating from aircraft to cars albeit some car has to be taken on taking lessons from fast aircraft where Reynolds Number (Re) effects might be significant.

Radiators Exhausting Directly to Rear

Where a radiator ducts immediately to its rear, the least drag and best cooling occurs where the air inlet's height is 1/6-1/3 of the radiator's at one radiator height ahead of the radiator and its exit is 115% of this to allow for heat expansion, again at one radiator height from the radiator. The rate of flow is entirely controlled by the exit height. A properly designed aircraft duct will actually generate thrust rather than drag from the Meredith Effect where the radiator heat gives the cooling air extra energy, as was the case for the legendary P51 Mustang fighter.

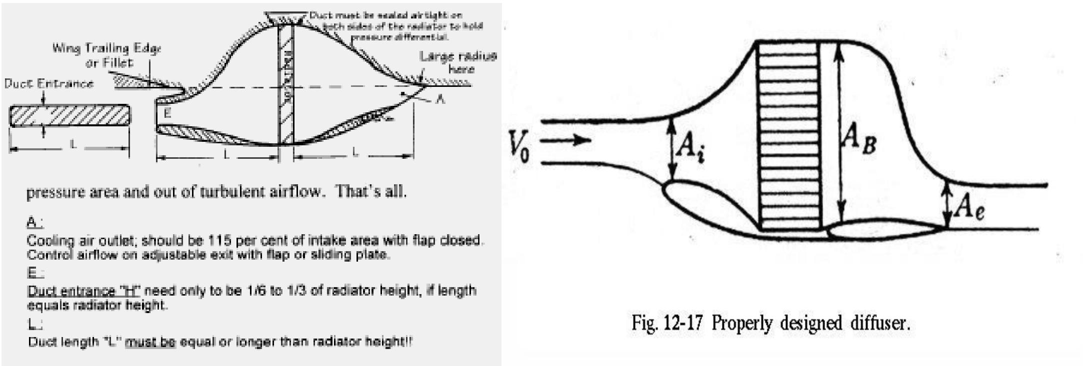


Figure 3. Examples of Ideal Ducts

Many claim the Mustang's arrangement to be the epitome of design. However, this is only true for its main role. Several Mustangs were converted for pylon racing using wing-mounted radiators sited at the now-vacant wing gun bay area; these aircraft were faster than otherwise similar Mustangs in their high speed, low level environment.

Research On Intercooler Cooling Air Flows

Some interesting information can be found in a 1944 US report, NACA ARR 4D07, covering cooling air flow and pressure drops across an intercooler.

The optimum cooling air speed for minimum drag was found to depend on the intercooler design, and these probably haven't changed much since then. The optimum speeds were those that gave pressure drops across the intercooler of 1-3 in water (250-750 Pa). Furthermore, when the system was operating at this optimum cooling-air pressure drop the optimum ratio of cooling air to charge-air weight flow was found to vary with intercooler efficiency as follows:

Intercooler Efficiency	1 in water (250 Pa)	2 in water (500 Pa)	1 in water (750 Pa)
30%	1.2	1.0	1.8
40%	1.7	1.3	1.1
50%	2.3	1.8	1.5
60%	3.1	2.4	2.1
70%	4.2	3.2	2.8
80%	5.7	4.4	3.8

To put these figures in perspective for a car:

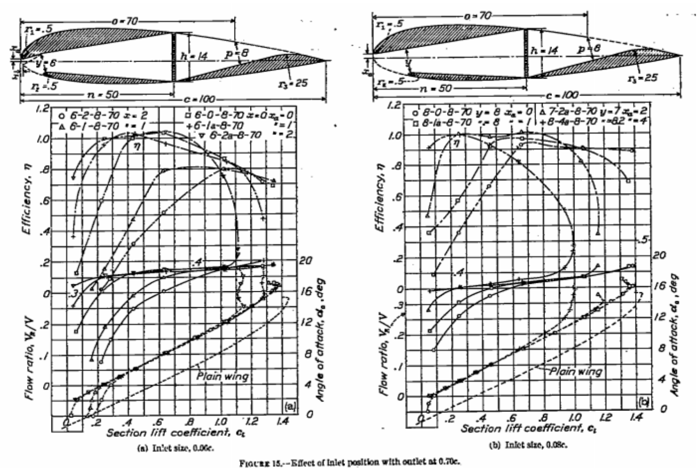
- 1 in/250 Pa and 3 in/750 Pa are the dynamic pressures at sea level at 45 mph and 78 mph respectively.
- I measured the static pressure drop across a car intercooler in the air flow from a hair dryer. This drop was 110 Pa on maximum flow, which was calculated at 30 mph from measuring the free stream dynamic pressure.
- If we take a minimum intercooler efficiency of 70% for cooling at a maximum charge air flow of 850 cfm then we are looking at cooling air flows of 3570 cfm at a 250 Pa drop, 2720 cfm at 500 Pa drop and 2380 cfm at 750 Pa drop. At 90 mph these cooling air flows imply inlet areas of 65 in^2/0.042 m^2, 49 in^2/0.032 m^2 and 43 in^2/0.028 m^2. For the intercooler of 0.13 m^2 that I intend to use this implies ducts with maximum diffuser area ratios of 3.1, 4.1

