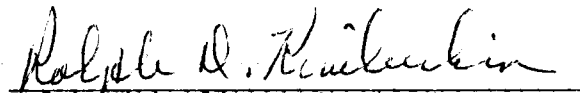
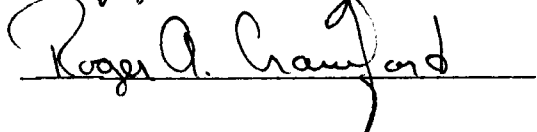


To the Graduate Council:

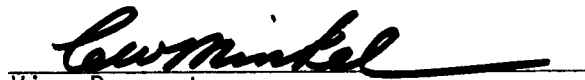
I am submitting herewith a thesis written by Russell D. Groff entitled "Liquid Cooled Engine Installations in General Aviation Aircraft: Design Considerations and Analysis of Coolant Heat Exchangers and Their Installation." I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Aerospace Engineering.


Ralph Kimberlin, Major Professor

We have read this thesis
and recommend its acceptance:

Accepted for the Council:


Vice Provost
and Dean of the Graduate School

STATEMENT OF PERMISSION TO USE

In presenting this thesis in partial fulfillment of the requirements for a Master's degree at The University of Tennessee, Knoxville, I agree that the Library shall make it available to borrowers under rules of the Library. Brief quotations from this thesis are allowable without special permission, provided that accurate knowledge of the source is made.

Permission for extensive quotation from or reproduction of this thesis may be granted by my major professor, or in his absence, by the Head of Interlibrary Services when, in the opinion of either, the proposed use of the material is for scholarly purposes. Any copying or use of the material in this thesis for financial gain shall not be allowed without my written permission.

Signature *Russell P. Goff*

Date *Aug 3 1987*

LIQUID COOLED ENGINE INSTALLATIONS IN GENERAL AVIATION
AIRCRAFT: DESIGN CONSIDERATIONS AND ANALYSIS OF
COOLANT HEAT EXCHANGERS AND THEIR INSTALLATION

A Thesis
Presented for the
Master of Science
Degree
The University of Tennessee, Knoxville

Russell D. Groff

August 1987

ACKNOWLEDGEMENTS

The author wishes to thank his committee chairman, Ralph D. Kimberlin, for his support and guidance in the preparation of this work.

The author is also indebted to the following people without whose assistance this thesis would not have been possible. The library staff of AEDC, especially Linda Love and Brenda Warren, for their invaluable assistance in obtaining research material. Mary Sue Norris for her assistance in the preliminary stages of this project, and Dena Driver for her graphical expertise.

A special thanks is due the author's wife, Mary, for her encouragement, patience and support during the preparation of this thesis.

ABSTRACT

Rising fuel costs, operating costs, and the trend to operate general aviation aircraft at altitudes above twenty thousand feet have shown deficiencies in the present air-cooled engine installations. Alternate engines and engine installations with better specific fuel consumption and lower cooling drag are needed. Liquid cooled installations could provide a more efficient general aviation aircraft.

A literature search reveals that this problem has been investigated in considerable detail by NACA and others in the 1940's. This data, however, has not been applied to general aviation aircraft. Design data and design guidelines were extracted from this data and are discussed in relation to the efficient integration of a liquid cooled engine installation in a general aviation aircraft. Design methods are developed and computerized to allow rapid sizing and evaluation of liquid cooled installations.

Results show that sharp intake lips should be avoided. The intake should be located within the propeller slipstream to take advantage of the increased dynamic pressure. The installation of the heat exchanger will be easier to accommodate in the fuselage than in the wing, although the wing location is more efficient aerodynamically. Diffusers can be designed to operate with relatively large divergence angles without separation by proper location of the heat exchanger core. An outlet flap should be provided to regulate the exit area and to provide a reduced static pressure at low speeds.

Liquid cooled installations are shown that have been designed to fit in typical general aviation aircraft space envelopes. Analysis of

these installations indicate that adequate engine cooling can be achieved for 2 percent or less of the engine shaft horsepower.

TABLE OF CONTENTS

CHAPTER		PAGE
	SECTION I. BACKGROUND INFORMATION	
I.	INTRODUCTION.	2
	Need for Improvement of General Aviation Aircraft . . .	2
	Deficiencies in Horizontally Opposed Engine Installations	2
	Use of Liquid Cooled Engines in General Aviation Aircraft.	3
	Importance of Engine-Airframe Integration to Total Airplane Drag	3
	Purpose of the Thesis	4
II.	HISTORICAL DEVELOPMENT OF LIQUID COOLING.	5
	Air-Cooling versus Liquid Cooling	5
	Development of Liquid Cooling Systems	9
	Radiator and Installation Development	16
	SECTION II. METHODS FOR DESIGN AND PERFORMANCE PREDICTION	
III.	MECHANICS OF COOLING.	22
	Method of Energy Recovery	23
	Thermal Cycle	23
	Thrust Recovery	25
	Thermal Efficiency.	27
IV.	INLET AND DUCT DESIGN	31
	Inlet Design Considerations	31
	Inlet Operation	31
	Inlet Size.	31
	Practical Intake Contours, Lips and Intake Velocity Ratio	33

CHAPTER	PAGE
Contours	35
Inlet Velocity Ratio	37
Location of the Entry	39
Boundary Layer Considerations	40
V. DIFFUSERS	41
Optimum Divergence Angles	41
Boundary Layer	43
Pressure Rise	43
Diffuser Followed by a Resistance	43
Three-Dimensional Considerations and Cross Sectional Shape	44
VI. EXIT CONSIDERATIONS	46
Flow Regulation	46
Wake Formation	46
Exit Design	47
Outlet Flaps	47
VII. INSTALLATION CONFIGURATIONS	50
Basic Types	50
Ring Installations	50
Underslung Installations	50
Wing Installations	52
VIII. HEAT EXCHANGER DESIGN	53
Mechanical Considerations	53
Heat Transfer Considerations	54
Heat Exchanger Core Sizing	54
Heat Rejection Variables	56

CHAPTER	PAGE
Heat Exchanger Surface Selection.	58
Properties of the Atmosphere.	58
Conditions at the Heat Exchanger Core Face.	61
Heat Exchanger Wall Temperature	62
Estimate of Heat Exchanger Core Frontal Area.	62
Heat Exchanger Core Pressure Drop	63
Entrance Effects.	63
Exit Effects.	64
Flow Acceleration Effects	64
Friction Losses	64
Determination of Mass Flow.	65
Overall Coefficient of Heat Transfer.	67
Heat Transfer Units and Exchanger Effectiveness	68
Internal Drag	68
External Drag	71
SECTION III. EXAMPLE INSTALLATIONS AND THE PREDICTION OF THEIR PERFORMANCE	
IX. DESIGN PROCEDURE.	74
X. EXAMPLE INSTALLATIONS	76
XI. CONCLUSIONS	91
XII. RECOMMENDATIONS	94
BIBLIOGRAPHY	95
APPENDICES	102
I. ALTITUDE CONSIDERATIONS	104
II. OPTIMUM INTAKE CONTOURS	109
III. FORTRAN PROGRAM	118

CHAPTER	PAGE
APPENDICES (Continued)	
IV. BOUNDARY LAYER CONTROL FOR VARIOUS INLET TYPES.	129
V. BOUNDARY LAYER DIVERTER DRAG.	143
VI. WING INTAKES.	146
VITA	149

LIST OF TABLES

TABLE	PAGE
I. COMPUTER PROGRAM OUTPUT FOR CLIMB CONDITION.	83
II. COMPUTER PROGRAM OUTPUT FOR HIGH SPEED CONDITION	84
III. NACA-1 ORDINATES	116

LIST OF FIGURES

FIGURE	PAGE
1. Frontal areas for a liquid cooled and air cooled radial engines of similar power output	8
2. Basic condenser cooling system	11
3. Composite cooling system.	12
4. Retractable radiator installation	15
5. Lamblin radiator.	17
6. Lamblin wing radiator	19
7. Heat exchanger duct	24
8. Temperature entropy diagram for radiator duct cooling cycle	24
9. Thrust coefficient vs. area ratio for various density ratios.	26
10. Ideal efficiency vs. area ratio for various density ratios.	28
11. Thrust force vs. aircraft velocity.	29
12. Thermal efficiency vs. aircraft velocity.	30
13. Types of inflow	32
14. Optimum contour	34
15. Pressure distribution on outer surface of optimum intake. .	34
16. Modified nose radius of NACA 1-70-030 nose inlet.	38
17. Separated flow in a diffuser.	42
18. Effect of flap angle on static pressure	49
19. Basic cooler installation	51
20. Heat rejected to coolant for Allison V1710 for various output horsepower	57
21. Heat exchanger surface geometry	59
22. Heat exchanger dimensions	77 x

FIGURE	PAGE
23. Effect of engine operating temperature on exit area required.	78
24. Effect of engine operating temperature on internal drag . .	79
25. Effect of diffuser efficiency on exit area.	80
26. Effect of diffuser efficiency on internal drag.	81
27. Inlet area sizing curve	85
28. 150 HP wankle installation ring cooler side view.	86
29. 150 HP wankle installation ring cooler plan view.	87
30. 150 HP wankle installation, underslung installation side view	88
31. 150 HP wankle installation, underslung installation front view.	89
32. Altitude effect on the relative mass of required cooling air	106
33. Altitude effects on the required volume of cooling air. . .	106
34. Effect of altitude and losses on required pressure drop . .	107
35. Ratio of corrected to uncorrected values as a function of altitude	107
36. Thrust forces on intakes.	111
37. Kuchemann series inlet geometry	113
38. NACA-1 nomenclature	115
39. Guide vane installation	130
40. Boundary layer bypass	130
41. Boundary layer diverter	132
42. Flush slot nomenclature	135
43. Submerged slot.	137
44. Convergent and divergent slot	138
45. NACA submerged inlet.	140
	xi

FIGURE	PAGE
46. Boundary layer diverter drag variation with free stream Mach no.	140

NOMENCLATURE

A_{BO}	Heat Exchanger Frontal Area (ft^2)
A_E	Exit Area Required (ft^2)
A_f	Core Frontal Area (ft^2)
A	Air Side Heat Transfer Area (ft^2), Total Exchanger Heat Transfer Area On One Side (ft^2)
A_C	Cowl Frontal Area, Exchanger Minimum Free Flow Area (ft^2)
A_S	Wing Planform Area, Projected Frontal Area of Scoop (ft^2)
A_m	Projected frontal area of the scoop (ft^2)
A_2	Area at Station 2 (ft^2)
A_3	Area at Station 3 (ft^2)
C_p	Specific Heat of Air at Constant Pressure
C_D	Drag Coefficient
C	Chord Length
C_{Ds}	Drag Coefficient Based on Projected Frontal Area
D	Hydraulic Diameter of the Passage
E	Included Wedge Angle (Deg) of Boundary Layer Diverter
F_H	Normal Force
F_C	Compressibility Correction Factor
f	Friction Factor
g, g_c	Gravitational Constant
G	Air Stream Mass Velocity

h	Height of Boundary Layer Diverter, Inside Height of Flush Slot
h_a	Unit Film Conductance Air Side
h_w	Unit Film Conductance Coolant Side
H	Rate of Dissipation of Heat Energy (ft-lbs/sec)
k_c	Loss Coefficient at the Entrance
k_e	Loss Coefficient at the Exit
K	Constant $(T/32.2(778.0)C_p)$
L	Length of Tube
M	Mach Number
n	Number of Tubes
P_s	Drag Power (HP)
P	Power Absorbed (HP)
P_c	Power Consumed (HP)
P_1	Pressure Ahead of the Core
ΔP	Change in Pressure (Pressure Drop)
P_t	Total Pressure
Q	Volumetric Flow Rate (Ft^3/sec), Heat Rejected to Coolant (ft/sec)
$q_o = 1/2 \rho_o V_o^2$	Flight Dynamic Pressure
R_e, R_x, R_N	Reynolds Number
T	Thrust
T_w	Wall Temperature
T_B	Air Inlet Temperature

T_0	Total Temperature at Station 0
T_2	Total Temperature at Station 2
T_w	Wall Temperature of Heat Exchanger
T_{B_i}	Air Inlet Temperature (temperature at the face of the heat exchanger)
U_a	Overall Coefficient of Heat Transfer
V	Velocity of Flow in Tube
V_o^*	Flight Speed at Which Heat Exchanger is Evaluated
V_i	Internal Velocity
V_1	Flight Velocity (Ft/sec)
V_o	Flight Velocity (ft/sec)
V_e, V_4	Exit Velocity (ft/sec)
W	Weight Flow (lbs/sec)
X	Longitudinal Distance (ft)
α_w	Ratio of Total Transfer Area on Coolant Side of the Exchanger to Total Volume of the Exchanger (Surface Type Dependent)
α_a	Ratio of Total Transfer Area on Air Side of the Exchanger to Total Volume of the Exchanger (Surface Type Dependent)
η	Efficiency η_o , Thermal, or Mechanical
η_o	Overall Surface Effectiveness
η_f	Fin Effectiveness
σ	Ratio of Total Cross-Sectional Area to Free Flow Area
ρ_1	Density of Unheated Air
ρ_2	Density of Heated Air

v_1	Specific Volume of the Air at the Core Entrance
v_2	Specific Volume of the Air at the Core Exit
v_m	Mean Specific Volume
μ	Viscosity of Air (Slugs/ft-sec)
T	Coefficient of Thermal Expansion of Air at Constant Pressure

SECTION I. BACKGROUND INFORMATION

CHAPTER I

INTRODUCTION

Need for Improvement of General Aviation Aircraft

Rising fuel costs, operating costs, and the trend toward higher operating altitudes, i.e., above 20,000 feet, will require future general aviation aircraft to be more efficient than their predecessors. Areas of general aviation design that promise large gains in performance are the improvement of internal combustion engines and the efficient integration of the power plant into the airframe, specifically an improvement in specific fuel consumption and the reduction of the installed cooling drag.

Deficiencies in Horizontally Opposed Engine Installations

General aviation aircraft designs have been dominated by the horizontally opposed air-cooled piston engine installation since the 1940's. The relative simplicity and light weight made the installation of these engines particularly attractive; however, after more than 40 years of use, the efficient operation and cooling of the horizontally opposed installation appears to be difficult to achieve. Studies by Miley, et al., (Ref. 43, 44) have indicated that (1) general aviation aircraft are still subject to insufficient cooling, leading to increased fuel consumption and reduced performance; (2) the use of supercharging, while increasing the available horsepower, also increases the heat rejection and increases significantly the difficulty of cooling with increased operational altitude; (3) the horizontally

opposed engine layout does not lend itself well to efficient cooling aerodynamics; and (4) engine manufacturers do not supply adequate cooling data. Miley (Ref. 44) and Ellerbrocke (Ref. 17) indicate that significant modifications are needed to improve engine performance and reduce the cooling drag.

Use of Liquid Cooled Engines in General Aviation Aircraft

Various studies by NASA (Ref. 8, 9, 28, 72, 79), Continental Engines (Ref. 59), and others (Ref. 2) indicate that the specific fuel consumption of general aviation power plants could be improved significantly. The most attractive methods of achieving this appear to be (1) changing from air-cooled piston engines to liquid cooled piston engines; (2) the utilization of a different type of engine such as the stratified charge wankle engines. Although the specific fuel consumption of existing horizontally opposed piston engines could be improved through stratified charge, better head design, variable timing, etc., the already marginal cooling would assume greater proportions; therefore, this approach does not appear particularly attractive.

Importance of Engine-Airframe Integration to Total Airplane Drag

Wind tunnel studies of various aircraft (Ref. 10) indicate that the largest single contributor to drag is the power plant installation. These studies indicated that the power plant installation increased the drag as much as 46 percent over the sealed and faired condition. All other items combined (canopies, antenna, etc.) contributed 19 percent of the total drag. Attention to detail in the engine

installation could decrease the power plant contribution from 46 percent to 26 percent.

Purpose of the Thesis

The installation of a wankle or liquid cooled piston engine in general aviation aircraft could lead to an improvement in performance over present aircraft; however, care must be used in the design of these installations. If proper attention is not paid to the cooling drag, any advantage in specific fuel consumption will be lost.

Review of studies such as that of Cessna Aircraft (Ref. 28) and Beech Aircraft (Ref. 79) indicates that lessons of the past are being ignored and that a means of predicting the aerodynamic performance of liquid cooled engine installations needs to be developed.

CHAPTER II

HISTORICAL DEVELOPMENT OF LIQUID COOLING

Air-cooling versus Liquid Cooling

It has been said that "water cooling an aeroplane engine makes about as much sense as air cooling a submarine" (Ref. 22). It appears that there has always existed two views on the proper method of cooling aircraft internal combustion engines. Both approaches have advantages and disadvantages.

The proponents of air cooling have claimed lower installed weight, better reliability, and simpler installation. The development of transport aircraft in the 1920's and 1930's was based largely on the development of air-cooled radial engines. The installations of that period did have a lower installed weight than the equivalent liquid cooled installation. Since this significantly improved the payload of these aircraft, much emphasis was put on developing the air-cooled radial.

The United States Navy initiated much of the air-cooled research in these early days. Overwater operations demanded a reliable power plant and shipboard operations dictated ease in maintenance. Experience with liquid cooled engines indicated that the reliability was inferior to the air-cooled radial and that maintaining the cooling system and coolant were significant problems aboard ship.

The liquid cooled engine was therefore not pursued with the same enthusiasm as the air-cooled engine in the United States. This was not the case however in Europe, especially Great Britain. The

Rolls Royce firm in particular spent considerable time and money on the development of an efficient liquid cooled engine and its installation. The Rolls Royce "R" engine developed in 1929 for the Supermarine S6B (Britain's Schneider Cup entry) developed 2650 HP for a weight of 1400 lbs. This power to weight ratio was not exceeded until the arrival of gas turbines in the late 1940's.

The liquid cooled engine has certain inherent advantages over the air-cooled engine. The first of these is that when operating at relatively high altitudes, i.e., above 20,000 feet with supercharging, the liquid cooled engine can maintain an even cylinder bore and head temperature. This eliminates hot spots on the cylinders and allows the engine to operate at high power settings without detonation.

Air-cooled engines on the other hand encounter considerable difficulties operating at high altitudes (see Appendix 1). As the altitude increases, the density of the atmosphere decreases at a faster rate than the temperature difference between the air and the cylinder cooling fins. Therefore the cooling fin area is not sufficient at the operating altitude to transfer the same amount of heat that could be transferred at lower altitudes. This causes excessive operating temperatures and, therefore, a reduction in power settings at altitude.

As a result at high altitudes or in conditions of marginal cooling, i.e., climbing, air-cooled engines are run with an excessively rich mixture. This uses the excess fuel to transport some heat from the cylinder. This allows the engine to operate at

high power settings, but drastically increases the specific fuel consumption.

Liquid cooled engines, cooled by a heat exchanger, do not encounter this limitation. Since the heat exchanger can be incorporated into the airframe, the size can be increased without great difficulty. Also, since the heat exchanger wall temperature is much cooler than the fins of an air-cooled engine, the rate of change of the temperature difference between the heat exchanger wall and the atmospheric temperature largely compensates for the decrease in the air density.

The liquid cooled engine also excels in the ratio of power to frontal area. The use of liquid cooling allows the use of a compact cylinder arrangement while still providing effective cooling. A comparison of frontal areas for a liquid cooled engine and a radial air-cooled engine of similar power output are shown in Figure 1.

"The simple fact is, liquid cooled aeroplanes held the world speed record (piston engine) for a full half a century, and until 1969, no fixed aircooled engine ever held the speed record!" (Ref. 22). It is interesting to note that once again the speed record (piston engine) is held by a liquid cooled engine powered aircraft (The Red Baron, a Griffon powered modified P-51).

Development of liquid cooled engines occurred between the years of 1920 through 1945. Development was fairly rapid from 1920 to 1930, after 1930 the effort receded to some extent until the late 1930's and early 1940's. Then World War II forced the development of high powered aircraft that demanded maximum streamlining. The development continued

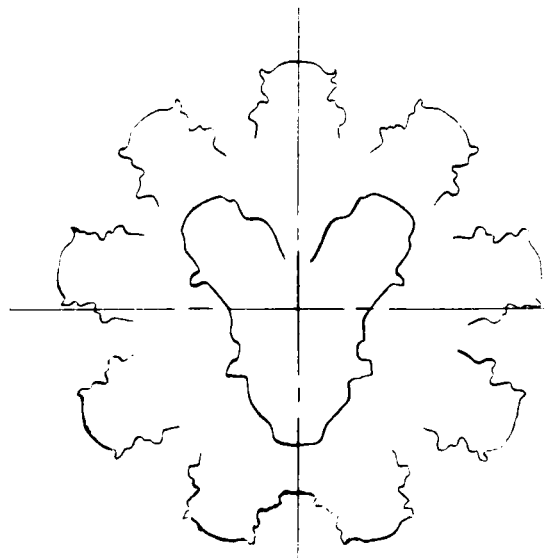


Figure 1. Frontal areas for a liquid cooled and air cooled radial engines of similar power output.

until the end of the war in 1945. The introduction of the gas turbine made the installation of liquid cooled engines in frontline military aircraft obsolete.

Development of Liquid Cooling Systems

All liquid cooling systems, regardless of the type of coolant, require a circulation of the coolant through the cooling passages of the engine where rejected heat is transferred to the coolant. The coolant is then pumped (by various means) through a heat exchanger where the heat is rejected to the atmosphere.

The first type of cooling system developed was the thermosiphon system. This type of cooling system is found on many of the early automobiles. It was, therefore, natural to try and extend its use to aircraft. Its absence of mechanical parts and low maintenance made it look particularly attractive from a mechanical point of view. This system does not use a mechanical circular pump, but relies on sufficient temperature gradient to promote convective currents in the system. These convection currents provide the pressure differential to circulate the coolant. This type of system was never successful in aircraft applications. The main drawback being that a sufficient temperature gradient could not be maintained at high power settings. This led to many of these systems being modified with the addition of a mechanical circulation pump and, in turn, developed into the condenser type system.

The condenser system requires that the coolant operate at a constant temperature equal to the boiling point. The coolant in the engine coolant passages is permitted to boil, the vapor (steam in

the case of water) is ducted through a heat exchanger (radiator) where it is allowed to condense. The condensate is then pumped back into the engine cooling passages and the cycle repeated.

The first form of this system is shown in Figure 2, and the basics are retained in most liquid cooled aircraft cooling systems. This figure also shows one other important advantage of liquid cooling. Since the heat is transferred from the engine to the liquid coolant, the heat can be transported to the heat exchanger (radiator) which can be located nearly any place on the airframe. In this case, the radiator is shown located away from the propeller gearbox for cleaner aerodynamics. This system also incorporates a circulation pump and header tank. Temperature control is achieved by placing shutters in front of the radiator that are controlled from the cockpit.

The major drawback of this system was that the coolant (in this case water) was allowed to boil off. Only part of the coolant was reclaimed and recirculated. This obviously limits flight time.

The system, however, was quite efficient from a thermodynamic standpoint. Allowing the coolant to boil in the engine cooling passages permits more heat per unit of coolant to be transferred by using the latent heat of vaporization.

A development of this basic system is composite cooling and is shown in Figure 3. In this system, steam condensers were fitted along the leading edge of the wings. The coolant was maintained in the liquid form for most normal operations. The system was sized so that boiling (and boil off of the coolant) was permitted at the high power settings.

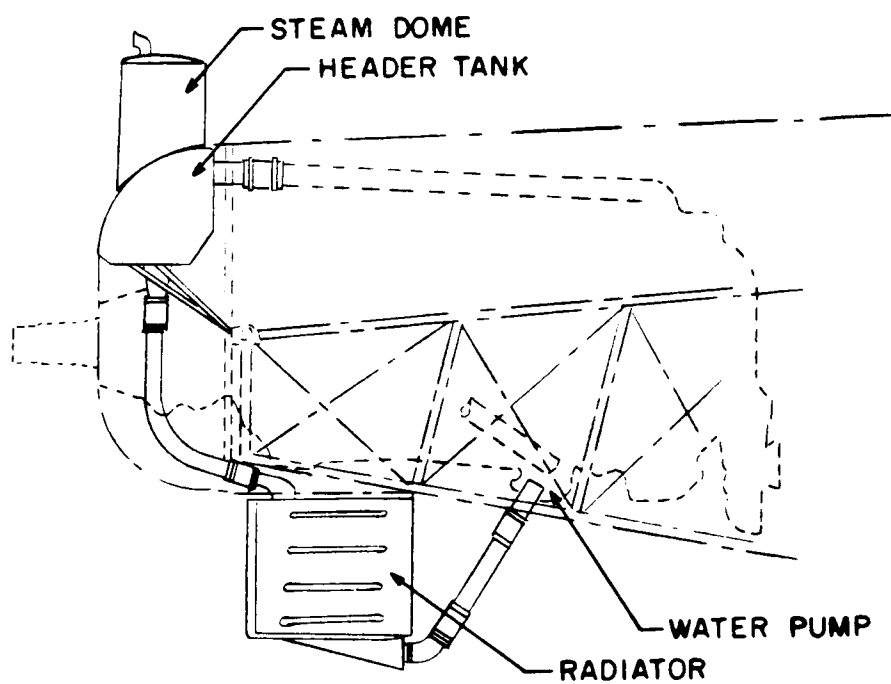


Figure 2. Basic condenser cooling system.

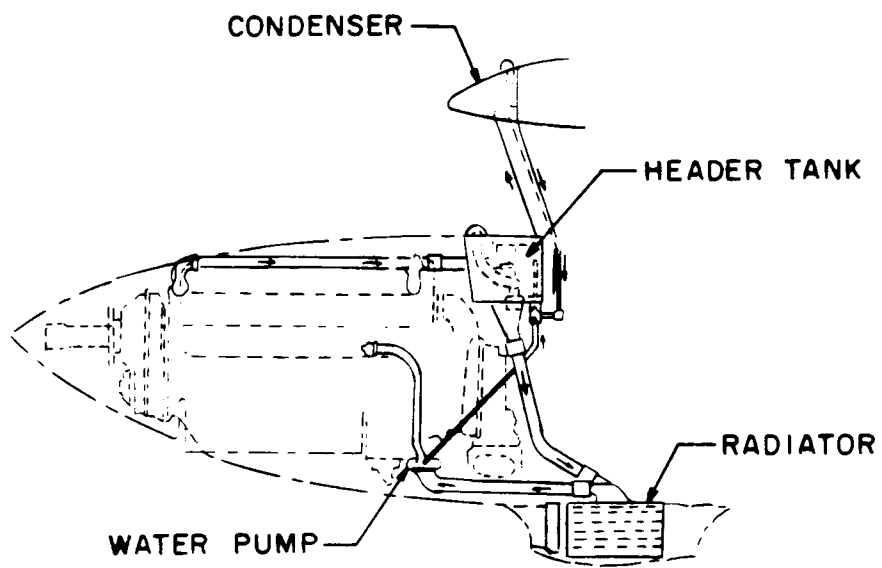


Figure 3. Composite cooling system.

The main disadvantages of this system were excessive weight and a design layout which could only be incorporated in biplanes. With the introduction of the low wing monoplane, certain deficiencies in this approach became readily apparent. Since the condensers were still incorporated in the leading edge of the wing, the condensers were below the header tank and the condensate had to be pumped up to the header tank. The condensate, being essentially at the boiling point, entered the pump at saturation conditions. The reduced pressure in the pump inlet caused the condensate to flash to steam, thus cavitating the pump. This cavitation resulted in the loss of circulation and the composite cooling approach was abandoned.

The final form of the liquid coolant system is the forced circulation pressurized system. The forced circulation system was developed directly from the condenser system. The coolant, however, was changed from water to ethylene glycol.

The introduction of ethylene glycol overcame the boiling problem associated with water cooled systems. Ethylene glycol has a boiling point of 383 °F and a freezing point of 0 °F. The higher boiling point had two main advantages. One is that the operation at higher coolant temperatures allowed the engine to operate at higher temperatures, thus increasing its thermodynamic efficiency and perhaps more important, the higher operating temperatures allowed a significant reduction in radiator size. Since the rate of heat transfer is proportional to the temperature differences between the radiator wall and the ambient air, operation at higher wall

temperatures meant that for the same amount of heat transfer, less area was required.

The first glycol systems used 100 percent glycol, and the disadvantages of using 100 percent glycol soon became apparent. Glycol has a specific heat of 0.62 that of water. The substitution of glycol in a system originally designed for water meant that the engine, designed for water cooling, ran approximately 70° hotter. This significantly reduced the life of some engine components and at the extreme required either a redesign of these components or a complete redesign of the engine. Other disadvantages were an affinity for rubber, its power of penetration, and its flammability. It would dissolve rubber connections and penetrate through castings and joints that were "water tight." It had been tried in 1929 by Rolls Royce on the 1929 Schnieder entry, but was abandoned because it would leak from the riveted joints in the surface radiators. The use of 100 percent glycol systems were then abandoned.

Engine studies at this time indicated that engine life could be considerably improved, if the engine could be cooled with water. Investigations of radiator installations--retractable radiators, Figure 4--indicated that with proper design the larger radiator using water could be designed to give no more drag than the smaller glycol radiators. The final form of the coolant consisted of 70 percent water with 30 percent glycol added primarily as an antifreeze.

The development of pressurized systems was a result of operating aircraft in a tropical environment. These aircraft normally operated at higher temperatures than aircraft in other areas. As a result,

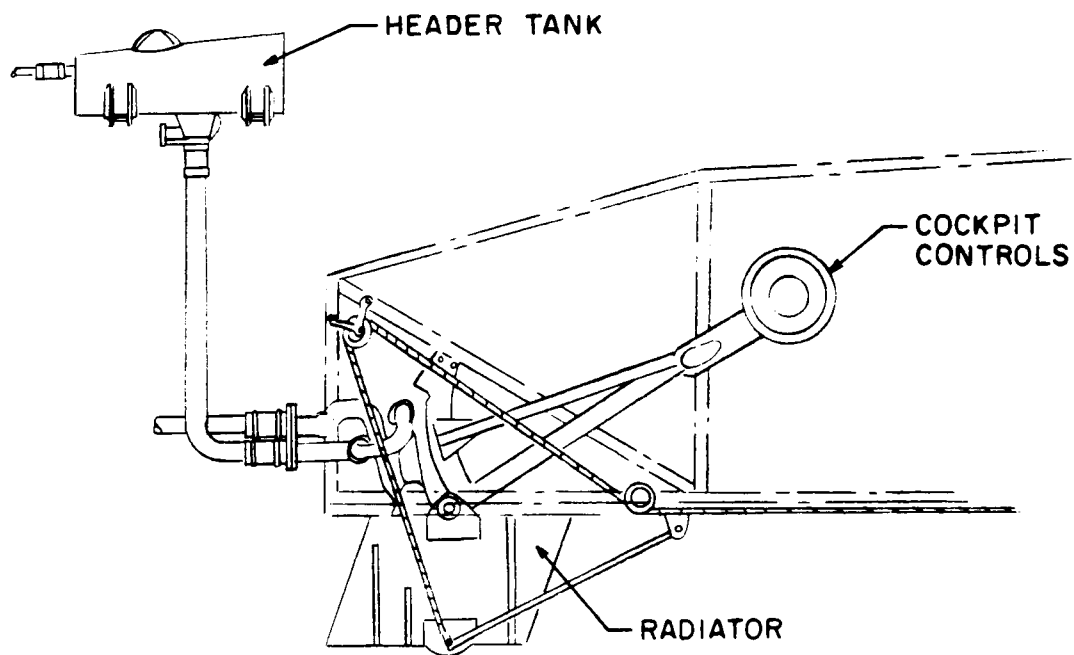


Figure 4. Retractable radiator installation.

pressurized systems were investigated. Since the boiling point of water can be raised by operating under pressure, the use of pressurized systems allowed the aircraft to be operated in the tropics without increasing the radiator size.

Radiator and Installation Development

Radiators of early aircraft were usually of honeycomb construction, used water as the cooling medium, and invariably were located in the nose of the aircraft. The prevailing theory until the 1930's was that the radiator drag could be reduced to a minimum by using the smallest radiator cross section located such that it would receive air at the free stream velocity; therefore, nose and side positions prevailed.

The Germans during World War I tried using radiators conforming to the wing center section contour on some fighter aircraft. This attempt was not particularly successful. The construction was not particularly durable, not easily repairable, and hard to produce. These drawbacks outweighed the slight improvement in aerodynamics; therefore, this approach was abandoned (Ref. 22).

The first real development came from a French designer, A. Lamblin, in 1918. Lamblin's design utilized copper fins mounted radially in which the water coolant was circulated. This is shown in Figure 5 (Ref 22). The advantage of this design was the provision of a large heat transfer area with a minimum of frontal area. The ratio of the radiating surface to the frontal area was on the order of 100:1. It was the forerunner of what is called today, compact heat transfer surfaces. The use of this design reduced the drag of

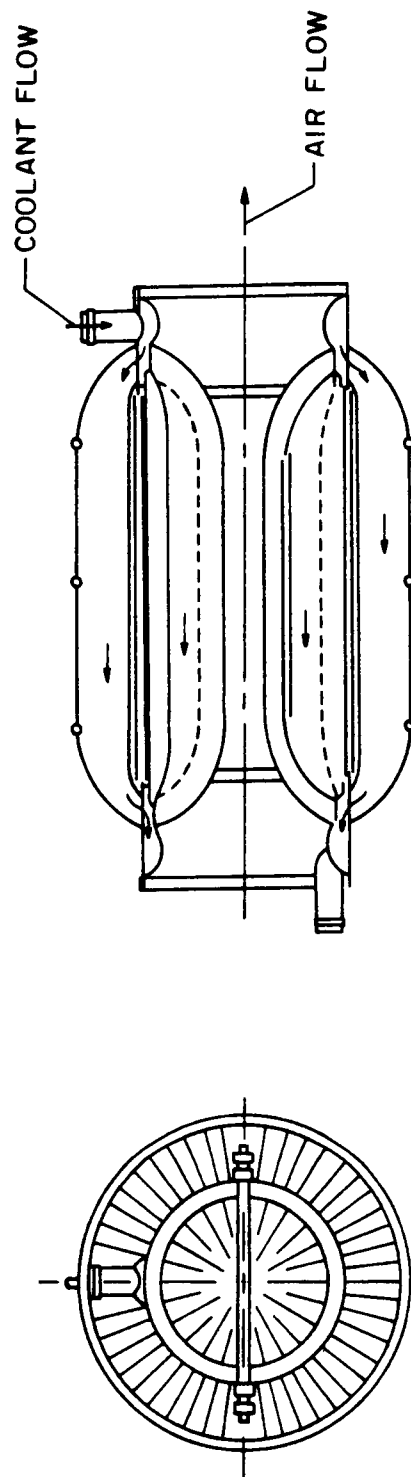


Figure 5. Lamblin radiator.

radiator installations considerably, while providing easy maintenance and replacement.

In 1920 Curtiss designed what was then considered the ultimate radiator (perhaps even today). This design was the surface radiator. The goal was to use the skin of the wings (or other parts of the aircraft) as the heat transfer surface. The advantage, of course, being the transfer of heat without incurring any drag penalty. The success of the R3C-4 Curtiss racers proved the aerodynamic advantages of this approach and also demonstrated its reliability.

The French in 1924 developed a version of the Lamblin concept that was somewhere between the surface radiator and the original Lamblin. Essentially, it was a Lamblin radiator cut longitudinally and unrolled (Figure 6). This radiator was then mounted on the undersurface of the wing (Ref. 22). This approach proved to be inferior to the Curtiss scheme and was dropped.

Surface radiators were tried on operational aircraft but were too delicate for day-to-day operations. The Curtiss PW-8 Hawk attempted operation with surface radiators in 1924. Pressure changes induced in the system from operation at sea level to 25,000 feet led to fatiguing of the metal and development of cracks. This concept was attempted again with modifications in the early 1940's on the Heinkel HE-100. The biggest drawback of this system was considered to be its vulnerability to gunfire in combat.

In the 1930's, Rolls Royce became concerned that the development of radiators was not keeping pace with the development of engines, and undertook development of lighter, more efficient radiators and

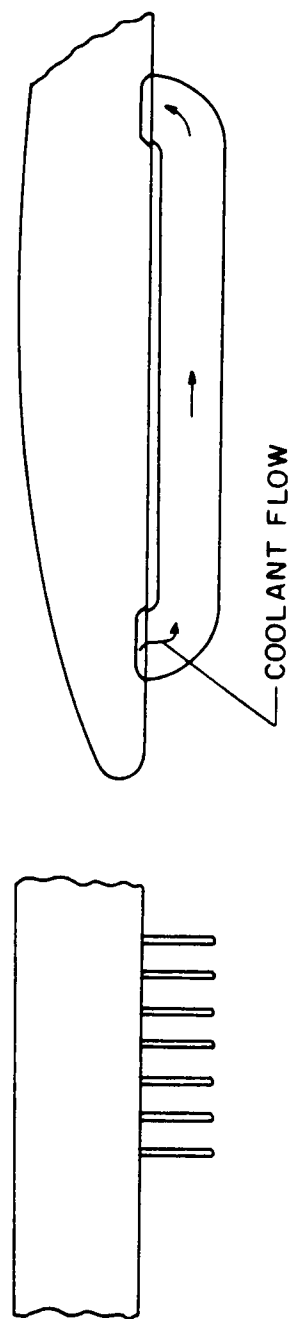


Figure 6. Lamblin wing radiator.

installations. The first attempts were aimed at improving the honeycomb tube layout. After two years of development, it was found that the tube diameter should be reduced from the standard 7 mm diameter to 5 mm. This required an increase in frontal area by about four percent for the same free flow area, but the thickness could be reduced by about 15 percent, which resulted in a 19 percent reduction in weight. The reliability was substantially improved by the substitution of copper for the standard brass construction. In 1937, Rolls Royce had concluded that they had improved the honeycomb radiator about as far as it reasonably could go and that a new approach was needed.

The new approach was found in the secondary surface radiator used in the automobile and refrigeration industries. Experiments indicated that by suitable modifications, such as increasing the depth of the coolant tubes and decreasing their number, a satisfactory design could be achieved. Using the same depth as a honeycomb radiator, the frontal area could be decreased by 10 percent and the weight, with coolant, could be decreased by 38 percent. This approach proved to be an efficient method of cooling liquid cooled engines, and forms the basis of the following design analysis.

SECTION II. METHODS FOR DESIGN AND PERFORMANCE PREDICTION

CHAPTER III

MECHANICS OF COOLING

The engine cooling problem is approached from a generalized energy balance of an engine installation. The internal combustion engine is at best approximately 50 percent efficient, i.e., only 50 percent of the heat energy released by the burning of the fuel is recovered as mechanical work. The remaining 50 percent of the energy is transported from the engine through the heat and kinetic energy of the exhaust, heat transferred to the lubricating oil, heat transferred to the cooling airflow (through finned cylinders or liquid heat exchangers), conduction heat transfer to the mounting structure and radiative heat transfer. The conduction and radiative heat transfer can be ignored with negligible error in most engine installations, since the energy transferred through these mechanisms is many orders of magnitude smaller than the other mechanisms.

This quantitative heat balance does not describe the entire process, however. The installation of the engine in an airframe requires that some of the mechanical shaft energy, used for propulsion, be used to dissipate the rejected heat (drag). If the installation (engine-airframe integration) is not given careful consideration, a disproportionate amount of shaft energy will be used to dissipate the rejected heat. This shaft energy loss will result in performance penalties. The cooling problem, thus, becomes one of attempting to recover as much of the dissipated heat as possible to offset the cooling power loss.

Method of Energy Recovery

Meredith (Ref. 41) appears to be the first to suggest that the heat energy from the cooling medium and exhaust could be recovered to offset the drag produced by the engine/cooling installation.

Meredith's proposal suggested that if a heat exchanger (i.e., either an air-cooled engine or in the liquid cooled case, a separate radiator) could be enclosed in a properly shaped duct, a propulsive force could be developed, offsetting the internal and external drag of the installation and possibly producing a net thrust. This process describes a very low speed ram jet engine.

Thermal Cycle

The heat exchanger is considered to be located in a duct as shown in Figure 7. It is assumed that the compression and expansion processes are reversible, and adiabatic, and that the heat transfer from the heat exchanger takes place at a constant pressure. This, of course, is an ideal cycle. The actual cycle would have to consider losses. The assumptions, however, allow a prediction of maximum performance.

The thermodynamic cycle of the duct is shown in the temperature-entropy diagram of Figure 8. The air is slowed and isentropically compressed from the free stream conditions of position 1 to the stagnation state 02 at station 2. Heat is then added at constant pressure between states 02 and 03, between stations 2 and 3. The exit nozzle expands the stream isentropically to the ambient pressure at station 4. Note that the isentropic expansion requires that the exit pressure equals the ambient pressure.

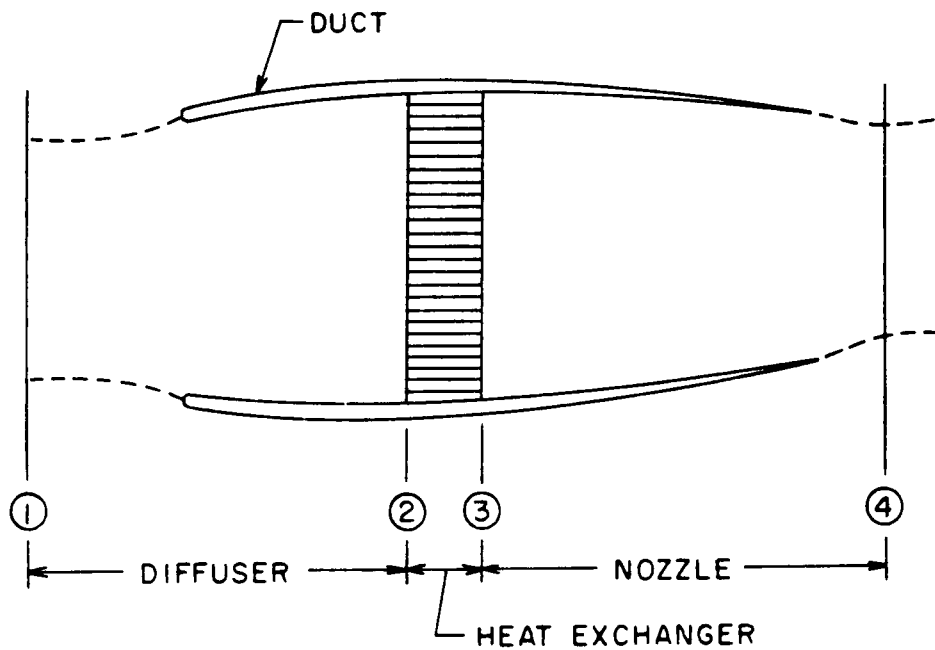


Figure 7. Heat exchanger duct.

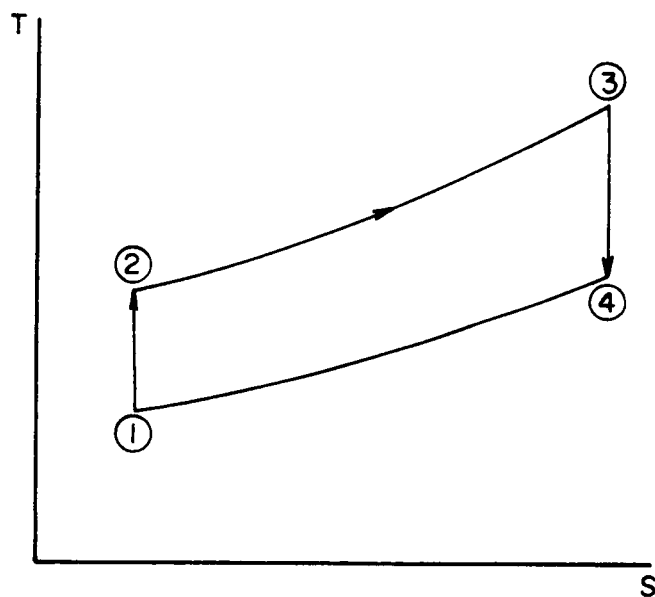


Figure 8. Temperature entropy diagram for radiator duct cooling cycle.

Thrust Recovery

The object of the duct design is to recover as much heat as possible as thrust. Using the theoretical cycle described, it is possible to determine certain constraints of the duct design and the maximum thrust that can be developed.

The thrust equation for the ideal cycle in terms of density ratio and area ratio is given by: (Ref. 58)

$$T = \rho_1 A_1 V_1^2 \left[\sqrt{\rho_1/\rho_2 \left[1 + \left(A_1/A_2 \right)^2 \right]} - \left(\rho_1/\rho_2 A_1/A_2 \right)^2 - 1 \right]$$

The subscripts refer to the station numbers of Figure 7.

The entrance velocity V_1 and the entrance density ρ_1 are given by the flight conditions. The density ρ_2 can be determined by the temperature rise through the heat exchanger. The temperature rise and the rate of heat transfer determine the mass flow through the heat exchanger, and thus determines A_1 . A plot of the thrust coefficient for various area ratios is shown in Figure 9.