and 4.6 respectively; at the face speeds implied the pressure drop is likely to be 250 Pa or less, implying a diffuser ratio of 3.1 or less.
- Similarly, for cruising at 70 mph, 150 cfm charge air flow and a radiator efficiency of 90% we are looking at a diffuser ratio of around 4.0.

<u>WW2 Research On Wing-Mounted Radiators</u>

Several successful WW2 aircraft, notably the De Havilland Mosquito, used wing-mounted radiators. Another NACA report, ARR 743, analysed a multitude of such arrangements in a wind tunnel and provides some valuable insights for exhausting air over the bonnet/hood where the low static pressure mimics that over a wing.

The report is not for the faint-hearted as the volume of detail is mesmerising and the multitude of charts are hard to decipher; see Figure 4 below for an example.



<u>Figure 4. Example of Chart</u>

The main conclusions relevant to exhausting air over a bonnet/hood are as follows:
- In general, efficiency (defined as the power expended in forcing air through the duct as a proportion of the power to pull the ducted wing through the air) is highest when (a) the inlet area is about 70% of the radiator's free area (here 64.5% of the radiator area) and (b) the duct opening was at the stagnation point.
- The quantity of cooling air through the radiator could be throttled efficiently by simultaneous variation of duct inlet and outlet areas.
- High efficiencies could be obtained with the outlet at any position on the upper surface of the wing from 25% to 75% of the wing chord (length from leading edge to trailing edge), corresponding in our case from 25% to 75% on a line from the lower bumper to the windscreen root.
- Duct inlets work better with rounded lips and the duct outlet works better with a rounded lower surface. The shape of these lips and lower surface depend on the duct shapes and openings.
- In the absence of air heating, the optimum efficiencies occur when inlet and outlet are the same size.

Analysis of the charts suggests that for an RX8 using Option C, the best compromises between efficiency and cooling power are with:
- An inlet duct throat of approximately 40-45% radiator height positioned close to the stagnation point on the lower bumper. If space is tight then a throat of 30-40% radiator height reduces drag at the expense of reduced cooling power.

- An outlet duct throat of 100-115% of the inlet throat, ie 30-55% of the radiator height.
- Inlet lip radii of 3-10% of the radiator height.
- Outlet lower surface radii of 60-85% of the radiator height.

FLOW IN DUCTING

In diffusing ducts the air sees increasing area, decreasing velocity and increasing static pressure. The increasing pressure gradient slows the boundary layer at the duct sides, which can quickly separate and become turbulent, restricting the area for fluid flow and causing drag. If we try to expand the duct too quickly we will cause significant problems for ourselves. On the other hand, a converging duct as we have at the exit of the radiator sees decreasing area, increasing velocity and decreasing static pressure; this decreasing pressure encourages the boundary layer to stay attached and we can therefore be fairly liberal with the duct shape.

Diffusing (Expanding) Ducts

The behaviour of air in diffusers/diffusing ducts depends heavily on Re, which is much lower for our ducts than that those in the majority of research. This means we avoid the problems seen in most descriptions of flow in diffusers.

Flow at Higher Re (Fully Developed Flow). At higher Re momentum forces dominate viscous forces and all diffusers have a degree of flow separation and recirculation along the diverging section, as shown in Figure 5 below.

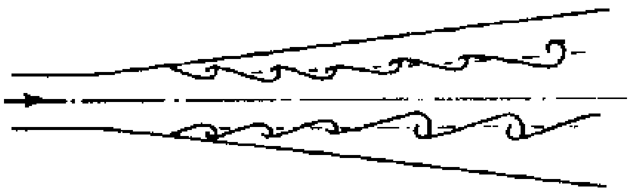


Figure 5. Diffuser Showing Boundary Layer Separation

- As the air moves along the diffuser the positive pressure gradient slows the boundary layer until the flow at the sides reverses, separates and forms unstable eddies. These regions of separation effectively reduce the downstream cross-sectional area of the diffuser, thus increasing flow velocity and reducing pressure recovery, and the eddies waste energy that is seen as drag. Regions of flow separation also cause pressure fluctuations and significantly reduce air flow through the edges of a radiator at the end of the duct.
- If the divergence angle is small then the boundary layer is relatively thin and these effects are trivial. However, if the divergence angle is large then the flow restrictions and loss of efficiency become a serious problem.
- With developed flows in pipes the optimum angle for the sides is at 6-7 degrees to the centre line in a conical diffuser and 11-13 degrees in a rectangular diffuser. Efficiency falls away rapidly above these values.
- Achieving an expansion ratio of 3 in a rectangular duct, for example when the inlet is 1/3 of the radiator height, requires a duct length of almost 3 times the outlet height, far too long for our purposes.
- Duct lengths can be shortened by using guide vanes that split the flow into a series of annular rings (pipes with a conical diffuser) as shown in Figure 6 or lower-height rectangular diffusers.