These curves indicate that a positive thrust is developed only when the area ratio A_1/A_2 is less than one, i.e., the entrance area should always be less than the exchanger core frontal area. The maximum thrust is developed when the area ratio is zero, i.e., the exchanger frontal area is infinite; obviously in practice, this is not possible. The plots indicate that for reasonable thrust recovery, the area ratio should be between 0.2 and 0.3. The curves tend to be flat in this region for the lower density ratios.

The thrust equation also shows that the thrust is proportional to the square of the velocity. At low speeds one cannot expect the

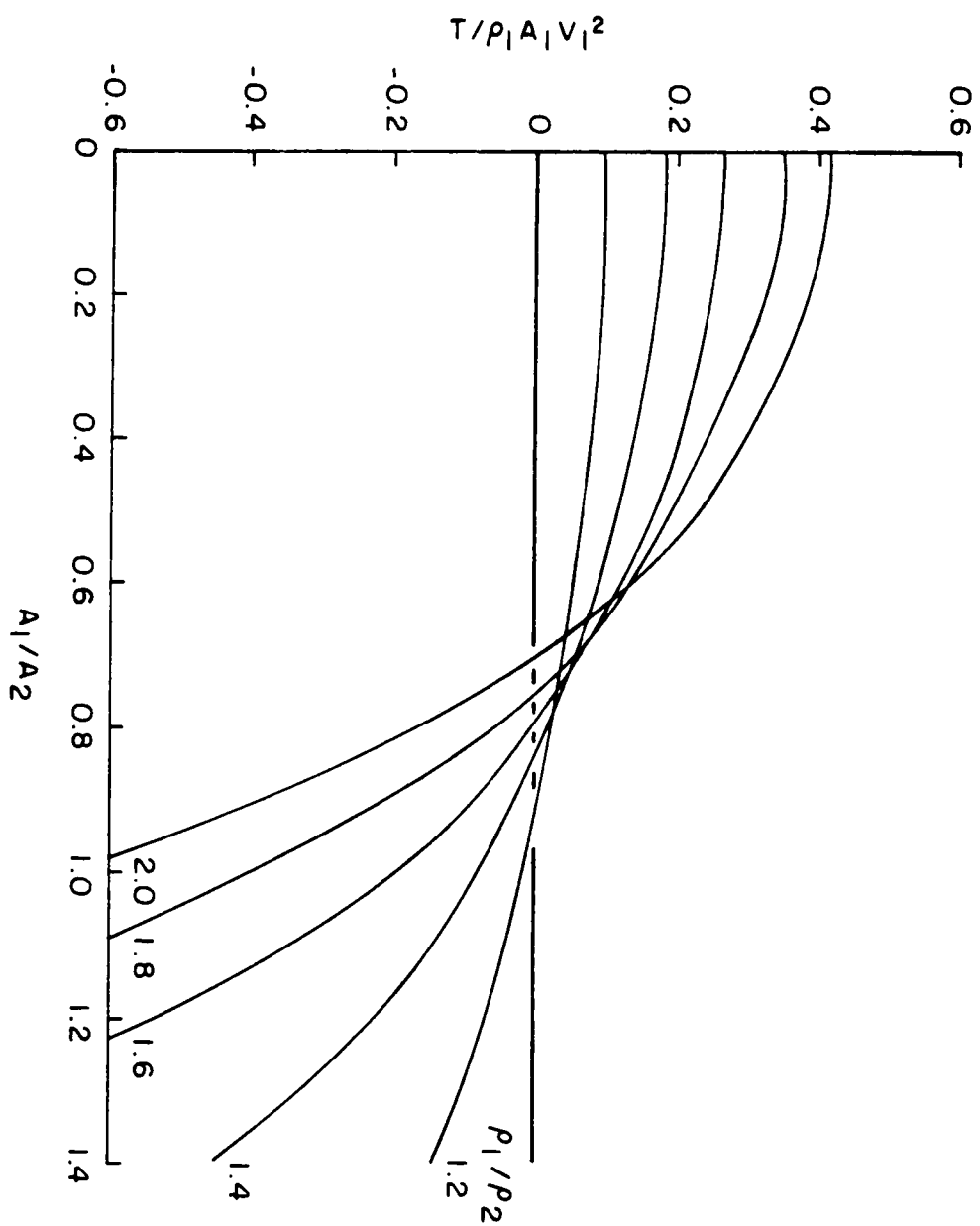


Figure 9. Thrust coefficient vs. area ratio for various density ratios.

thrust to be high. A plot of thrust vs. aircraft velocity for typical general aviation flight conditions is shown in Figure 10.

Thermal Efficiency

The thermal efficiency in this case indicates the efficiency with which the heat energy (dissipated by the heat exchanger) can be converted into mechanical work.

This is given by:

$$N_{th} = TV_1/H$$

The thermal efficiency is given in terms of area ratio and density ratio as: (Ref. 58)

$$\eta = \frac{V_1^2}{k} \left(\frac{\rho_1/\rho_2 \sqrt{\rho_1/\rho_2 \left[1 + \left(A_3/A_2 \right)^2 \right] - \left(A_3/A_2 \right)^2}}{\left[\rho_1/\rho_2 - 1 \right] \sqrt{\rho_1/\rho_2 \left[1 + \left(A_3/A_2 \right)^2 \right] - \left(A_3/A_2 \right)^2}} \right)$$

A plot of this equation for various density ratios and area ratios is shown in Figure 11. This curve also indicates that the maximum efficiency is achieved with an infinite core area. The curves show that for area ratios 3 or 4 (as before), there is little loss in efficiency from that of an infinite core. From the equation, the efficiency is proportional to the square of the velocity. Thus, the efficiencies at low velocities will be very low. A plot of efficiency vs. aircraft velocity for typical general aviation aircraft is shown in Figure 12.

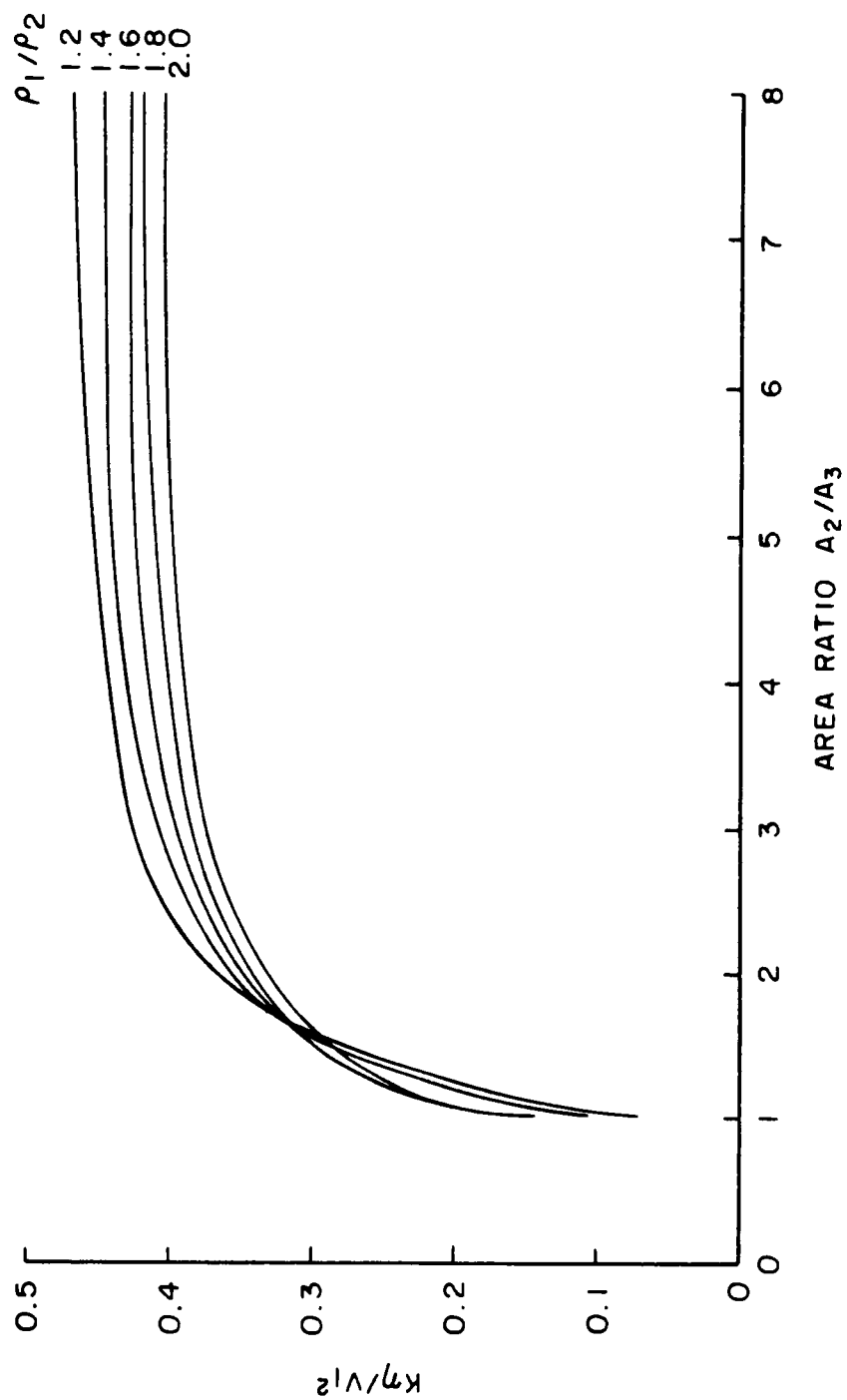


Figure 10. Ideal efficiency vs. area ratio for various density ratios.

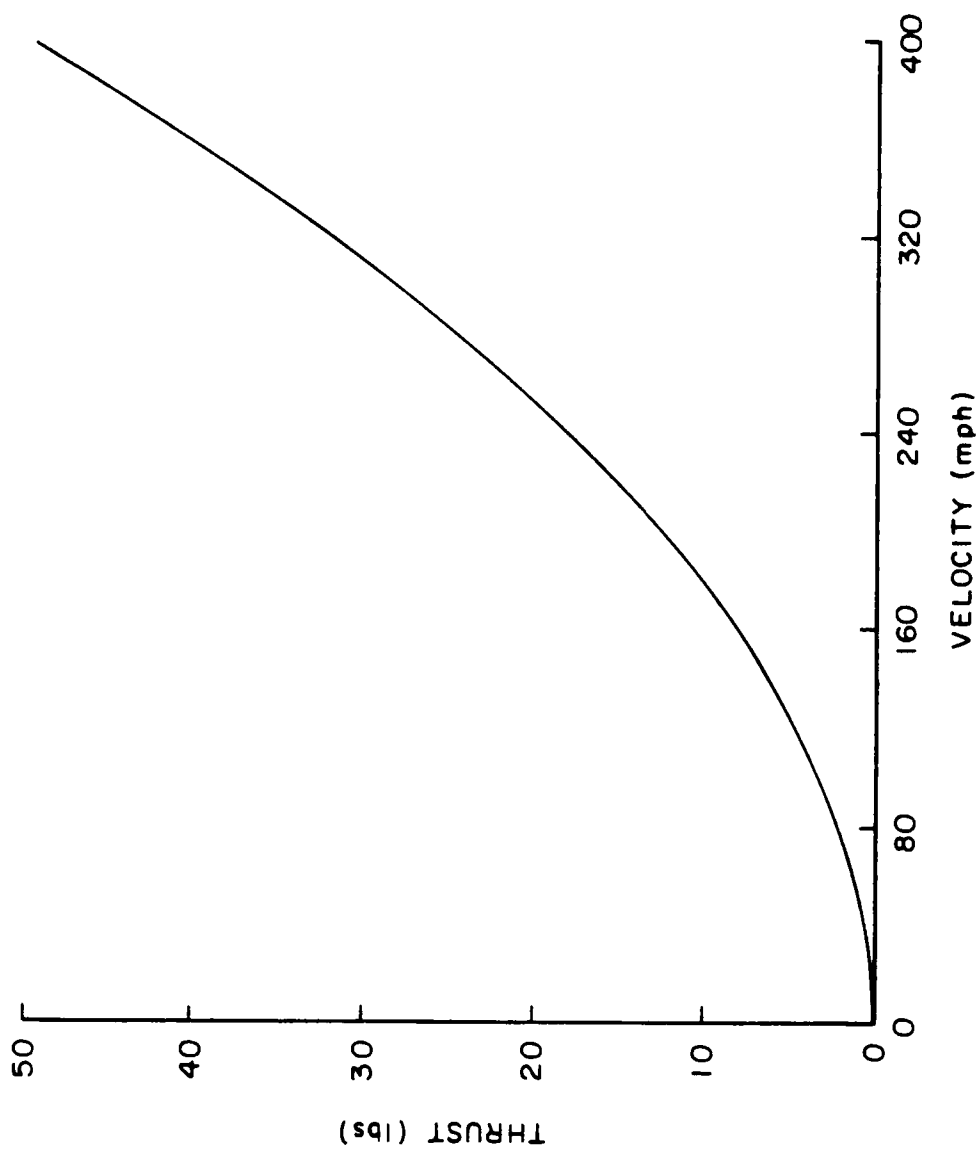


Figure 11. Thrust force vs. aircraft velocity.

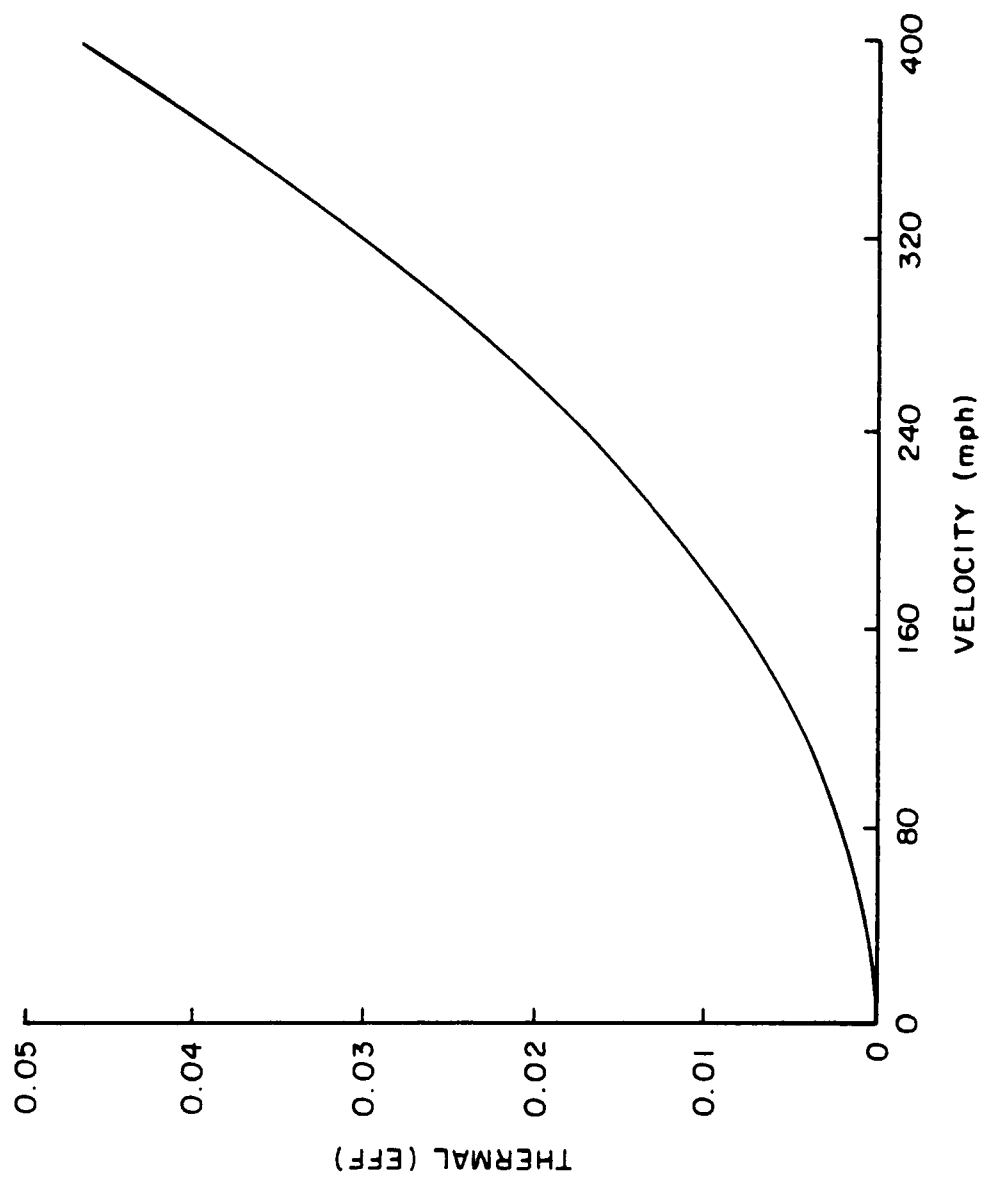


Figure 12. Thermal efficiency vs. aircraft velocity.

CHAPTER IV

INLET AND DUCT DESIGN

Inlet Design Considerations

There are many items to consider when designing for the efficient operation of the inlet. Such considerations are size and location of the entry, entry lip shape, orientation, cross-sectional shape, and boundary layer removal.

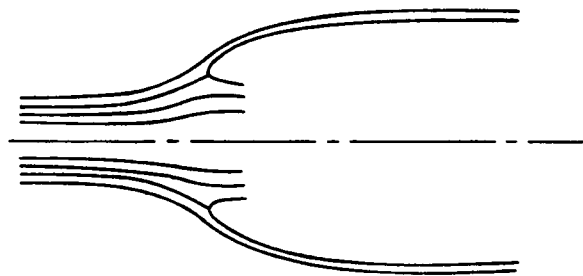
Inlet Operation

The conditions at the entry of the inlet are dependent upon the volumetric flow rate required for cooling and the flight (velocity) conditions. The flow may be retarded in high speed flight, (Figure 13a), be accelerated in low speed flight and climbing conditions (Figure 13b) and approach the inlet from all directions under static conditions (Figure 13c).

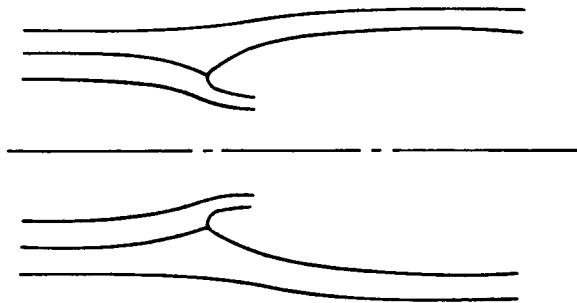
It was also recognized in early research (Ref. 23) and subsequently shown that the most important parameter in the design of inlets is the ratio of the mean velocity at the narrowest section of the inlet to the free stream velocity (V_i/V_o). This is the inlet velocity ratio.

Inlet Size

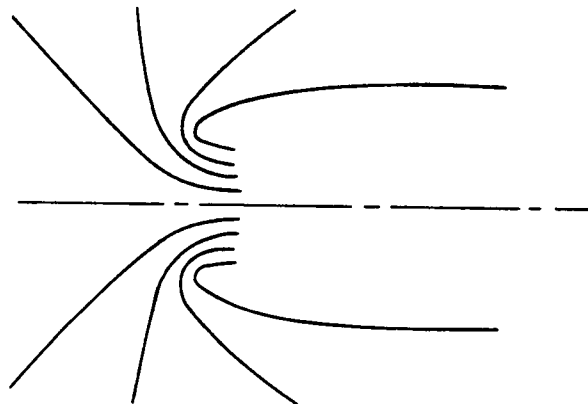
It is these two variables, volumetric flow rate and velocity ratio, that determine the intake area. It would seem possible that an intake could be designed with a minimum frontal area for the required area. This is indeed the case, and is referred to as the



a. HIGH SPEED



b. CLIMBING



c. STATIC

Figure 13. Types of inflow.

"optimum intake." Details of the optimum intake are discussed in Appendix II.

Practical Intake Contours, Lips and Intake Velocity Ratio

One characteristic of the optimum intake contour is the relatively small (sharp) nose radius, shown in Figure 14. A plot of the theoretical pressure distribution for a typical optimum intake is shown in Figure 15.

This figure shows that for velocity ratios different from the design velocity ratio, pressure "peaks" develop about the leading edge. At lower than design inlet velocity ratios, a pressure peak exists on the outside of the leading edge, and at higher than design velocity ratios, a pressure peak exists on the inside of the leading edge.

It should be remembered that the calculations for the optimum intake contour are based on inviscid flow assumptions. When actual flow conditions are used (flow with viscosity), the flow separates from the leading edge near the calculated positions of the pressure peaks. This separation causes an increase in drag and a reduction in duct efficiency.

In looking at the operation of the intake, it is seen that the flow about the fairing (inner and outer surfaces) is similar to the flow about an airfoil. The lower surface of the airfoil corresponding to flow on the inner fairing surface, and the upper surface of the airfoil corresponding to flow on the outer surface. The intake lip corresponds to the airfoil nose.

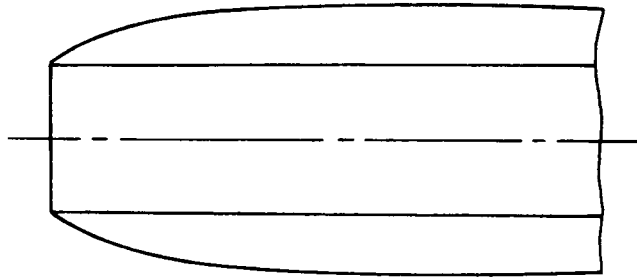


Figure 14. Optimum contour.

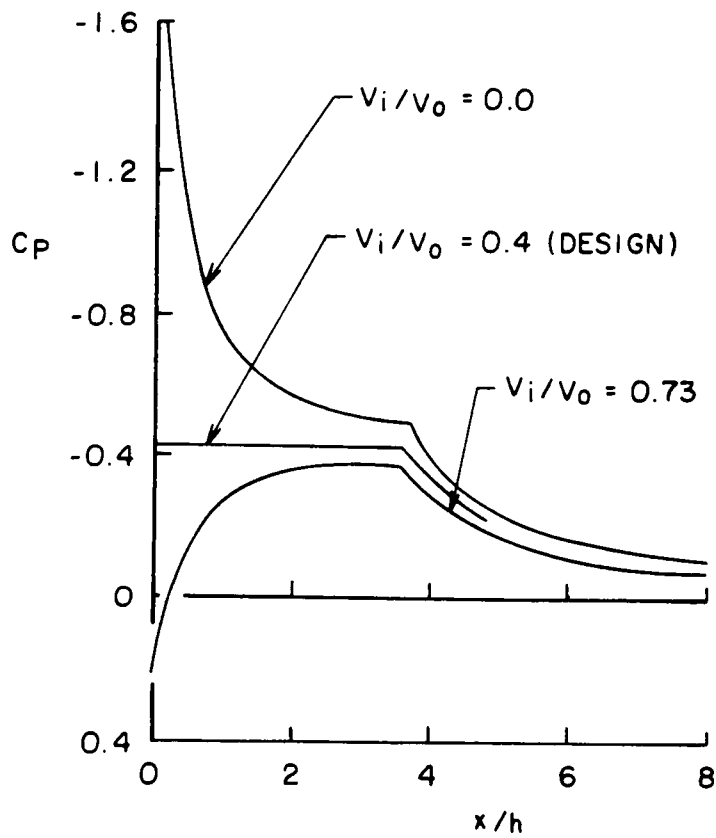


Figure 15. Pressure distribution on outer surface of optimum intake.

The inlet velocity ratio is a function of the angle of attack of the airfoil. Figure 15 shows that at low inlet velocity ratios (such as would be encountered in high speed level flights), the angle of attack can be at high positive values. At high inlet velocity ratios, i.e., climbing flight, the angle of attack can be at high negative values. Thus, it follows from airfoil theory that in high speed flight, the flow may separate on the outside of the fairing, and in climbing flight the flow may separate on the inside of the fairing. In either case, this flow separation is an undesirable condition since it creates unnecessary drag and leads to unacceptable duct efficiencies.

The ideal inlet would be one designed with an optimum contour that would employ a variable geometry so that the intake lip would always operate at the optimum angle of attack. In practice such a design would lead to increased mechanical complexity and increased weight. A fixed geometry inlet, which is the best compromise for the operating conditions, is a more practical approach. Sharp intake lips (small radius) and thin fairings should be avoided in the design of fixed geometry inlets. These features lead to designs that are too sensitive to operating conditions, i.e., small changes in operating conditions may cause separation. Contours that provide the most uniform pressure distributions, i.e., approach the optimum contour, will be the most efficient.

Contours

The need for practical contours for aircraft designs was recognized early in the development of aircraft, primarily with the

advent of cowlings for radial engines. Studies such as Ref. 1 developed empirical contours for efficient operation. Studies by other researchers, Ref. 47, developed geometrically similar contours. The results of these studies led to the development of two basic contours of practical significance. Such contours were developed for axisymmetric circular intakes. However, Ref. 47 and Ref. 37 indicate that these contours can be extrapolated to other two-dimensional and three-dimensional inlets. These contours are the Kuchemann inlet and the NACA-1 Series inlets. The details of these inlets are shown in Appendix III. The Kuchemann inlets consist of two quarter ellipse segments which fair into the nose radius. The ellipses are proportioned according to classes, each class being defined by a series of constants (see Appendix III). The Class A inlet is characterized by a well-rounded nose, the Class B has a moderately rounded nose, and the Class C has a small radius.

The intake then is defined by class and by the area ratio, such as B-40, where B indicates the series and 40 indicates the area ratio as a percentage. The use of low percentages will define inlets with thick lip contours and large radii of curvature, while high percentages will produce thin lip contours and small radii of curvature. Aerodynamically, this leads to the Class A being the least sensitive and the C Class being the most sensitive, with the C Class approaching the theoretical "optimum" intake. For most general aviation aircraft's operating velocity ratios, the Class A with well-rounded inlet lips would probably be a good starting point.

The NACA-1 inlet description and coordinates are detailed in Appendix III. These inlets are defined by two parameters--the area ratio and the length, and are designated in the form 1-40-150. The first number designates the series, the second number specifies the inlet diameter in percent of the maximum diameter, the third number specifies the nose-inlet length in percent of the maximum diameter. The NACA-1 series differs from the Kuchemann inlets in that the area near the nose tends to be flatter. This reduces the velocity increment at the nose and reduces the resulting pressure peaks. The profile, however, tends toward the C Class Kuchemann inlet. For inlets with high area ratios, the NACA-1 series tends to produce thin inlets with sharp nose radii, a condition which leads to greatly increased sensitivity. This sensitivity can be alleviated to some degree by modifying the NACA-1 series with an increased nose radius and internal contour as shown in Figure 16. The nose radius may also be modified for short inlets with small area ratios. This results in a large external curvature with small nose radius.

Inlet Velocity Ratio

Studies of inlets for heat exchangers (Ref. 31) have indicated that there is a range of inlet velocity ratios that lead to efficient operation. This range appears to be $0.3 < V_i/V_o < 0.6$. It should be noted that every effort should be made to design the inlet to operate at the high end of this range.

Low inlet velocity ratios (especially in the presence of a boundary layer) do not have enough energy to overcome the existing adverse pressure gradient. This leads to separation at the junction

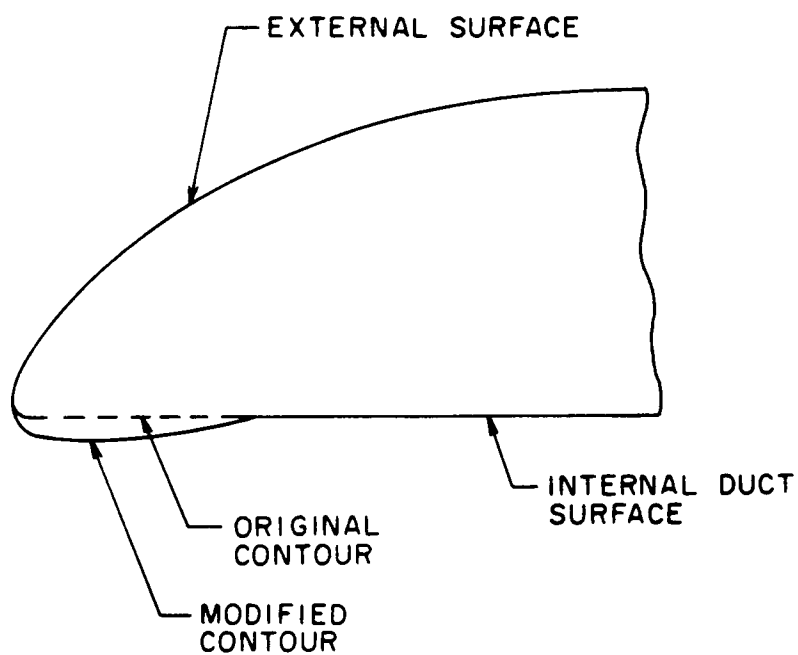


Figure 16. Modified nose radius of NACA 1-70-030 nose inlet.

of the surface and the inlet which reduces duct efficiency and greatly increases the drag.

Inlets with high velocity ratios do not usually have the problem of overcoming the adverse pressure gradient; however, at high inlet velocity ratios there is a tendency for the flow to separate on the inside of the inlet lip, causing further separation in the diffuser.

Location of the Entry

The freedom to locate the entry at any position on the airframe does not usually exist. Component location (i.e., engines, payload, etc.) and airframe design may require the inlet to be located at some position other than what is considered optimum.

When possible, the ideal position (Ref. 46) is to locate the entrance at an existing airframe flow stagnation point. This has the advantage of providing for maximum pressure recovery with no addition of frontal area.

When scoops, etc., are employed, every effort must be made to provide maximum pressure recovery with minimum additional drag.

Data indicates (Ref. 11) that when inlets are located on a typical fuselage, the lower quadrant is the best, followed by upper quadrant next, and then side locations. The lower quadrant, while providing possibly the best flow location, requires that some thought be given to foreign object damage, since foreign objects picked up by the propeller and landing gear may enter the intake. Locations in the upper quadrant eliminate most foreign object considerations; however, care must be taken in the aerodynamic design. Operation of the aircraft at high angles of attack may place

the inlet in a region of separated flow. Side locations are also usually difficult because of high flow angularity and increased local flow velocity.

Boundary Layer Considerations

When the inlet is located adjacent to, or on, a body (i.e., fuselage or wing), considerations should be given to the boundary layer in the vicinity of the inlet. Various tests (Ref. 70) indicate that at sufficiently high velocity ratios (i.e., 0.6-0.7) reasonably smooth flow can be achieved; however, when the inlet velocity ratios are reduced to about 0.3-0.4, the flow ahead of the inlet separates at the junction of remounting surface and the inlet opening causing high duct losses. The primary cause of the separation is the ingestion of the boundary layer air, and since the boundary layer air has insufficient energy to overcome the existing adverse pressure gradient, the flow separates. It is obvious that to achieve good duct efficiencies, the boundary layer in the vicinity of the duct must be controlled. Appendix IV contains a discussion of boundary layer control for various inlet types.

CHAPTER V

DIFFUSERS

The purpose of the diffuser is to convert the flow kinetic energy into potential energy (i.e., convert the dynamic pressure to static pressure). This is accomplished by allowing the flow area to expand. In subsonic flow an increase in flow area results in a decrease in velocity and a corresponding rise in static pressure. If the resulting adverse pressure gradient is too large, the flow will separate from the walls of the diffuser (Figure 17). This separation results in increased pressure losses and a reduction in the cooling efficiency of the heat exchanger. The main problem in diffuser design is to achieve the required pressure rise without flow separation, and the main design variable is the rate of expansion.

Optimum Divergence Angles

The optimum rate of expansion (divergence angle) is dependent upon the shape of the diffuser. The optimum expansion angle (2θ) for a circular conical diffuser is between 5° and 8° with the efficiency dropping rapidly for angles (2θ) greater than 10° .

The corners adversely affect the flow in square or rectangular diffusers, resulting in efficiencies below those of the circular conical diffuser. Maximum efficiency of square or rectangular diffusers is also achieved at different expansion angles. Square diffusers achieve the maximum efficiency at an expansion angle (2θ)

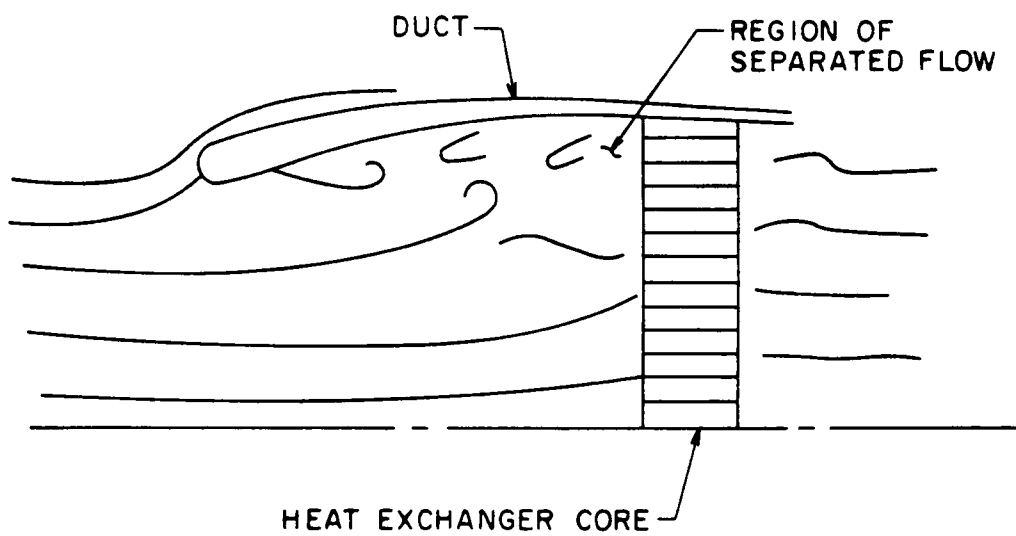


Figure 17. Separated flow in a diffuser.

of approximately 6° . Rectangular diffusers, where one pair of walls remain parallel, are most efficient at an expansion angle (2θ) of about 11° .

Boundary Layer

Thinner boundary layers are better than thick ones, in that there is less of a tendency for the flow to separate with a thin boundary layer. Where possible short ducts should be used ahead of the diffuser. Where this is not possible, boundary layer removal (bleeds, etc.) should be considered.

Pressure Rise

Tests of diffusers (Ref. 37) indicate that the pressure continues to rise for some distance behind the exit of the diffuser. The increase in efficiency by employing a straight section behind the exit is on the order of 5 percent to 15 percent, depending on the duct shape and the entrance velocity distribution. In most cases, this length should be approximately equal to two diameters to six diameters. As the divergence angle increases or the turbulence increases, the distance downstream from the diffuser exit to the position of maximum static pressure increases.

Diffuser Followed by a Resistance

An interesting effect is observed when a diffuser is followed by a resistance. The resistance has a beneficial effect in that with a properly shaped diffuser high expansion angles can be achieved without separation. This effect appears to be caused by the spreading of the flow as it approaches the resistance. This

increases the normal velocity gradient near the diffuser wall, may decrease the pressure gradient along the wall, or both.

References 37 and 42 provide guidelines and geometric data for the proper design of these diffusers. Utilizing this data, it appears possible to design diffusers having wall angles (2θ) up to 25° to 30° that have high efficiencies.

Three-Dimensional Considerations and Cross Sectional Shape

In most applications, the intake will not be two-dimensional, nor will it be symmetrical. However, design and analysis can be achieved by considering the basic forms. An oval inlet can be achieved by considering the basic forms. An oval inlet can be considered to be made up of a two-dimensional intake, with circular side walls. In this case, the wall thickness should not be constant. In the design of three-dimensional intakes, it is necessary to keep the local area ratio (the ratio of the intake area of that section to the total frontal area of that section) within a specified amount to insure that the local velocities over the outer surface are minimized. If proper attention is paid to the design of various cross-sectional shaped intakes, the drag developed should be no greater than that of a circular intake.

In all cases, the three-dimensional properties of the flow must be considered. Flow angularity, separation, and boundary layer require consideration of 3-D effects.

Wing (leading edge) intakes suffer three-dimensional effects more than others. The lip contour of the leading edge intake can be developed by considering the flow to be two-dimensional.

However, when the inlet is installed, the three-dimensional flow over the wing should be considered. If the leading edge is swept, there could be substantial spanwise flow.

For wings with a swept leading edge, the spanwise flow introduces a parallel component to the flow. The flow must then be turned to flow into the inlet. This turning causes the velocities to be higher on the outboard end of the intake than on the inboard end. This effect requires that walls of the outboard end be thickened and an increased nose radii provided. Also with an increase in sweep angle, there is a tendency for the flow to also separate from the inner wall of the inboard end.

CHAPTER VI

EXIT CONSIDERATIONS

Flow Regulation

Changes in flight operating conditions will necessitate changes in the volumetric flow rate through the heat exchanger.

Ignoring heat transfer, the power absorbed by the duct is given by:

$$P = Q \Delta P_t$$

where:

P = power absorbed (HP)

Q = volumetric flow rate (ft^3/sec)

ΔP_t = change in total pressure between inlet
and exit stations

The attainment of maximum efficiency requires that only the required flow rate be utilized (any excess will only decrease efficiency) and that the total pressure loss be minimized. The total pressure loss is minimized by providing for the maximum recovery and minimum momentum loss. This minimization is done by providing a means of varying the exit area. Flow control by varying the inlet area (internal shutters, etc.) will result in low efficiencies and should not be used in design.

Wake Formation

Early research (Ref. 23) recognized that the outlet flow may affect the flow over adjacent surfaces. Changes in flow viscosity due to heating, and/or an exit velocity different from the free stream

velocity in the vicinity of the exit, may adversely affect the external flow and result in increases in external drag, primarily due to increased skin friction and wake formation. If the cooling flow discharge is of lower energy than the free stream, the cooling airstream may induce separation over the adjacent surface with the resulting increase in drag.

Exit Design

The cooling air exit should be designed as a converging duct opening downstream. This results in an essentially parallel flow behind the exit. Provisions should be made so that the exit area can be varied to insure that the static pressure of the exiting stream is the same as the free stream (i.e., the velocities are matched) in the vicinity of the exit. Exits designed such that the cooling air is discharged over sharp corners, pipes, etc. are inefficient. Every effort should be made to provide a clean exit. The cooling air should be discharged as close to the free stream direction as possible; any discharge that requires the exit flow to turn will result in large losses, with corresponding increases in drag.

Outlet Flaps

Outlet flaps serve two purposes. One is to provide a convenient method of changing the outlet area. The other is to provide a means of enhancing the cooling flow under marginal conditions.

The pressure available for cooling is the difference between the static pressure in the inlet and the static pressure at the exit.

Therefore, assuming that maximum recovery is designed into the inlet, a method of providing for a higher ΔP is to lower the static pressure at the outlet. This may be necessary in the take-off or climb conditions where the inlet velocities are low. The flow over the aft section of a cowl or cooling duct is such that the static pressure on the outside surface is approximately equal to the free stream static pressure. If a flap is deflected into this stream, an increase in local velocity is achieved with the resulting drop in static pressure. This increases the ΔP available for cooling.

The maximum deflection of the flap is limited, since if the deflection angle is too large, the flow will separate. This does nothing for the reduction of the static pressure and greatly increases the drag. The following curve (Figure 18) from Ref. 32 indicates that the reduction in static pressure for flap deflection angles of 15° to 30° is on the order of 0.4 to 0.6 of the free stream dynamic pressure. Flap angles greater than 30° are not considered since at angles larger, the drag increases significantly. This curve is derived from data taken on a typical radial engine cowl. Ref. 32 indicates that the effect at the rear of an underslung duct is probably somewhat reduced due to the lower aspect ratio of the flap. Data for short span flaps shows that the drag can be minimized by closing the ends (similar to end plates on a wing).

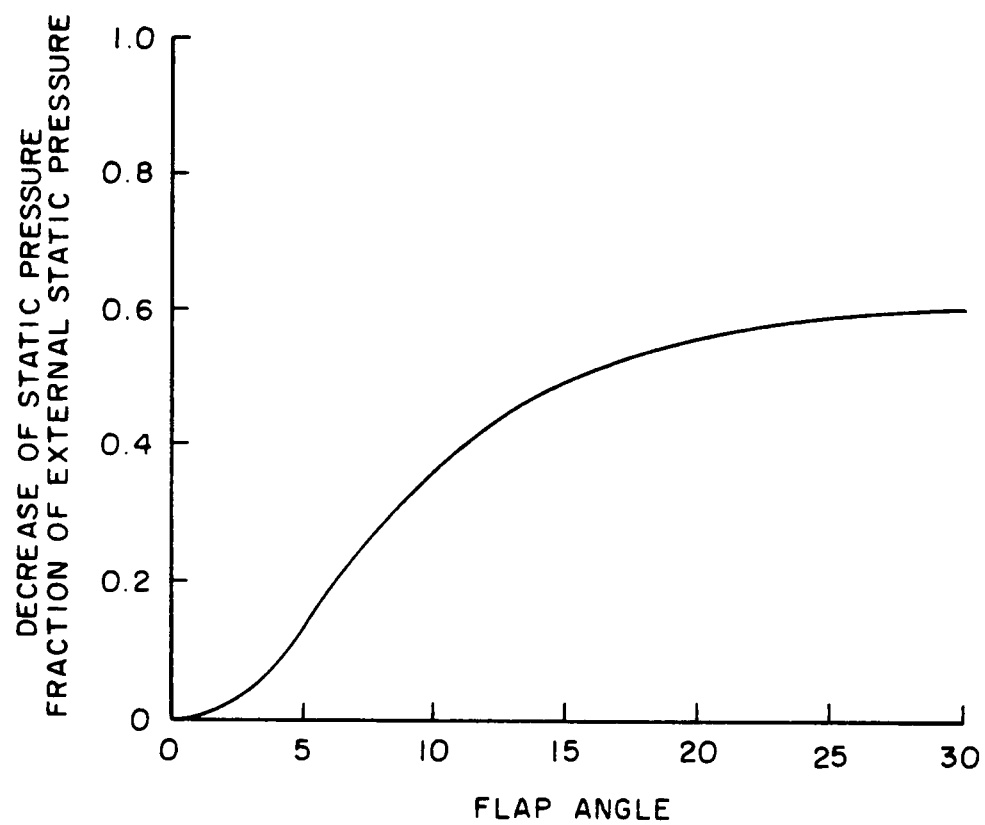


Figure 18. Effect of flap angle on static pressure.

CHAPTER VII

INSTALLATION CONFIGURATIONS

Basic Types

There are three basic types of installations. These are the ring, underslung, and wing installations (Figure 19a, b, and c).

All other installations (ring coolers, etc.) are variations of these designs.

Ring Installations

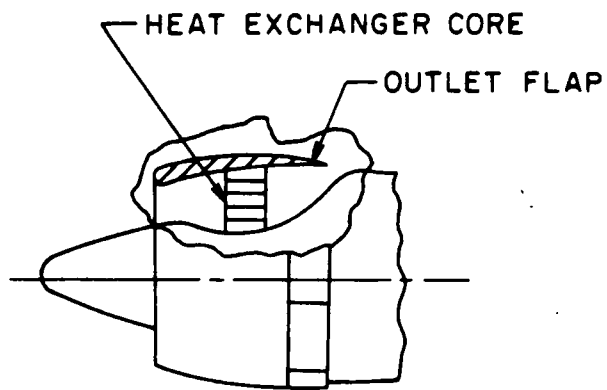
The main advantages of the ring cooler installation, either axial or radial, are: (1) no additional frontal area, (2) proximity to the engine results in shorter connections, etc., and (3) cooling at low speed can be enhanced by proper design of the propeller blade root sections, i.e., addition of cuffs. This increases the dynamic pressure at the inlet, resulting in a larger ΔP for cooling.

The design of the cowl is essentially the same as for air cooled radial engines. Reference 47 provides a fairly complete report on spinner and cowl arrangements. It should be noted that the spinner and cowl should be used since this provides a good method of achieving the proper inlet velocity ratio.

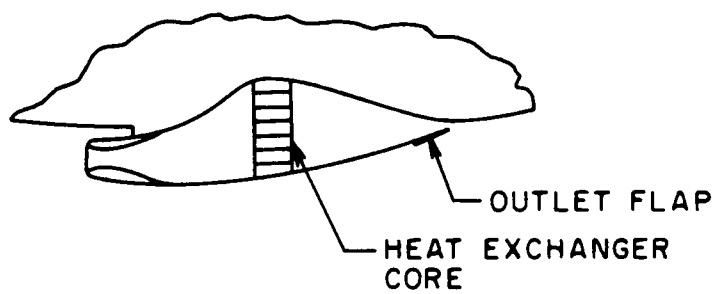
A disadvantage of this design is that the boundary layer built up on the spinner cannot be easily removed and therefore will cause inflow losses.