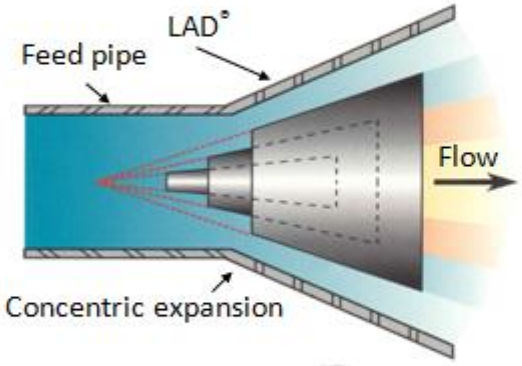


Figure 6. Guide Vanes in Conical Large Area Diffuser (LAD)

Flows at Lower Re (Start of Pipe or Duct). Fortunately for us, the behaviour of flow with low Re and laminar boundary layers is quite different because viscous forces dominate momentum forces. At low Re we can have large side angles without causing large losses. This is only possible if the air flow is stable and the diffuser has smooth rounded inlet lips to avoid triggering separation. Examples of possible diffuser profiles are shown in Figure 3 earlier and Figure 7 below.

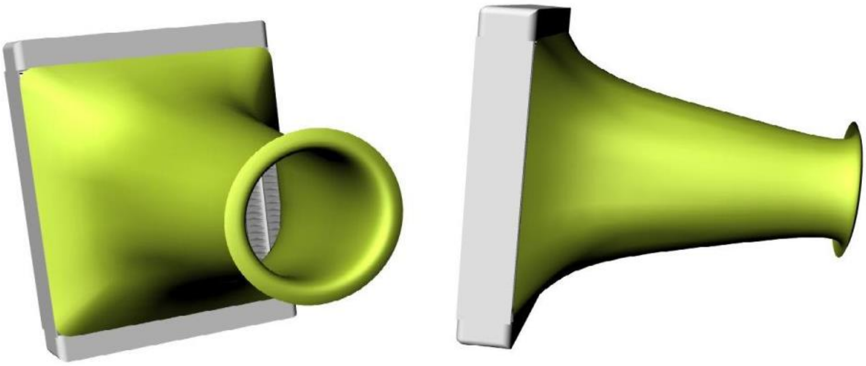


Figure 7. 3-D Representation of Ideal Duct With Circular Inlet

Effect of Radiator. The radiator will affect diffusion near the end of the duct. The airstreams will start to adjust to flow through the radiator a few cms ahead of it and pressure recovery will become less efficient. This effect can generally be ignored in the diffuser design.

Converging Ducts

As mentioned earlier, converging ducts have negative pressure gradients that help keep flow attached to the duct walls and can therefore be shaped aggressively without incurring large penalties. However, care must still be taken to avoid flow separation when bending the air flow away from the duct centreline, for example when turning exhaust air to exit a bonnet/hood, as the flow will separate from the inside surface if a turn has too narrow a radius.

FLOW IN INTERCOOLER HEADER TANKS

Intercooler header tanks need to slow down the charge air efficiently for the core then speed it up after the core. The inlet tank is a diffuser with an inlet of high-pressure, high-temperature and high-speed air that, close to the compressor outlet, will be noticeably turbulent; momentum forces will swamp viscous forces, making the flow particularly susceptible to diffuser losses. Inlet tanks therefore have to be designed carefully to avoid flow restrictions and large pressure losses. On the other hand, outlet header tanks are converging ducts with slower, cooler and denser air so will need less care in their design.

Inlet header tanks therefore need smooth curves not sharp angles. Guide vanes are also useful. Figure 8 shows the flow in a well-designed inlet tank from AWE compared to an inlet tank with sharp turns; note the flow separations on the outer surfaces, which occurs later on the lower-angle long side and immediately from the sharp edge of the guide vane. Figures 9 and 10 show the flow with inlets perpendicular to the core and various guide vanes.