Underslung Installation

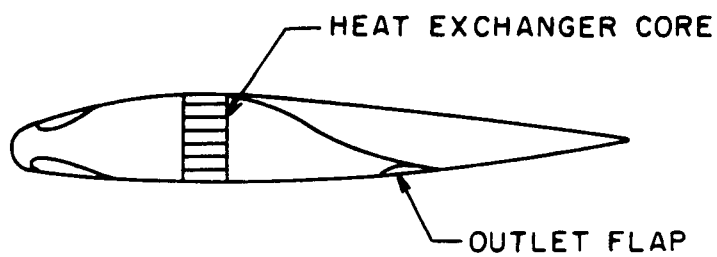
Installations of this type have two main drawbacks: (1) they add to the frontal area of the aircraft, and (2) if attention is not



a. RING INSTALLATION



b. UNDERSLUNG INSTALLATION



c. WING INSTALLATION

Figure 19. Basic cooler installation.

paid to the external shape, they may promote separation of the flow locally and thus add significantly to the external drag of the aircraft. These installations do, however, provide for the mounting of larger heat exchanger cores internally than can be achieved with other types of installations, while providing a minimum additional frontal area. Such a design provides for low duct losses and low internal drag. Attention to details such as intake lips, diffuser design, boundary layer removal, and external shape will result in an efficient installation.

Wing Installations

The wing installation is from an aerodynamic viewpoint probably the best engineering compromise for the duct design. The installation adds no additional frontal area, is located at a natural stagnation location so that inflow losses are minimized, and a heat exchanger core of optimum length can usually be accommodated. The design, however, must consider flow about the entire wing. Various tests, Ref. 45, 50, and 60, indicate that there is an optimum location and design for the inlet and outlet. Reference 45 indicates that, in general, wing installations increase the drag, decrease maximum lift, and change the lift curve slope so as to give lower lift for a given angle of attack. Further discussion of wing intakes is contained in Appendix V.

CHAPTER VIII

HEAT EXCHANGER DESIGN

Mechanical Considerations

The most important mechanical consideration is vibration and its effect on the heat exchanger. The materials used for aircraft heat exchangers tend to be thin to reduce weight and enhance the heat transfer. Tube thicknesses are on the order of 0.006 inch to 0.012 inch wall thickness, and fin material is on the order of 0.00325 inch to 0.005 inch thick. It is obvious that any mechanism constructed of such thin material cannot tolerate much in the way of vibration.

The mounting of the heat exchanger is therefore very important. Designs for mounting should include methods to isolate the heat exchanger from the airframe to minimize the transmission of vibration and shock loads. Provisions should also be incorporated to allow the heat exchanger to grow thermally, so that thermal induced stresses are minimized. Connections to the heat exchanger should be made through some type of flexible connections. This reduces the transmission of engine induced vibrations and accommodates the flexible mounting of the heat exchanger.

Seals also need to be provided between the ducting and the heat exchanger to prevent leakage of the cooling air. This leakage leads to a reduction in the cooling air momentum at the exit which shows up as an increase in drag.

Heat Transfer Considerations

The heat exchanger core should be designed such that (1) the heat exchanger is capable of the required rate of heat dissipation, (2) the pressure drop required does not exceed the pressure drop available, and (3) the dimensions do not exceed the space available for installation. The heat exchanger core sizing can be approached from any of these variables. In most cases, the space available is the critical factor and perhaps best defined in the early stages of design. This approach is used for this study. Assuming the installation envelope and flight operating conditions are specified, the pressure drop available, the core pressure drop and the power cost can then be calculated.

Heat Exchanger Core Sizing

The design of the heat exchanger core requires knowledge of the heat rejected from the engine to the coolant. This data appears to be difficult, if not impossible, to obtain without testing. Review of the literature has produced very little data concerning this subject. It appears, however, that a reasonable estimate can be made based on the rated shaft horsepower (RSHP) of the engine.

There is a wide variation in the published literature for the estimate of the heat rejected to the coolant. The Cessna and Beech studies (Ref. 28, 79) include NASA estimates for the proposed engines. The heat rejection values shown for the piston and wankle engines vary from 50 percent RSHP to 100 percent RSHP. Ref. 36 shows heat rejection values from 30 percent RSHP to 110 percent RSHP.

Variations of this magnitude seriously affect the final size of the cooling installation and therefore need to be as accurate as possible.

Two NACA reports (Ref. 55, 56) give some idea of quantity of heat rejected to the coolant, at least for piston engines. These reports attempt to define the effect of engine and coolant variables on the heat rejected to the coolant for a 12 cylinder liquid cooled engine. The engine variables include charge flow, carburetor air temperature, engine speed, fuel-air ratio, ignition timing, and exhaust pressure. The coolant variables included coolant composition, coolant flow rate, coolant temperature, and coolant pressure. The heat transferred to the coolant is a function of the cylinder diameter (area), gas temperature, and temperature of the cylinder liner (barrel and lead). The engine design essentially fixes the cylinder diameter. The gas temperature and cylinder temperature, however, can be varied by varying engine operating variables. Results of the NACA studies imply that the most important engine variables are charge flow (engine power), fuel-air ratio, and engine speed. The charge flow determines the total amount of heat generated and the fuel-air ratio determines the gas temperature. Leaning the mixture closer to the stoichiometric ratio results in large increases in the gas temperature. The engine speed increases the heat generated, primarily by friction between the piston rings and the cylinder wall.

Coolant variables can also affect the rejection of heat. By varying the coolant composition, i.e., by increasing or reducing the percentage of ethylene glycol in the coolant, the heat rejection

is changed. This occurs because of the change in specific heat of the mixture. As the percentage of ethylene glycol is increased, there is a reduction in specific heat; therefore, for a given coolant flow rate, the amount of heat rejected is less and there is a corresponding rise in cylinder temperature (and cylinder heat temperature). Variation of the coolant flow directly affects the heat rejection. Lower flow rates result in lower heat rejected to the coolant. At lower flow rates there is less heat capacity and less heat can be transferred to the coolant. This results in higher engine operating temperatures and can result in local boiling of the coolant. Increasing or decreasing the coolant operating temperature can also affect the heat transfer. Increasing the coolant operating temperature decreases the heat rejected since the heat transfer is a function of the ΔT between the cylinder wall (and head) and the coolant.

Heat Rejection Variables

Three coolant variables must be defined to size the heat exchanger: the heat rejected to the coolant, the coolant flow rate, and the coolant operating temperature.

From the data of Ref. 56, the heat rejected to the coolant varies from 30 percent to 41 percent of the brake horsepower (Figure 20). The heat rejected is for a constant coolant flow rate and varying engine power. It is proposed that a value of 40 percent of the engine operating horsepower be used for sizing of the heat exchanger. This value will no doubt vary with engine type; the wankle and small bore engines are expected to be somewhat lower.

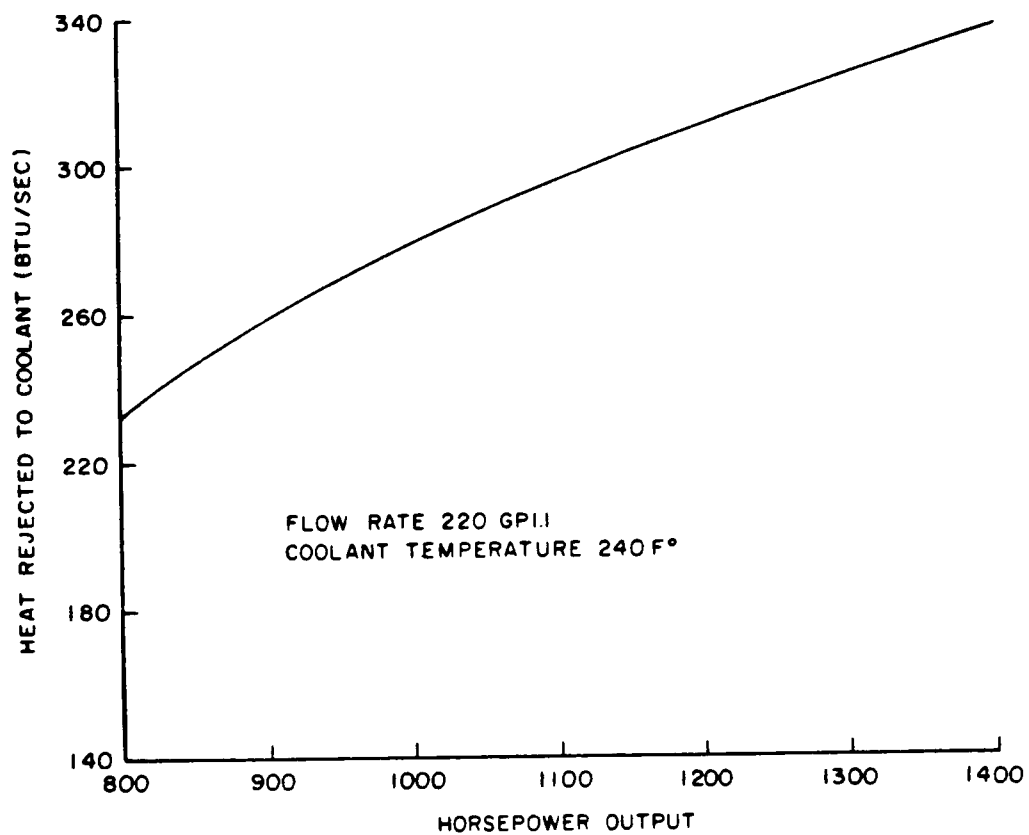


Figure 20. Heat rejected to coolant for Allison V1710 for various output horsepower.

Most general aviation power plants, however, do not vary significantly from the cylinder dimensions used in the tests.

Coolant flow rate data is meager at best. Ref. 36 indicated that coolant flow rates should be set at 1/2 to 1 gpm per horsepower. This flow rate appears to be excessive. The coolant flow tests in Ref. 56 used flow rates from 0.0277 gpm/hp to 0.3846 gpm/hp. At very low rates, there is a risk that the coolant will boil locally, resulting in uneven cooling and possibly high engine operating temperatures. A conservative estimate for flow rate to use for design purposes appears to be 0.4 gpm/hp.

Heat Exchanger Surface Selection

The surface used in this study is the secondary surface type, flat tubes with continuous fins, found in most automotive applications shown in Figure 21. The surface data is from Ref. 30. The designation of the surface utilizes the number designation of Ref. 30. The surface is defined by a number such as 9.68-0.87R. The first number(9.68) defines the number of fins per inch. The second number (0.87) defines the depth of the flat tube. The letter (or letters) define the tube orientation and fin surface, i.e., no letter indicated inline, S indicates staggered, SR indicates staggered and ruffled.

Properties of the Atmosphere

The air is assumed to be dry and to obey the perfect gas laws. Since aircraft may be operated in locations where the temperature may be significantly different from the NACA standard atmosphere, and the inlet temperature is a major factor in the design of the heat exchanger, an atmospheric model with higher temperatures is needed. 58

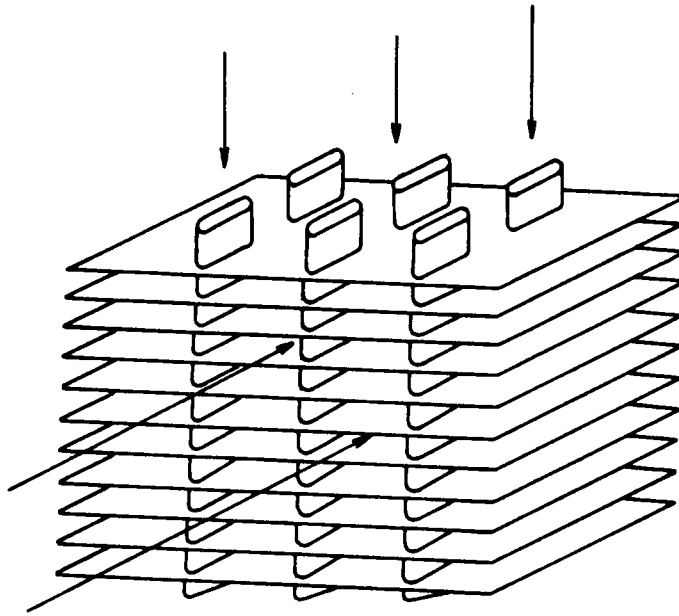


Figure 21. Heat exchanger surface geometry.

Ref. 69 indicates that in the 1940's the Army Air Corps specified a summer atmosphere to be used for aircraft cooling calculations. This atmospheric model is used in this study. The summer atmosphere is essentially the same as a standard NACA atmosphere with respect to pressure. The sea level temperature is specified to be 100 °F and then varies linearly at a lapse rate of 3.6 °F per thousand feet up to 46,500 feet. The density is calculated from the temperature and pressure by the perfect gas law.

The variation of atmospheric properties is calculated according to the functions of Ref. 16, with the modification of a bias of 100 °F being used in the temperature calculation. Although these functions do not predict the atmospheric properties exactly, with reference to the new NACA atmosphere model, the error is considered to be acceptable for this application.

The pressure at any altitude is given by:

$$P = \delta P_0$$

$$\text{where: } \delta = \theta^{5.2561}$$

$$P_0 = \text{Sea Level Pressure}$$

$$\theta = 1 - 6.875E-6(h)$$

$$h = \text{Altitude (ft)}$$

The density at any altitude is given by:

$$\rho = \rho_0 \sigma$$

$$\text{where: } \sigma = \theta^{4.2561}$$

$$\rho_0 = \text{Sea Level Density}$$

The temperature at any altitude is given by:

$$T = T_0 - h/1000.0 \text{ (3.57)}$$

Conditions at the Heat Exchanger Core Face

Temperature conditions at the core surface are calculated based on stagnation conditions. The air is considered to be slowed isentropically from the flight speed to rest. The change in temperature is given by:

$$\Delta T = 0.832 \left[\left(\frac{V_0}{100} \right)^2 - \left(\frac{V_2}{100} \right)^2 \right]$$

where: ΔT = Temperature F°

V_0 = Flight velocity ft/sec

V_2 = Velocity of the cooling air

at the core face

For purposes of sizing V_2 is considered to be equal to zero, therefore, the temperature of the cooling air at the core face is given by:

$$T_2 = T_0 + 0.832 \left(\frac{V_0}{100} \right)^2$$

The free stream dynamic pressure is given by:

$$q_0 = 1/2 \rho_0 V_0^2 F_c$$

(F_c is a compressibility correction factor. For most General Aviation applications, this can be considered equal to 1.) If the inlet is located in the propeller slipstream the dynamic pressure available is approximately 30 percent higher. The dynamic pressure is then:

$$q = q_0 + 0.3 q_0$$

All of the dynamic pressure is not available at the core face due to losses in the diffuser; therefore, the dynamic pressure at the core face is given by:

$$q_2 = q_0 \eta$$

where: η is the diffuser efficiency.

Heat Exchanger Wall Temperature

The heat exchanger wall temperature is assumed to be the same as the average temperature of the coolant. The temperature is also assumed to be constant along the length of the tube. Ref. 77 shows that these assumptions are very close to the actual operation of the heat exchanger.

Estimate of the Heat Exchanger Core Frontal Area

The minimum heat exchanger frontal area is given by (Ref. 37), assuming maximum efficiency and no losses,

$$A_{B_0} = \frac{Q}{C_p g (T_w - T_{B_i}) V_0} * \frac{2.22}{\sigma}$$

Due to losses in actual installations, it is necessary to use more area than the minimum area calculated. Test data from Ref. 37 suggests that a good first estimate of the needed area is:

$$A_B = 1.44 A_{B_0}$$

The actual area used may further have to be increased. It should be noted that the heat transfer area can be increased by increasing the length rather than the frontal area. Ref. 37 says that for the heat transfer to be large, the velocity should be low, the hydraulic

diameter of the passages should be small, and the heat exchanger core should be long. There is a limit, however, since as the hydraulic diameter decreases the pressure drop increases. Therefore, a point will be reached where a small gain in heat transfer is obtained at a large increase in pressure drop.

Two methods are used to increase the heat transfer. One is to stagger the tubes and the other is to provide slots or ruffles on the fins. Both of these methods do increase the pressure drop somewhat, but provide large gains in the heat transfer.

Heat Exchanger Core Pressure Drop

The general equation for core pressure drop is: (Ref. 30)

$$\frac{\Delta P}{P_1} = \frac{G^2}{2q_c} \frac{v_1}{P_1} \left[(K_c + 1 - \sigma^2) + 2 \left(\frac{v_2}{v_1} - 1 \right) + f \frac{A}{A_c} \frac{v_m}{v_1} - (1 - \sigma^2 - K_e) \frac{v_2}{v_1} \right]$$

Entrance Effects

The term $(K_c + 1 - \sigma^2)$ represents entrance losses.

Flow into the heat exchanger core is accompanied by a change in area, i.e., the free flow area of the heat exchanger core is not as large as the duct area. This results in an increase in velocity and resulting drop in static pressure.

Also accompanying this abrupt contraction is a pressure loss due to the free expansion following the abrupt contraction. This results because of the boundary layer separation on the leading edge of the fins, i.e., vena contracta and the changes in momentum with changes in the downstream velocity profile.

Exit Effects

The term $(1 - \sigma^2 - k_e)$ represents the exit losses.

Exit effects are essentially the same as the entrance effects. The flow encounters a change in area (drop in velocity) and a free expansion resulting in momentum losses as before.

Flow Acceleration Effects

The term $2 \left(\frac{v_2}{v_1} - 1 \right)$ represents the losses due to flow acceleration in the cone.

As the air flows through the passages in the heat exchanger, heat is added to the airstream. This heat addition decreases the density of the air while increasing the volume. Since this is flow in a constant area tube, the velocity of the air is increased. This increase in velocity is accompanied by a decrease in static pressure and an increase in momentum.

Friction Losses

The term $f \frac{A}{A_c} \frac{v_m}{v_1}$ represent the losses due to friction in the core.

Friction losses in the core result from the increase in velocity due to the reduction of flow area, the increase in velocity due to heating, and from the increase in viscosity due to heating.

Reference 30 provides curves for the friction factor that account for the entrance and exit losses, therefore, the equation for pressure drop reduces to:

$$\frac{\Delta P}{P_1} = \frac{G^2}{2g_c} \frac{v_1}{P_1} \left[(1 - \sigma^2) \left(\frac{v_2}{v_1} - 1 \right) + f \frac{A}{A_c} \frac{v_m}{v_1} \right]$$

Determination of Mass Flow

The mass flow through the heat exchanger is determined by the amount of heat that must be transferred to the cooling airstream and the entrance temperature of the cooling air using the methods outlined in Ref. 77. An equation can be developed to determine the mass flow.

The heat transfer from the surface of a heat exchanger by convection is:

$$H = h A \overline{\Delta T}$$

where: H = Heat flow (Btu/Hr)

h = Convection coefficient (Btu/hr-ft²-°F)

$\overline{\Delta T}$ = Logarithmic-mean temperature difference

The lent balance equation for the cooling air is:

$$H = W C_p (T_o - T_i)$$

where: H = Heat flow (Btu/Hr)

C_p = Specific heat of air at constant pressure

T_o = Outlet temperature of the air

T_i = Inlet temperature of the air

If these two equations are combined then

$$h A \overline{\Delta T} = w C_p (T_o - T_i)$$

noting that the definition of ΔT is:

$$\Delta T = \frac{\overline{\Delta T}_i - \Delta T_o}{\frac{\log_e \Delta T_i}{\Delta T_o}}$$

Then:

$$\frac{hA}{WC_p} = \text{Log}_e \frac{T_w - T_i}{T_w - T_o}$$

Rearranging:

$$T_o = T_i = (T_w - T_i) \left(1 - e^{-\frac{hA}{WC_p}} \right)$$

Substituting this expression into the heat balance equation yields:

$$H = WC_p (T_w - T_i) \left(1 - e^{-\frac{hA}{WC_p}} \right)$$

For the flow of air through a constant area tube the convection coefficient h can be represented as:

$$h = 0.2 (\mu/D)^{0.2} (\rho V)^{0.8}$$

The heat transfer area is given by:

$$A = \pi D L$$

The weight flow is given by:

$$W = g A_f \rho V$$

Substitution yields:

$$H = WC_p (T_w - T_i) \left(1 - e^{-0.1 R_N^{-0.2} L/D} \right)$$

This can be rewritten for mass flow as:

$$W = \frac{H}{C_p (T_w - T_i) \left(1 - e^{-0.1 R_N^{-0.2} L/D} \right)}$$

Note, however, that velocity (which is unknown) is still contained in the Reynolds number on the right side of the equation. The solution to this equation is done in an iterative manner. A Reynolds number

is assumed the mass flow calculated. Then using the mass flow a new Reynolds number is calculated, compared to the previous Reynolds number, and if there is a significant difference, the difference is halved and the sequence repeated until convergence is reached.

Overall Coefficient of Heat Transfer

The overall coefficient of heat transfer is determined by calculating the Stanton number for the surface type involved, calculating the unit film conductance, and determining the fin effectiveness.

The Stanton number varies with the surface type involved. Curves presented in Ref. 56 are used to determine the Nusselt number based on the airside Reynolds number.

The unit film conductance (h) is then calculated according to:

$$h = N_{ST} G C_p$$

where: h = Unit film conductance (Btu/Hr-ft²-°F)

N_{ST} = Stanton number

G = Mass velocity lbm/(hr-ft²)

C_p = Specific heat of air (Btu/LB-°F)

The unit film conductance and surface geometry is then used to calculate the fin effectiveness. Again the data presented in Ref. 56 is used to determine the fin effectiveness.

The overall surface effectiveness is then calculated according to:

$$\eta_o = 1 - \frac{A_f}{A} (1 - \eta_f)$$

The overall coefficient of heat transfer is then calculated

as:

$$\frac{1}{U_a} = \frac{1}{\eta_o h_a} + \frac{1}{(\alpha_w/\alpha_a) h_w}$$

Heat Transfer Units and Exchanger Effectiveness

The number of heat transfer units is calculated according to the data of Ref. 56 for both fluids unmixed.

The heat exchanger effectiveness is then calculated according to the series representation of Ref. 21, which is given as:

$$\epsilon = \frac{1}{R_N(NTU)} \sum_{n=0}^{\infty} \left[1 - e^{-NTU} \sum_{m=0}^n \frac{(NTU)^m}{m!} \right] \left[1 - e^{-R_N(NTU)} \sum_{m=0}^n \frac{[R(NTU)]^m}{m!} \right]$$

where: R_N = Reynolds number

NTU = Number of heat transfer units

Internal Drag

The internal drag is the force required to move the cooling air through the duct-heat exchanger combination minus the force recovered as thrust from the heat energy added to the air. The force required to move the cooling air is the result of losses in the total pressure due to inlet-diffuser effects, wall friction, and heat exchanger core pressure drop. Unless the duct is exceptionally long or operating at high subsonic mach numbers, the duct wall friction can be ignored without causing significant errors in the drag prediction. The internal drag (equated to the power consumed) is equal to the decrease

in momentum that occurs in the cooling air stream in its passage through the cooling installation. This is given as:

$$P = Q/g V_o (V_o - V_e)$$

The calculation of the internal drag then requires the calculation of the conditions at the duct exit (i.e. exit velocity). This is accomplished by calculating the conditions in a station to station manner through the entire duct as shown in Figure 7 of the mechanics of cooling section.

The calculation is accomplished in the following manner.

1. The flow is considered to be slowed isentropically from flight conditions at station 0 to rest at station 2.

The temperature is given (as before) as:

$$T_2 = 0.822 \left(\frac{V_o}{100} \right)^2$$

The pressure is given as:

$$P_2 = P_1 + q - (1 - \eta)q$$

where: η = Diffuser efficiency

The density is given by:

$$\rho_2 = 0.000583 \frac{P_2}{T_2}$$

where: P_2 = Pressure in (LBs/ft²)

T_2 = Temperature (R°)

2. Using previously calculated values for the temperature change through the heat exchanger core, and the pressure drop through the core the conditions at station 3 are calculated.

The temperature is given by:

$$T_3 = T_2 + \Delta T$$

where: ΔT = Temperature change through the core

The pressure at station 3 is:

$$P_3 = P_2 - \Delta P_{2-3}$$

where: P_{2-3} = Pressure drop through the core

3. The air is considered to expand adiabatically from the conditions at station 3 to the exit conditions (free stream conditions). A nominal value of 2 percent of the dynamic pressure is used as an estimate of the nozzle losses.

The pressure before expansion (accounting for nozzle losses) is:

$$P_4 = P_3 - 0.02 q_0$$

The temperature after expansion is given by:

$$T_4 = T_3 (P_0/P_4)^{0.286}$$

where: T_4 = Temperature at station 4 (R°)

Since the process is adiabatic the exit velocity can be calculated from the temperatures:

$$V_4 = 100 \sqrt{\frac{\Delta T_{3-4}}{0.832}}$$

The density is then calculated according to:

$$\rho_4 = 0.000593 P_4/T_4$$

Then using the weight flow determined by the heat exchanger. The volume flow is calculated, and the exit area determined

$$A_E = Q/V_4$$

4. The internal power consumption (drag) is then calculated:

$$P_C = W/32.2 \left(\frac{V_o (V_o - V_4)}{550} \right)$$

A drag coefficient can then be calculated based on the aircraft wing area by:

$$C_o = 1100(NP)/\rho_o V_o^3 A_s$$

where: C_o = drag coefficient

ρ_o = free stream density

V_o = free stream velocity

A_s = wing platform area of the aircraft

External Drag

Calculation of the external drag is not as straight forward as the calculation of the internal drag. Precise determination of the external drag will require testing. An estimate of the external effects can be made, however. This may provide some guidance for the design of the installation. The external drag is the sum of the skin friction drag, the pressure drag (form drag), and the interference drag due to boundary layer interaction, etc. For the general aviation aircraft (below critical Mach no.) the problem becomes one of delaying boundary layer transition and minimizing the wetted area. Buried installations tend to minimize the external drag since there may be

additional area, both lateral and frontal, and openings may be located at natural stagnation locations. If scoops or other openings are used to provide air into the the installations, this is not the case. Scoops add to the frontal and the wetted area of the aircraft. If the inlet (inlet lip, contour, etc.) is not properly designed and a proper velocity ratio not provided, there may be separation and a considerable increase in the pressure drag. Proper design of the inlet and duct contour should provide no significant increase in the pressure drag. The drag due to scoops is given by Ref. 32 as:

$$P_s = C_{O_s} q_o V_o A_s$$

It is also shown in Ref. 32 that a well designed scoop has a drag coefficient of about 0.06.

If any type of boundary layer control is used, there is a drag force produced that is charged to the external drag of the installation. If a boundary layer diverter is used, it is necessary to use the proper wedge angles, and provide for the proper flow. This results in a design that minimizes the pressure drag. The viscous component, however, is still quite significant. Appendix V contains data related to the calculation of diverter drag.

SECTION III. EXAMPLE INSTALLATIONS AND THE
PREDICTION OF THEIR PERFORMANCE

CHAPTER IX

DESIGN PROCEDURE

The following procedure is used to size a heat exchanger installation:

1. The aircraft layout and performance parameters are first established. The primary variables are the maximum engine horsepower, maximum operating altitude, and space envelope available.
2. The type of installation is chosen and a rough idea of the space envelope established.
3. The maximum horsepower with the minimum climb velocity is used to size the heat exchanger core. Several iterations may be necessary to achieve a satisfactory heat exchanger for the space available.
4. The resulting design is then investigated for other operating conditions, i.e., cruise and high speed. This sets other parameters such as airflow rates, exit areas, drag, etc.
5. Using the required airflow rates for the various altitudes and flight conditions, the intake area can then be calculated.

This is done by plotting the intake area vs. the velocity ratio on a logarithmic scale (Ref. 32). This results in straight lines. These can be used to pick off an intake area that assures a minimum velocity ratio of 0.4.

6. Depending upon the installation type, the intake lip profile can then be designed, the diffuser arrangement selected, and the exit (exit flap) can then be designed.
7. A prediction of the internal and external drag can then be made, which in turn can be used in the performance prediction of the aircraft.

CHAPTER X

EXAMPLE INSTALLATIONS

The design case considered for this study is an aircraft powered by a 150 HP Wankle engine. Since this study has not been applied to any existing aircraft the operating parameters are assumed. These are as follows:

maximum speed = 180 MPH

climb speed = 75 MPH

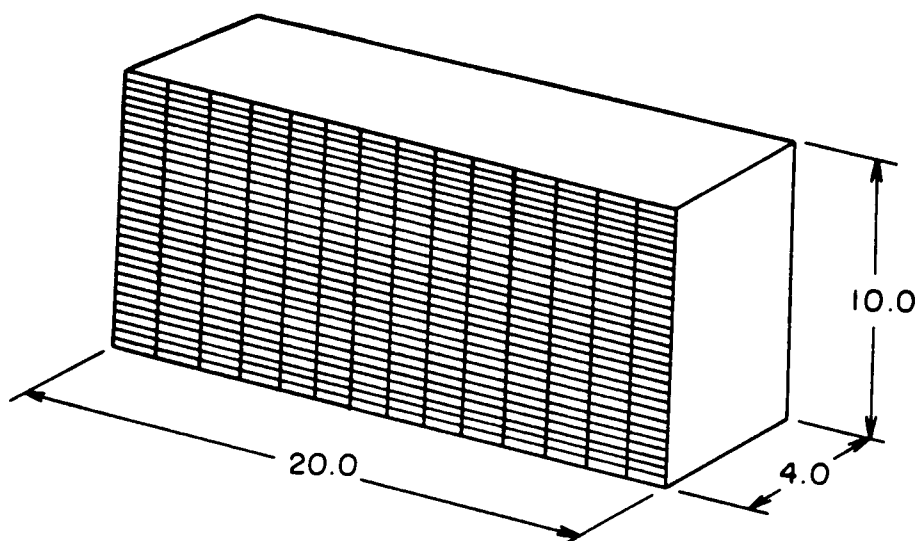
cruise speed (at 75 percent power) = 130 MPH

maximum operating altitude = 10,000 FT

The dimensions of the engine and a typical aircraft space envelope are determined from data in Ref. 28, 79. The engine is considered to be 52 inches long, 20.8 inches deep, and 18.4 inches wide. The installation location is continued to the nose of the aircraft and somewhat existing cowl lines are used.

Based on these dimensions a core frontal area of 1.39 ft^2 (20 inches wide x 10 inches high) was selected. The computer program of Appendix III (using equations outlined in Section VIII) is used to size the left exchanger. These runs indicated that a core of this frontal area would need to be 4 inches deep to accomplish the required heat transfer. The final core size is shown in Figure 22.

An investigation into the sensitivity of operating temperatures and diffuser efficiencies is shown in Figures 23 to 26. It is shown in Figures 23 and 24 that exit areas and internal drag are reduced with increasing operating temperature. If existing engines are used, i.e., automotive, 220 °F appears to be a good operating temperature.



SURFACE TYPE 9.21-0.737-SR
CORE FRONTAL AREA = 1.39 FT²
CORE VOLUME = 0.463 FT³

Figure 22. Heat exchanger dimensions.

CLIMB AT 75 mph
 150 HP ENGINE
 INLET IN SLIPSTREAM
 OUTLET FLAP DEFLECTED 15°
 SURFACE 9.21-0.737-SR

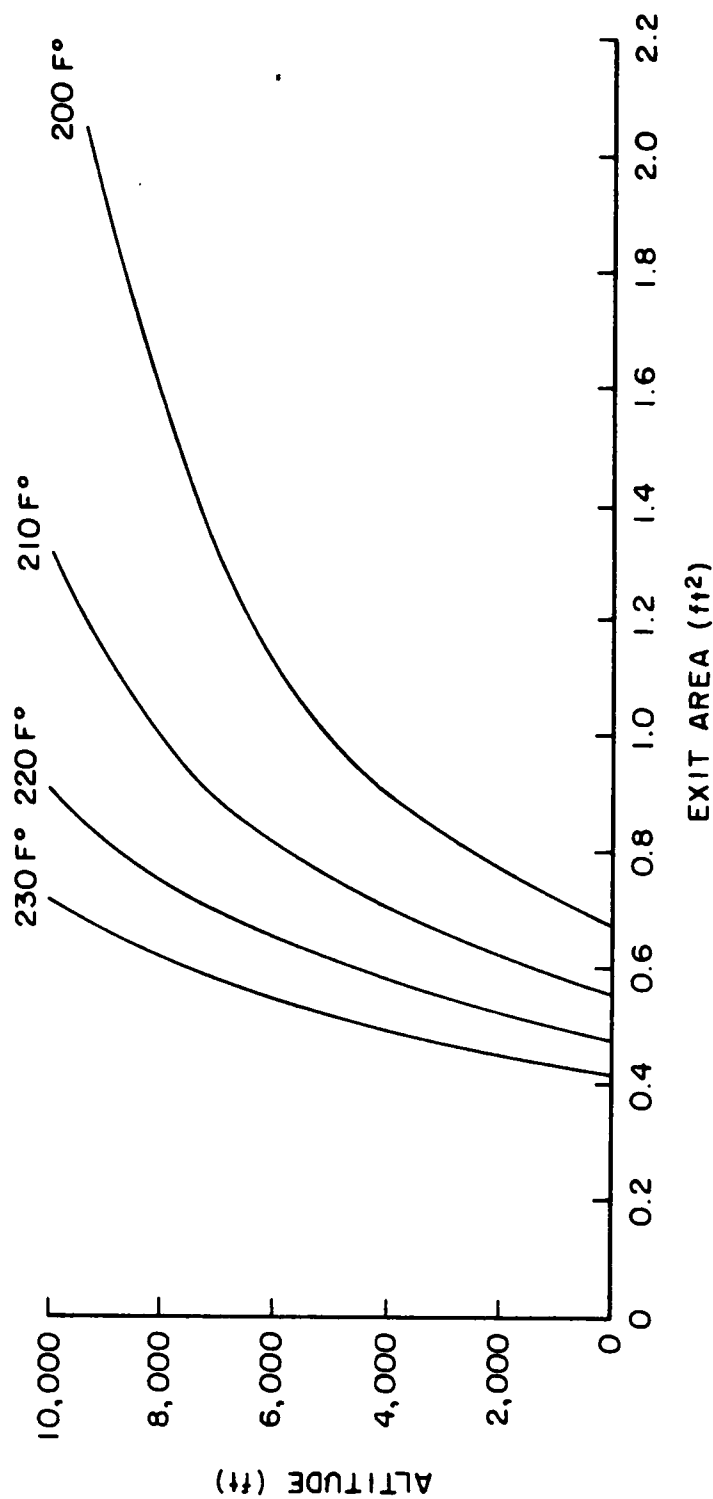


Figure 23. Effect of engine operating temperature on exit area required.

CLIMB AT 75 mph
 150 HP ENGINE
 INLET IN SLIPSTREAM
 OUTLET FLAP DEFLECTED 15°
 SURFACE 9.21-0.737 - SR

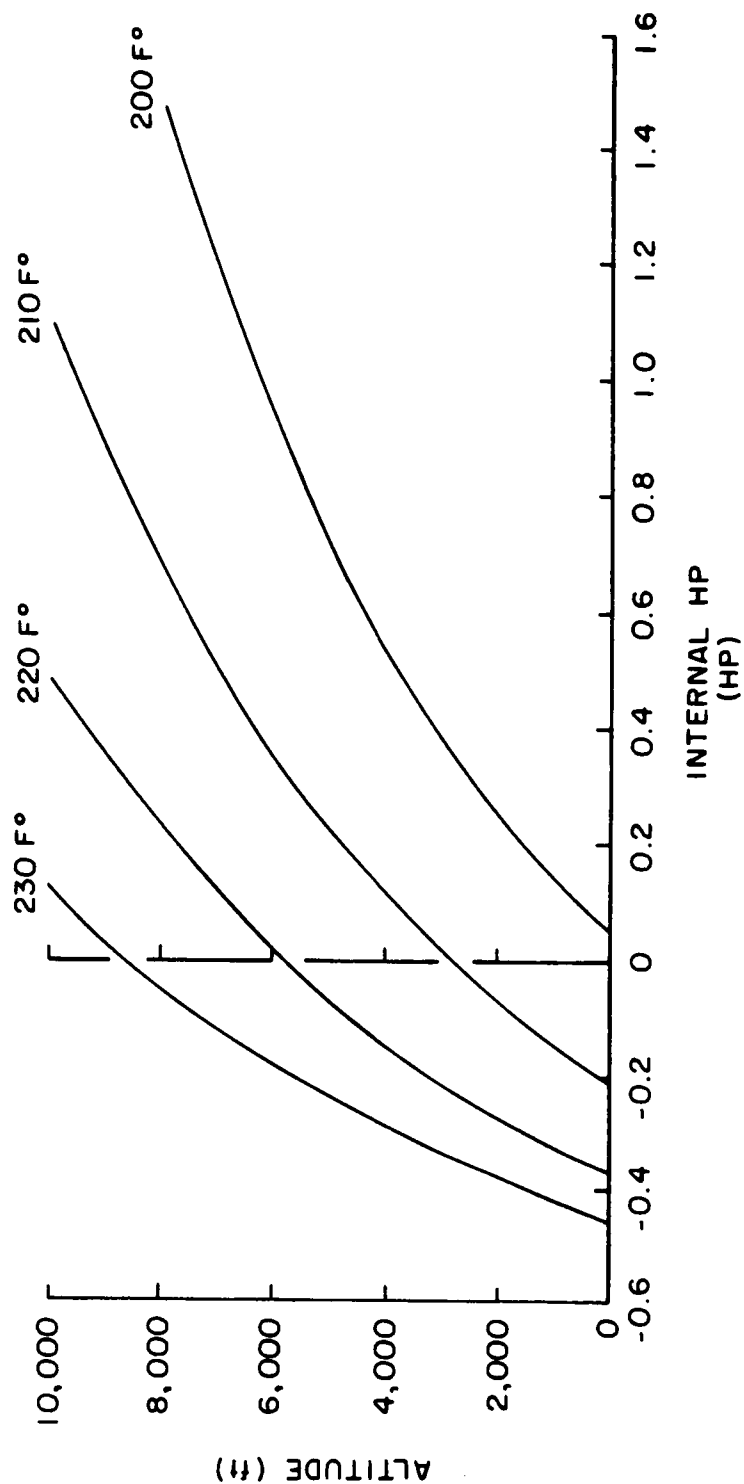


Figure 24. Effect of engine operating temperature on internal drag.

CLIMB AT 75 mph
 150 HP ENGINE
 INLET IN SLIPSTREAM
 OUTLET FLAP DEFLECTED 15°
 SURFACE 9.21-0.737-SR

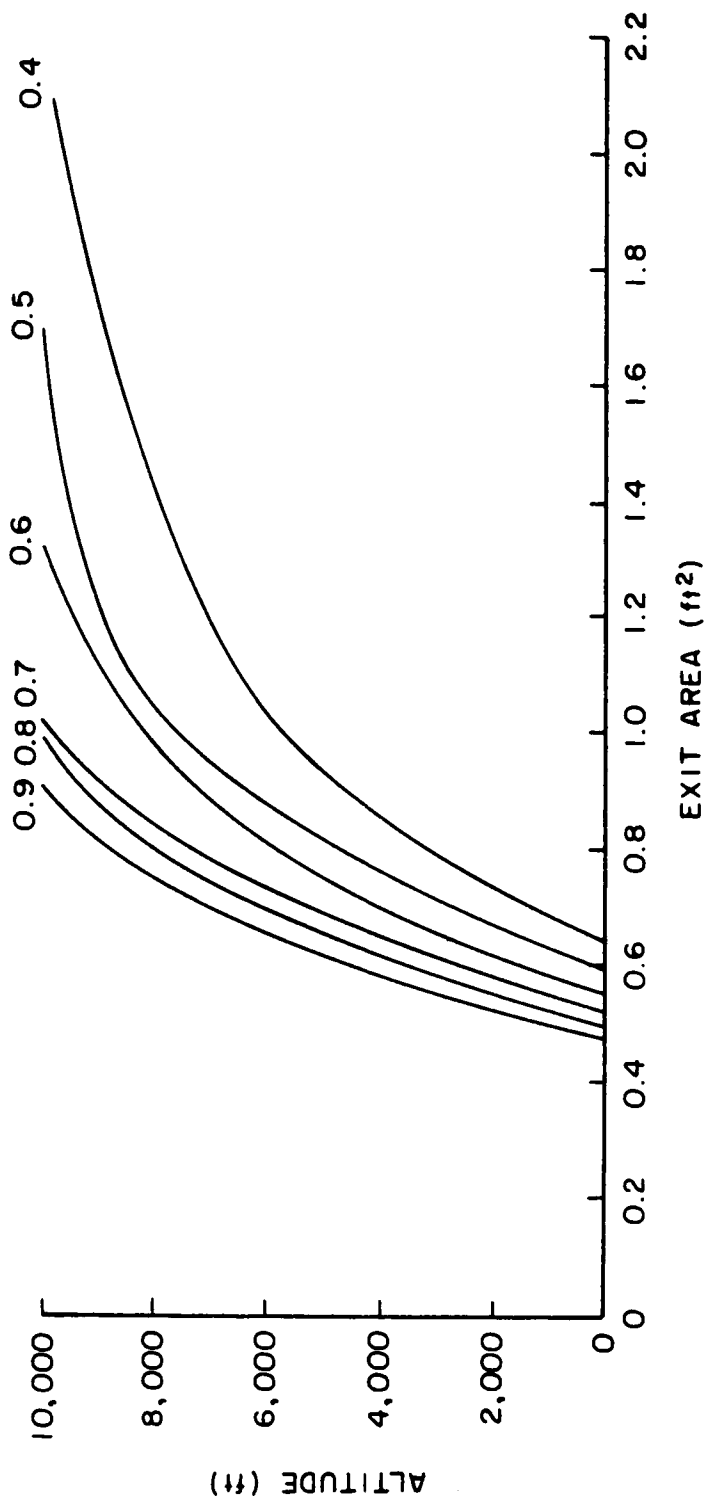


Figure 25. Effect of diffuser efficiency on exit area.

CLIMB AT 75 mph
 150 HP ENGINE
 INLET IN SLIPSTREAM
 OUTLET FLAP DEFLECTED 15°
 SURFACE 9.21-0.737-SR

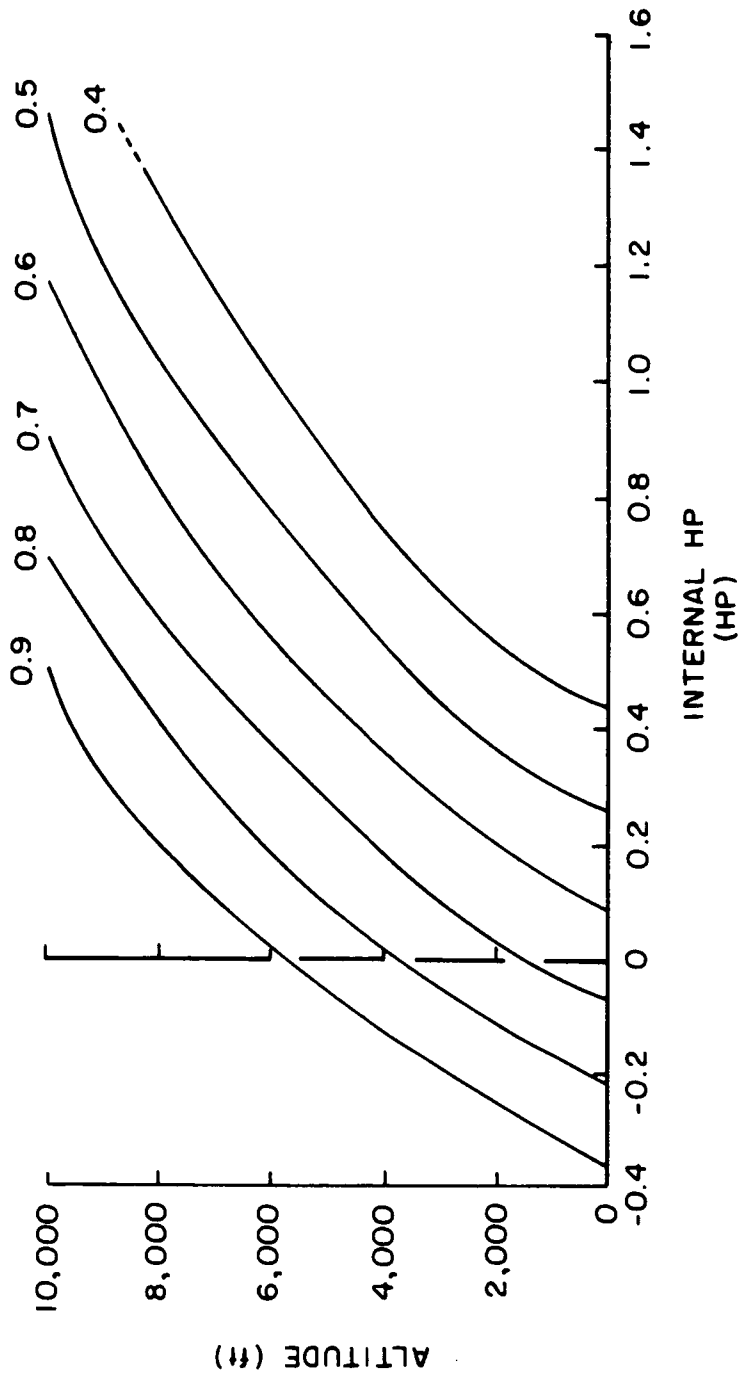


Figure 26. Effect of diffuser efficiency on internal drag.

It is shown in Figures 25 and 26 that exit areas and internal drag are reduced with increasing diffuser efficiency. From these curves it appears that the diffuser should be at least 70 percent efficient. Lower than this results in large increases in the exit area and the drag.