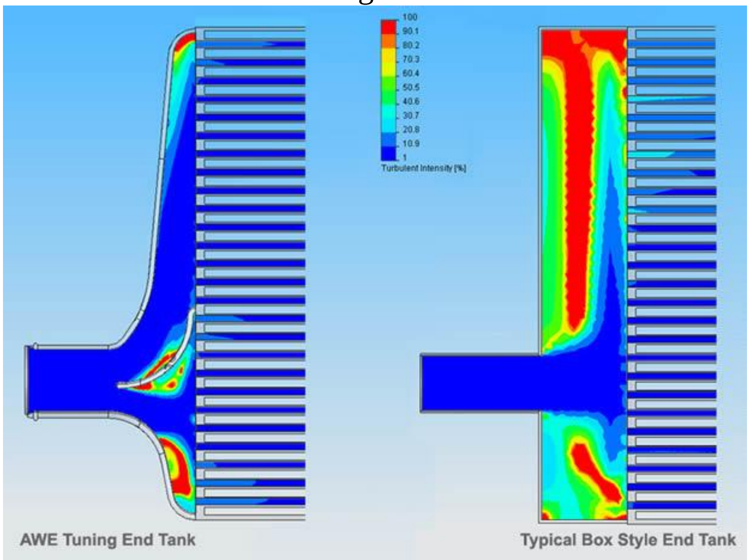


Figure 8. Flow In Well-Designed Tank and Box Tank

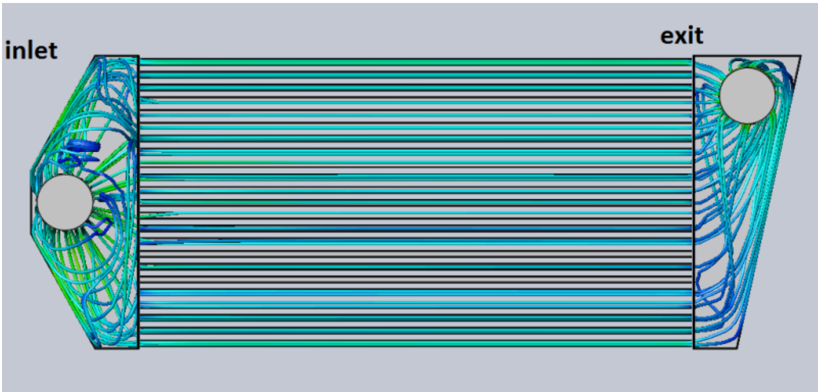


Figure 9. Flow with Pipes Perpendicular to Intercooler

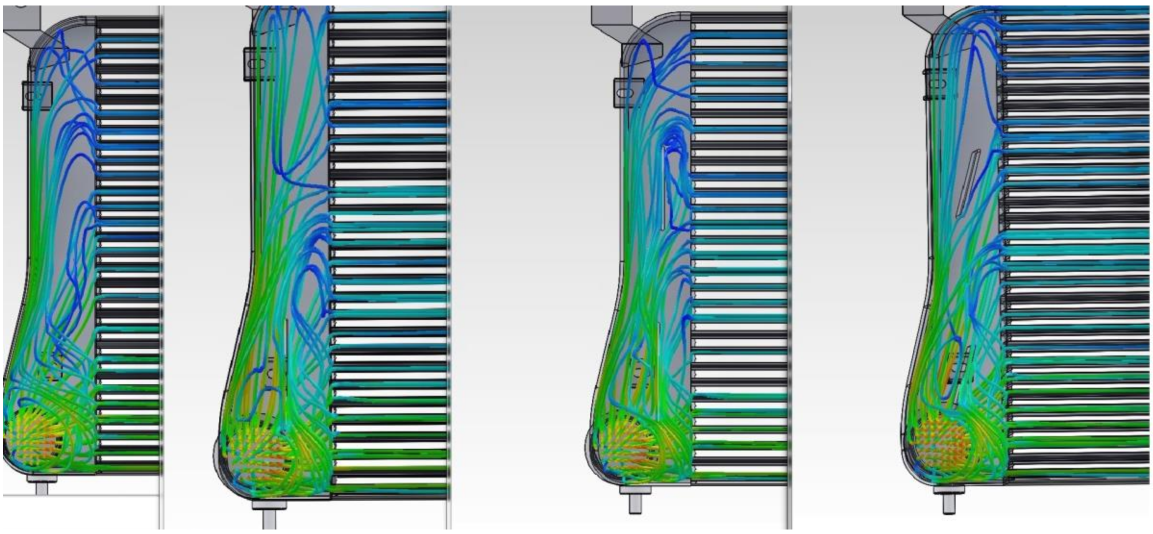


Figure 10 Flow With Different Guide Vanes

Wildman10
09 January 2016

Sources:
http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19930093263.pdf
http://naca.central.cranfield.ac.uk/reports/1942/naca-report-743.pdf