Numerous runs indicate that the worst conditions for design is the climb condition. This condition uses the maximum horsepower at a minimum velocity. In most cases the pressure drop of the core exceeds the pressure drop available from the free stream, therefore, the entry should be located so as to use the increased pressure available from the propeller slipstream, and an outlet flap deflected into the slipstream to reduce the static pressure at the outlet. For these reasons wing installations do not appear to be attractive.

Conceptual Designs

Computer runs for the climb conditions and high speed condition are shown in Tables I and II.

The inlet area vs. velocity ratio for these two conditions is shown in Figure 27. From this curve an inlet area of 0.7 ft^2 is selected. This results in a velocity ratio of 0.9 for the climb condition and 0.4 for the high speed condition.

Two conceptual designs based on the sizing data generated are shown in Figures 28 to 31. These depict a ring cooler installation (Figure 28) and a nose underslung installation (Figure 30). The ring cooler installation assumes a circular heat exchanger rather

Table I. Computer program output for climb condition.

CLIMB CONDITION 150HP

ENGINE HORSEPOWER..... 150.0 HP
ENGINE OPERATING TEMPERATURE....220.00 DEG F
MAXIMUM OPERATING ALTITUDE.....10000.00 FT
FLIGHT VELOCITY..... 75.00 MPH
DIFFUSER EFFICIENCY..... .90

*****HEAT EXCHANGER DETAILS*****

SURFACE TYPE IS 9.21-0.737-SR RUFFLED STAGGERED

AIRSIDE FRONTAL AREA.... 1.39SQ-FT

COOLANT SIDE FRONTAL AREA.... .56SQ-FT

CORE DIMENSIONS

HEIGHT= .83

WIDTH= 1.67

DEPTH= .33

HEAT EXCHANGER VOLUME= .463CU-FT

HEAT EXCHANGER WEIGHT= .0669LBS

ALTITUDE	MASS FLOW (LB/SEC)	VOLUMETRIC FLOW (CU FT/SEC)	PRESSURE DROP (PSF)	EXIT AREA (SQ FT)	INTERNAL HP (HP)
.0	3.93	59.48	5.19	.48	-.36
1000.0	3.90	60.88	8.38	.50	-.32
2000.0	3.90	62.71	8.67	.52	-.27
3000.0	3.90	64.68	9.00	.55	-.20
4000.0	3.90	66.74	9.34	.58	-.14
5000.0	3.90	68.89	9.70	.62	-.06
6000.0	3.90	71.12	10.08	.66	.02
7000.0	3.90	73.45	10.48	.70	.12
8000.0	3.90	75.88	10.89	.76	.23
9000.0	3.90	78.41	11.32	.82	.35
10000.0	3.90	81.05	11.77	.91	.49

Table II. Computer program output for high speed condition.

HIGH SPEED CONDITION 150HP

ENGINE HORSEPOWER..... 150.0 HP
ENGINE OPERATING TEMPERATURE.... 220.00 DEG F
MAXIMUM OPERATING ALTITUDE..... 10000 00 FT
FLIGHT VELOCITY..... 180.00 MPH
DIFFUSER EFFICIENCY..... .90

*****HEAT EXCHANGER DETAILS*****

SURFACE TYPE IS 9.21-0.737-SR RUFFLED STAGGERED

AIRSIDE FRONTAL AREA.... 1.39SQ-FT

COOLANT SIDE FRONTAL AREA.... .56SQ-FT

CORE DIMENSIONS

HEIGHT= .93

WIDTH= 1.67

DEPTH= .33

HEAT EXCHANGER VOLUME= .463CU-FT

HEAT EXCHANGER WEIGHT= .0669LBS

ALTITUDE	MASS FLOW (LB/SEC)	VOLUMETRIC FLOW (CU FT/SEC)	PRESSURE DROP (PSF)	EXIT AREA (SQR FT)	INTERNAL HP (HP)
.0	4.13	59.80	8.75	.20	-1.83
1000.0	4.09	60.85	8.96	.17	-5.62
2000.0	4.09	62.66	9.27	.18	-5.57
3000.0	4.09	64.61	9.62	.18	-5.53
4000.0	4.09	66.65	9.98	.19	-5.48
5000.0	4.09	68.76	10.37	.20	-5.43
6000.0	4.09	70.97	10.77	.20	-5.37
7000.0	4.09	73.27	11.19	.21	-5.31
8000.0	4.09	75.66	11.63	.22	-5.24
9000.0	4.09	78.15	12.09	.22	-5.16
10000.0	4.09	80.75	12.57	.23	-5.08

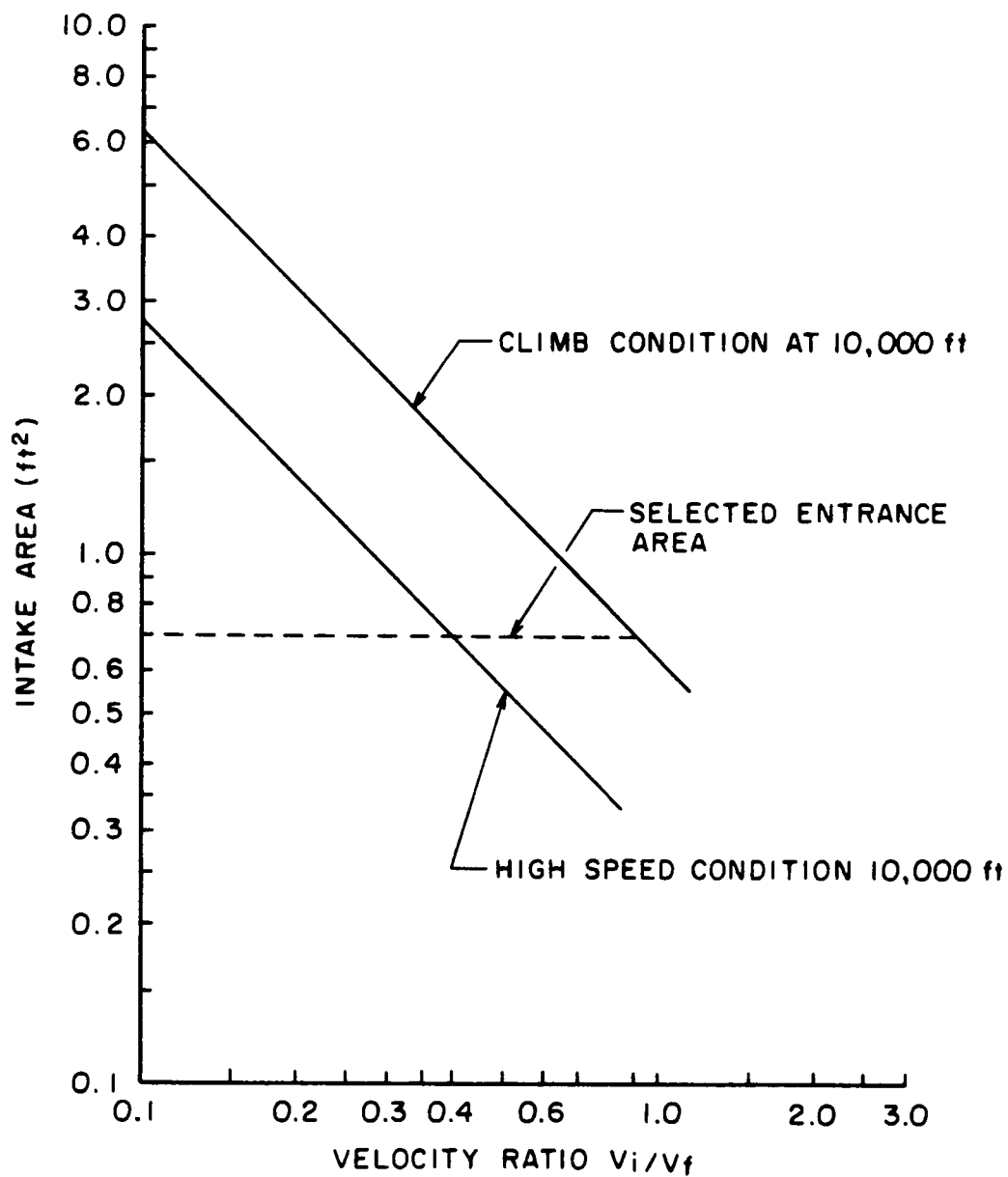


Figure 27. Inlet area sizing curve.

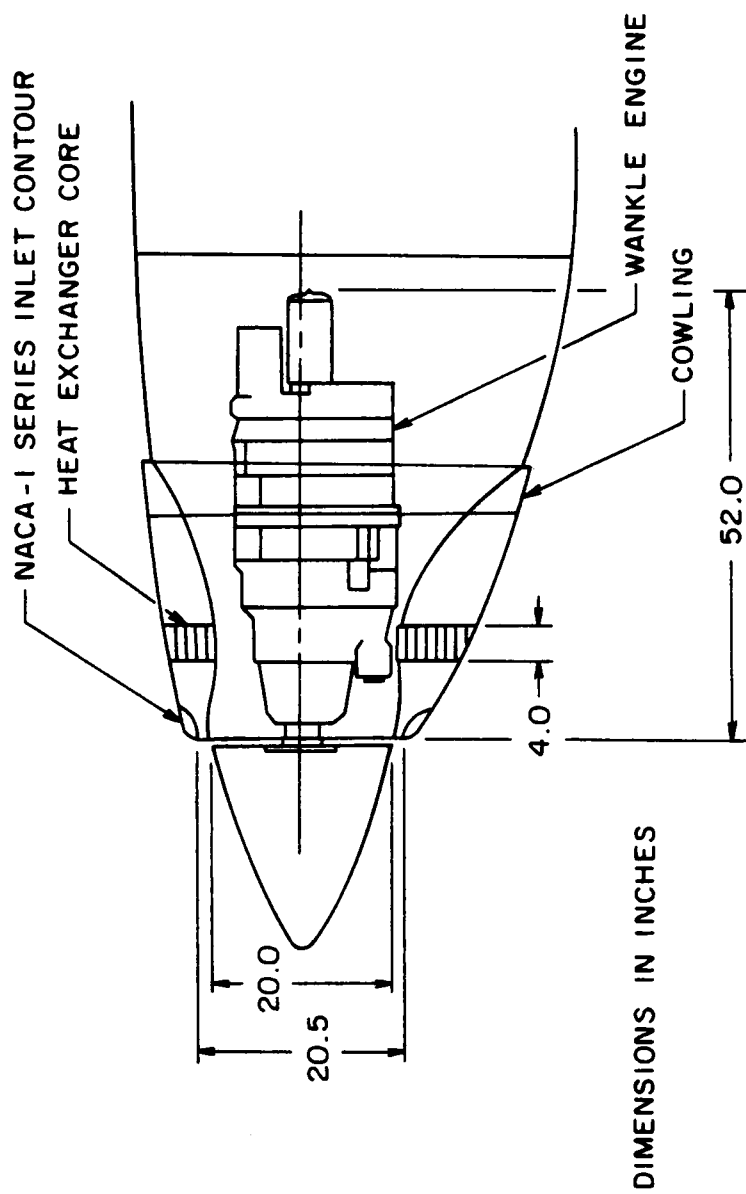


Figure 28. 150 HP wankle installation ring cooler side view.

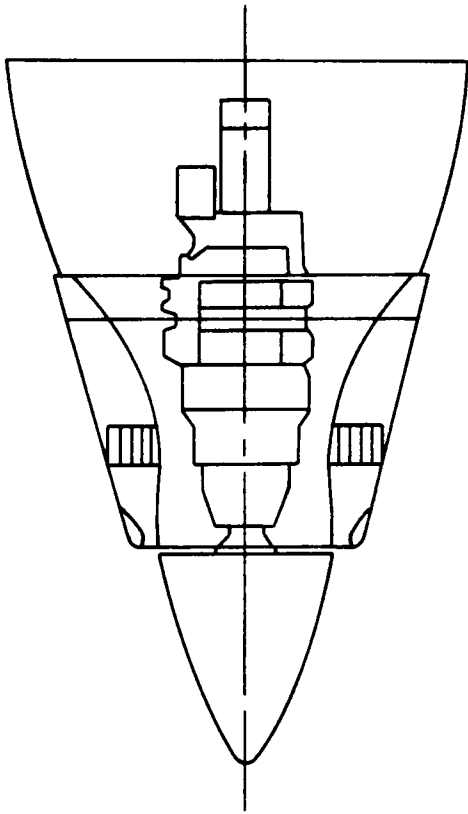


Figure 29. 150 HP wankle installation ring cooler plan view.

ENTRANCE AREA $\approx 0.7 \text{ ft}^2$
HEAT EXCHANGER CORE AREA $\approx 1.29 \text{ ft}^2$

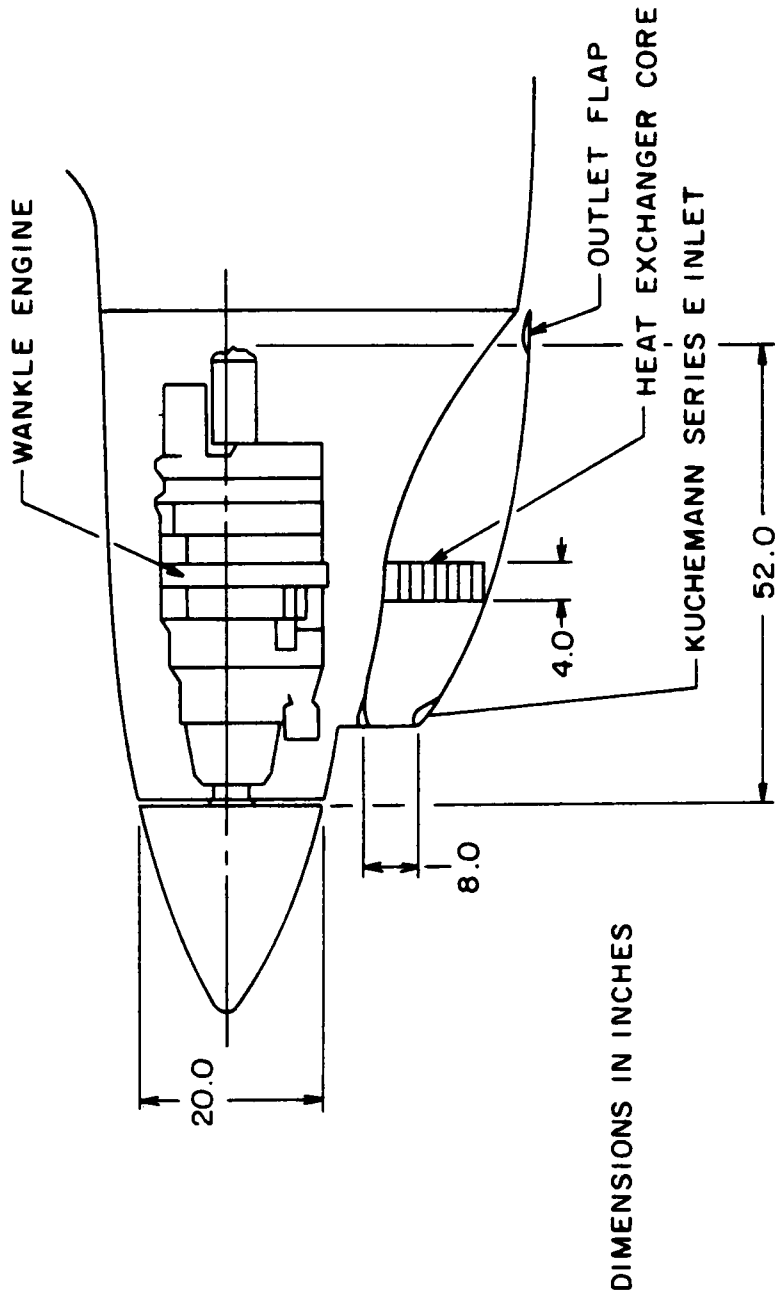


Figure 30. 150 HP wankle installation, underslung installation side view.

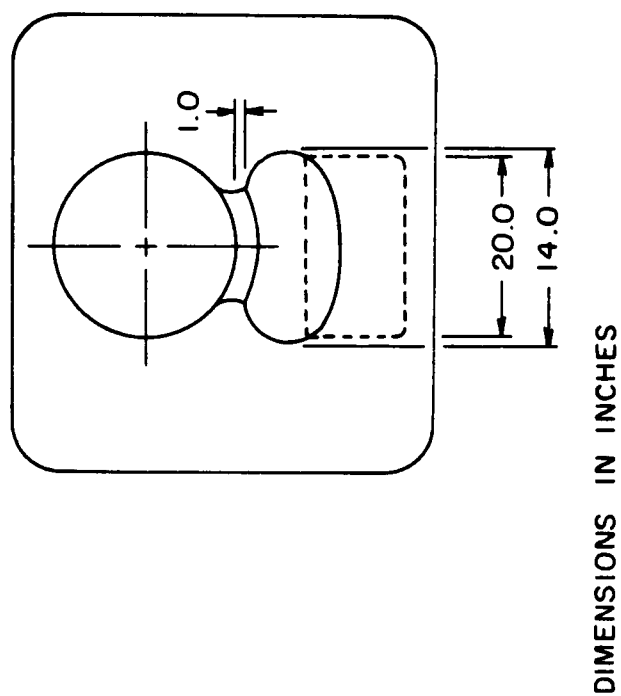


Figure 31. 150 HP wankle installation, underslung installation front view.

than rectangular, however, the frontal area and depth remain the same, and the entrance is sized to provide the 0.7 ft^2 of entrance area. The nose underslung installation uses the rectangular core shown in Figure 22.

The analysis indicates that at low altitudes some small thrust may be generated. This may be the case, however, the external drag may also be higher resulting in some net drag.

CHAPTER XI

CONCLUSIONS

Liquid cooled engine installations can be designed for general aviation aircraft that use 2 percent or less of the engine shaft horsepower to achieve adequate cooling. This could significantly reduce the aircraft drag allowing higher performance with no increase in engine horsepower. This would also result in better specific fuel consumption and more economical aircraft operation.

The engine operating temperature should be made as high as possible. The curves shown in Figures 23 and 24 show the performance as a function of the engine operating temperature. It can be seen that there is a significant increase in the volumetric flow required (i.e., increase in the exit area) and an increase in the drag horsepower with decreasing engine operating temperature.

Attention should be paid to the design of the diffuser. The curves shown in Figures 25 and 26 show the increase in volumetric flow required and the increase in drag with decreasing diffuser efficiency. These curves indicate that diffuser efficiencies should be between 0.7 and 0.9. Efficiencies below about 0.7 indicate a rapid increase in drag with altitude.

The heat exchanger should be designed for the climb condition. This condition requires the maximum horsepower output of the engine at the lowest operating speed.

The low power installations and possible extended ground operations encountered in general aviation aircraft will require the heat exchanger to be located in the propeller slipstream and use an 91

outlet flap deflected into the slipstream to provide sufficient ΔP to overcome the losses in the heat exchanger installation in climbing or low speed flight. The outlet flap should not be deflected more than 15 degrees since deflections greater than this increase the external drag without increasing the ΔP significantly. Conditions requiring flap deflections of over 15 degrees indicate that the heat exchanger core area is too small, and should be increased. The most effective method of increasing the core area will be to increase the depth.

Wing installations in general aviation aircraft will be difficult. This is due to the relatively thin wings encountered and the structural design of most general aviation aircraft wings. The heat exchanger installation will require large cutouts in the spar requiring structural redesign of the wing. If these designs are considered, the intakes should be located behind the propeller arc for reasons outlined above. This may be difficult to achieve in a high wing installation. Attention should also be paid to the location of the outlet since the aerodynamic characteristics of the wing can be adversely affected. It may be necessary to consider the use of thicker wing sections such as the GAW-1.

The fuselage installation appears to offer the best compromise for the installation in single engine GA aircraft and nacelle installation for twins. This allows the inlet to be located easily in the propeller slipstream and a minimum of frontal area and a minimum amount of weight.

The velocity ratio for the intake should not be below about 0.4, and ideally should be about 0.6. Velocity ratios lower than about

0.4 will induce separation of the flow ahead of the inlet and lead to low diffuser efficiencies.

Accurate sizing of the heat exchanger is difficult because accurate heat rejection values for liquid cooled engines suitable for general aviation aircraft is not available.

Prediction of the total drag requires knowledge of the external flow which cannot be predicted with sufficient accuracy by simple analytical methods.

Development of liquid cooled engines and engine installation appears to have ceased in the late 1940's, and therefore recent advances in heat transfer technology have been ignored.

CHAPTER XII

RECOMMENDATIONS

Test data is severely lacking for various parameters used in the design of liquid cooled installation. It is recommended that test installations be built and flown to determine values for various parameters to be used in design. Wind tunnel studies should be used to predict the external drag of specific installations.

Testing needs to be done to determine the optimum radial position of the inlet, with respect to the propeller arc, and optimum cowl shape. This could improve the dynamic pressure recovery while reducing external drag.

Engines should be tested to determine the actual heat rejection values. Automotive engines should also be tested to determine this variable. Wankle engines should be compared to piston engines to determine if the heat rejection rates vary significantly with engine type.

Engine operating temperature should be investigated. It may be possible, through modifications, to raise the operating temperature. The 220 °F assumed in the study could possibly be raised to 250 °F with corresponding gains in efficiency.

The installations of this study were based on existing compact heat transfer geometry, i.e., surfaces with fins. It would be interesting to determine if recent advances in technology could be incorporated in the design. Perhaps a better heat transfer fluid exists than glycol and water, or perhaps heat pipe technology could be used to transport heat from the coolant more effectively.

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Becker, John V., "Wind Tunnel Tests of Air Inlet and Outlet Openings on a Streamline Body," NACA L-300, 1940.
2. Blackaller, D. L., "Development and Application of a Liquid Cooled V-8 Piston Engine for General Aviation Aircraft," AIAA-83-1842, New York, American Institute of Aeronautics and Astronautics, June 1983.
3. Becker, John V. and Baals, Donald D., "Simple Curves for Determining the Effects of Compressibility on Pressure Drop Through Radiators," NACA L-6, 1944.
4. Baals, Donald D., Smith, Norman F., and Wright, John B., "The Development and Application of High-Critical Speed Nose Inlets," NACA TR-920, 1945.
5. Boswinkle, Robert W., Jr., "Air Flow Surveys in the Vicinity of Representative NACA 1-Series Cowlings," NACA RM L8A15a, 1948.
6. Boswinkle, Robert W., Jr. and Bryant, Rosemary P., "An Experimental Investigation of Flow Conditions in the Vicinity of an NACA D_s - Type Cowling," NACA L-747, 1946.
7. Brevoort, M. J., "Radiator Design," NACA L-233, 1941.
8. Brevoort, Maurice J., Joyner, Upshort, and Wood, George P., "The Effect of Altitude on Cooling," NACA L-386, 1943.
9. Brimly, D. E. and Brevoort, Maurice J., "Correlation of Engine-Cooling Data," NACA L-685, 1945.
10. Coe, P. L., Jr., "Review of Drag Cleanup Tests in Langley Full-Scale Tunnel (from 1935 to 1945) Applicable to Current General Aviation Airplane," NASA TNP-8206, June 1976.
11. Columbus Division, North American Rockwell Corporation, "Internal Aerodynamics Manual," June 1970.
12. Czarnecki, K. R. and Nelson, W. J., "Wind Tunnel Investigation of Rear Underslung Fuselage Ducts," NACA L-438, 1943.
13. Davis, Wallace F. and Sclerrer, Richard, "Aerodynamic Principles for Re Design of Jet-Engine Induction Systems," NACA RM A55F16, 1956.

14. Dearborn, C. H. and Silverstein, A., "Drag Analysis of Simple-Engine Military Airplane Tested in the NACA Full-Scale Wind Tunnel," NACA WR-L-439, October 1940.
15. Donovan, A. F. and Lawrence, H. R., Aerodynamic Components of Aircraft At High Speeds, Princeton University Press, Princeton, New Jersey, 1957, pp. 586-658.
16. Dommasch, D. O., Sherby, Sydney S. and Connolly, T. F., Airplane Aerodynamics, Third Edition, New York, 1961.
17. Ellerbrock, H. H., Jr. and R. O. Bullock. "Cooling and Performance Tests of a Continental A-75 Engine," NACA TN 816, Langley Memorial Aeronautical Laboratory, July 1941.
18. Ellis, D. L., "Cooling and Cowlings," The Aeroplane, July 12, 1940, pp. 40-42.
19. Ellis, D. L., "Cooling and Cowlings," The Aeroplane, July 19, 1940, pp. 73-75.
20. Flader F. and Child, E. R., "Studies of High Speed Airplanes," Journal of The Aeronautical Sciences, Vol. 7, pp. 235-243, November 1939, October 1940.
21. Fraas, P. Arthur and Obisik, M. N., Heat Exchanger Design, John Wiley & Sons Inc., New York, 1965.
22. Foxworth, Thomas G., The Speed Seekers, New York: Doubleday & Company Inc., 1934.
23. Gothert, B., "The Drag of Airplane Radiators With Special Reference to Air Heating (Comparison of Theory and Experiment)," NACA TM-896, May 1939.
24. Harris, Thomas A. and Recant, Isidone G., "Investigation in the 7-by 10-Foot Wind Tunnel of Ducts for Cooling Radiators Within an Airplane Wing," NACA TR-743, 1938.
25. Henry, John R., "Design of Power Plant Installations Pressure-Loss Characteristics of Duct Components," NACA L-208, 1944.
26. Hill, Philip G. and Porterson, Carl R., Mechanics and Thermodynamics of Propulsion, Hodison-Wesley Publishing Co., New York, 1965.
27. Hoerner, Sigward, F., Fluid Dynamic Drag, published by author, 1965.
28. Huggins, G. L. and Ellis, D. R., "Advanced General Aviation Comparative Engine/Airframe Integration Study," (Cessna Aircraft) NASA CR-165564, September 1981.

29. Johnson, C. L., "The Design of High-Speed Military Airplanes," *Journal of The Aeronautical Sciences*, Vol. 8, No. -4, pp. 467-474.
30. Kays, W. M. and London, A. L., Compact Heat Exchangers, McGraw-Hill Book Co., New York, 1964.
31. Katsoff, S., "The Desin of Cooling Ducts with Special Reference to the Bounday Layer at the Inlet," NACA L-321, 1940.
32. Katsoff, S., "High Altitude Cooling V - Cowling and Ducting," NACA L-775, 1944.
33. Katz, J., Corsiglia, V. R. and Barlow, P. R., "Effect of Propellor Slipstream on the Drag and Performance of the Engine Cooling System for a General Avaiaation Twin-Engine Aircraft," AIAA-80-1972, New York, New York, American Institute of Aeronautics and Astronautics, August 1980.
34. Kern, Donald Q. and Kraus, Allan D., Extended Surface Heat Transfer, McGraw-Hill Book Co., New York, 1972.
35. Kreith, Frank, Principles of Heat Transfer, Second Edition, International Test Book Co., Scranton, Pennsylvania, 1965.
36. Kreul, A. L., "Design of Radiators for Automotive and Industrial Engines," SP-122, SAE, New York, 1954.
37. Kuchemann, Dietrick. and Weber, Johanna, Aerodynamics of Propulsion, McGraw-Hill Book Co., Inc., 1953.
38. Linke, W., "Experimental Investigations on Freely Exposed Ducted Radiators," NACA TM-920, March 1941.
39. Litkin, S. G., "Radiator Research in the U.S.S.R., Some Russian Investigations into the Problems of Wing Radiators," Aircraft Engineering, May 1940.
40. Lundin, Bruce T., Povolny, John H., Chelko, Louis J., "Correlation of Cylinder-Head Temperatures and Coolant Heat Rejections of a Multicylinder, Liquid-Cooled Engine of 1710-Cubic-Inch Displacement," NACA TR-931, 1948.
41. Meredith, F. W., "Ducted Radiators, The Theory of Low Velocity Cooling, With a Base of Calculating the Drag," Aircraft Engineering, Vol. VIII, August 1936, pp. 218-220.
42. McLellan, Charles H. and Nichols, Mark R., "An Investigation of Diffuser-Resistance Combinations in Duct Systems," NACA L-329, 1942.

43. Miley, S. J., Cross, E. J. and Lawrence D. L., "Flight Test Method for the Determination of Reciprocating Engine Cooling Requirements," AIAA-81-2446, New York, New York: American Institute of Aeronautics and Astronautics, November 1981.
44. Miley, S. J., Cross, E. J., Owens, J. K. and Lawrence, D. L., "An Experimental Investigation of the Aerodynamics and Cooling of a Horizontally-Opposed Aircooled Aircraft Installation," NASA CR 3405, March 1981.
45. Nelson, W. J. and Czarnecki, K. R., "Wind Tunnel Investigation of Wing Ducts on a Single-Engine Pursuit Airplane," NACA L-407, 1943.
46. Nelson, W. J. and Czarnecki, K. R. "Wind-Tunnel Investigation of Carburetor-Air Inlets," NACA L-166, 1942.
47. Nichols, Mark R., Keith, J. R. and Arvid, L., "Investigation of a Systematic Group of NACA 1 Series Cowlings with and without Spinners," NACA TR 950, 1948.
48. Nichols, Mark R., "Investigation of Flow Through an Intercooler Set at Various Angles to the Supply Duct," NACA L-408, 1942.
49. Nicolas, Leland, M., Fundamentals of Aircraft Design, University of Dayton, Dayton, Ohio, 1975.
50. Nickle, F. R. and Freeman, Arthur B., "Full-Scale Wind-Tunnel Investigation of Wing Cooling Ducts," NACA ACR, October 1933.
51. Nielsen, Jack N., "High Altitude Cooling III Radiators," NACA L-773, 1944.
52. Patterson, G. H., "Re Design of Aeroplane Ducts," Aircraft Engineering, July 1939, pp. 263-268.
53. Peters, H., "Conversion of Energy in Cross-Sectional Divergences Under Different Conditions of Inflow," NACA TM-737.
54. Pettitt, Herriot J., "The Installation of a Racing Engine," Aircraft Engineering, Vol. IV, No. 38, April 1932, pp. 91-95.
55. Pinkel, Benjamin, Manganiello, Eugene J. and Bernardo, Everett,, "Cylinder-Temperature Correlation of a Single-Cylinder Liquid-Cooled Engine of 1710-Cubic Inch Displacement," NACA TR-853, 1946. 1948.
56. Povolny, John H. and Clelko, Louis, J., "Cylinder-Head Temperatures and Coolant Heat Rejections of a Multicylinder, Liquid-Cooled Engine of 1710-Cubic Inch Displacement, NACA TN-1606, 1948.

57. "Proceedings of the NASA, Industry, University General Aviation Drag Reduction Workshop," University of Kansas, Lawrence, Kansas, NASA-CR-145627, July 1925.
58. Rauscher, M. and Phillips, W. H., "Propulsive Effects of Radiator and Exhaust Ducting," Journal of the Aeronautical Sciences, February 1941, Vol. 8, No. 4, pp. 167-174.
59. "Rewriting the GA Engine Book," Flight International June 15, 1905, p. 10.
60. Rogallo, F. M., "Internal-Flow Systems for Aircraft," NACA TR-713, 1939.
61. Rubert, Kennedy F. and Knope, George S., "A Method for the Design of Cooling Systems for Aircraft Power-Plant Installation," NACA L-491, 1942.
62. Sacks, Alvin H. and Spreiter John R., "Theoretical Investigation of Submerged Inlets at Low Speeds," NACA TN-2323, 1951.
63. Schlaifer, Robert and Heron, S. D., Development of Aircraft Engines and Fuels, Boston: Graduate School of Business Administration, Harvard University, 1950.
64. Schubauer, G. B. and Spangenberg, W. G., "Effect of Screens in Wide-Angle Diffusers," NACA TR-949, 1947.
65. Silverstein, A. and Wilson, H. A., Jr., "Aerodynamic Characteristics of a 4-Engine Monoplane Showing Effects of Enclosing the Engines in the Wing and Comparisons of Tractor and Pusher-Propellor Arrangements," NACA WR-L-456, April 1938.
66. Silverstein, A. and Wilson, H. A., Jr., "Aerodynamic Characteristics of a 4-Engine Monoplane Showing Comparison of Air-cooled and Liquid cooled Installations," NACA WR-L-471, July 1939.
67. Silverstein, A. and Guryansky, E. R., "Development of Cowling for Long-Nose Air Cooled Engine in the NACA Full-Scale Wind Tunnel," NACA WR-L-241, October 1941.
68. Silverstein, Abe and Wilson, Herbert A., Jr., "Cooling in Cruising Flight with Low Fuel-Air Ratios," NACA L-570, 1942.
69. Silverstein, Abe, "High Altitude Cooling, I Resume of the Cooling Problem," NACA L-771, 1944.
70. Smith F. Norman and Baals Donald D., "Wind-Tunnel Investigation of a High-Critical-Speed Fuselage Scoop Including the Effects of Boundary Layer," NACA L-733, 1945.

71. Smith, R. H., "Resistance and Cooling Power of Various Radiators," NACA TR-261, 1921.
72. Stewart, W. L., Weber, R. J., Willis, E. A. and Sievers, G. E., "NASA Research on General aviation Power Plants," AIAA-79-0561, New York, New York, American Institute of Aeronautics and Astronautics, February 1979.
73. Stickle, George W., Nainian, Irven and Crigler, John L., "Pressure Available for Cooling with Cowling Flaps," NACA TR-720, 1940.
74. Theodorsen, T. and Brevoort, M. J. and Stickle, G. W., "Cooling of Airplane Engine at Low Airspeed," NACA TR-593, 1937.
75. Weise, A., "The Conversion of Energy in a Radiator," NACA TM-869, July 1938.
76. Winter, H., "Contribution to the Theory of the Heated Duct Radiator," NACA TM-893, April 1939.
77. Wood, George, P., Brevoort, Maurice J., "Design, Selection, and Installation of Aircraft Heat Exchangers," NACA L-341, 1943.
78. Zack, Marv, "Where Are Our Wankels," Airprogress, Vol. 36, No. 9, September 1976, pp. 11-15.
79. Zmroczet, L. A., "Advanced General Aviation Comparative Engine Airframe Integration Study," NASA CR-165565 (Beech Aircraft), March 1982.

APPENDICES

APPENDIX I

ALTITUDE CONSIDERATIONS: (Cooling Air Flow, Ref. 69)

APPENDIX I

Altitude Considerations: (Cooling air flow, Ref. 69)

The heat transferred from either a direct cooled (air-cooled) or indirect (liquid cooled engine with radiator) is proportional to the mass flow raised to some power and the difference between the wall temperature and the stagnation temperature of the cooling air at the inlet, i.e.,

$$Q \propto (T_w - T_a) M^k$$

where the constant k is approximately 0.85 for radiators and 0.65 for air-cooled engines.

If the heat transfer is constant, then

$$M \propto \frac{1}{(T_w - T_a)^{1/k}}$$

Since the ΔT , $(T_w - T_a)$ increases with altitude, (assuming the velocity does not increase significantly, the stagnation temperature decreases with altitude), the mass flow required for a given heat transfer decreases. This is shown in Figure 32. It is seen from this curve that the effect of altitude on the temperature differences is more pronounced for heat exchangers with low wall temperatures than for ones with high wall temperatures. For the radiator ($T_{WALL} \approx 240^\circ\text{F}$) the ΔT changes from 121°F at sea level to 240°F at 40,000 feet, the required mass flow at 40,000 feet is 47 percent of the required mass flow at sea level. For the air-cooled engine, however, ($T_{WALL} \approx 450^\circ\text{F}$) the ΔT changes from 330°F at sea level to 450°F at 40,000 feet and the required mass flow at 40,000 feet is 60 percent of that required at sea level.

Dividing the mass flow by the appropriate densities results in the required volume flow rate as a function of altitude. This is shown in Figure 33. This curve shows that the volume flow rate increases more rapidly for the air-cooled engine than for the radiator.

Pressure Drop Required for Cooling

If the density is assumed to remain constant through the heat exchanger, then the pressure drop for either air-cooled engines or radiators is approximately proportional to $M^{1.8}$. This is shown by the dashed lines of Figure 34.

The density, however, does indeed change significantly through the passage of the heat exchanger. This results in two additional sources of pressure drop.

Since the density decreases, there is an increase in velocity. This leads to an increase in friction, and a momentum loss that accompanies the acceleration. If the total pressure loss is corrected for these effects, the values of the solid lines of Figure 34 are obtained. The ratio of the corrected values to the uncorrected values are shown in Figure 35.

These curves indicate that as the altitude increases, the hotter the wall temperature of the heat exchanger and the higher the velocity of the cooling air, the greater will be the ratio of the pressure drop required at altitude to the pressure drop required at sea level. This indicates that the cooling of an air-cooled engine at high altitudes is more difficult than that of a liquid cooled engine radiator.

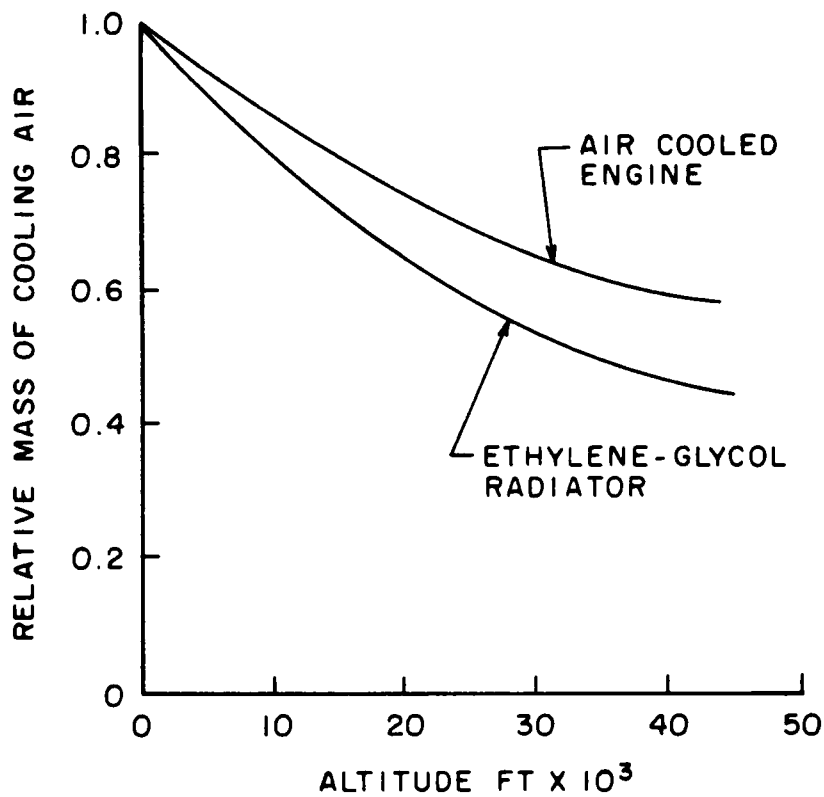


Figure 32. Altitude effect on the relative mass of required cooling air.

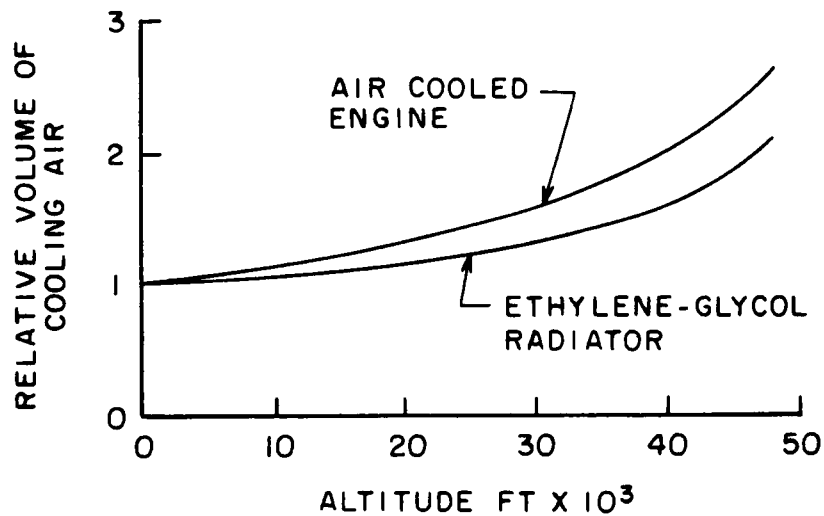


Figure 33. Altitude effects on the required volume of cooling air.

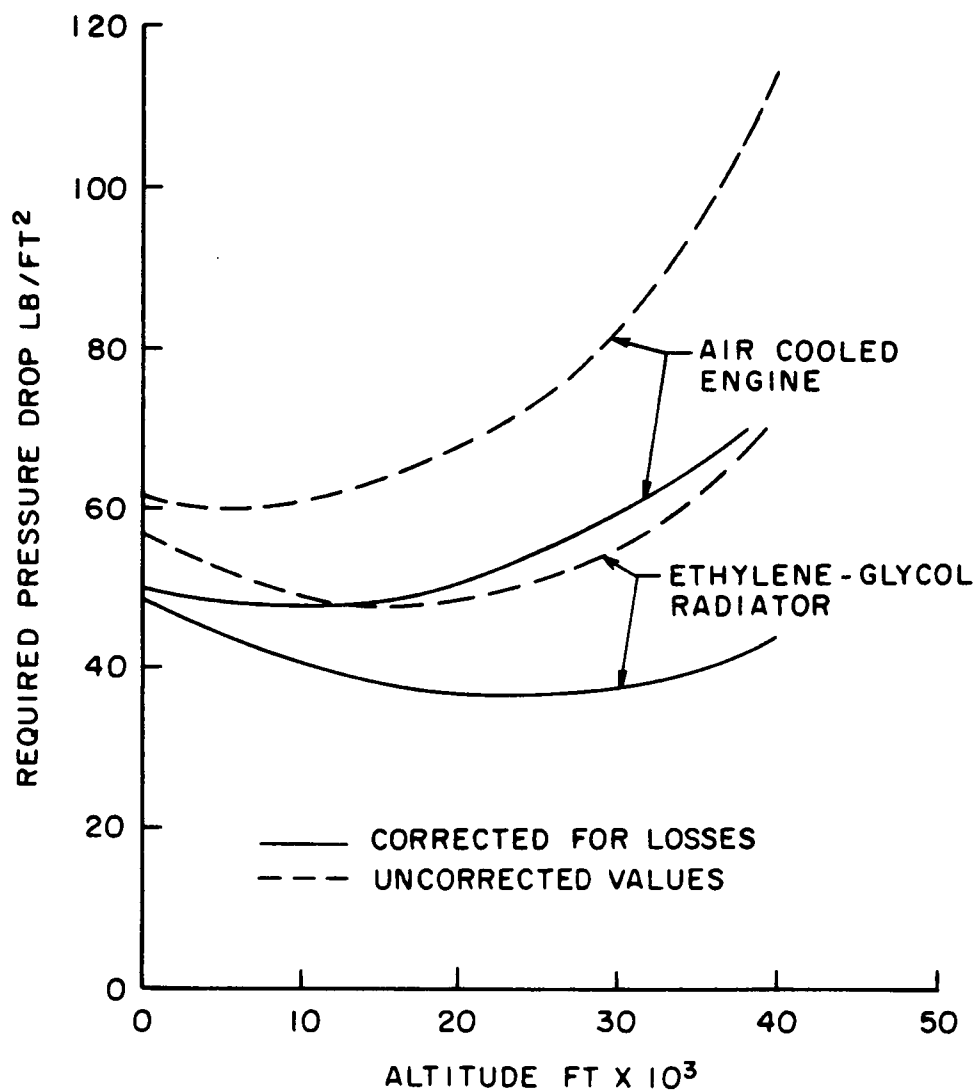


Figure 34. Effect of altitude and losses on required pressure drop.

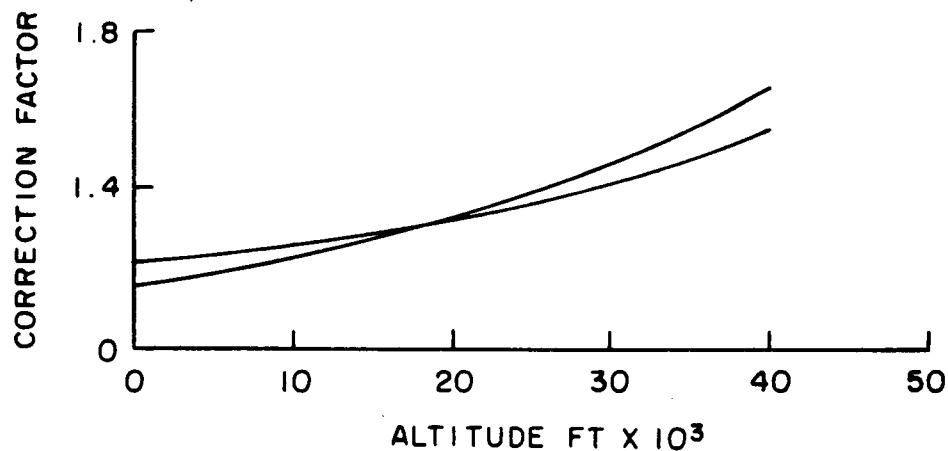


Figure 35. Ratio of corrected to uncorrected values as a function of altitude.

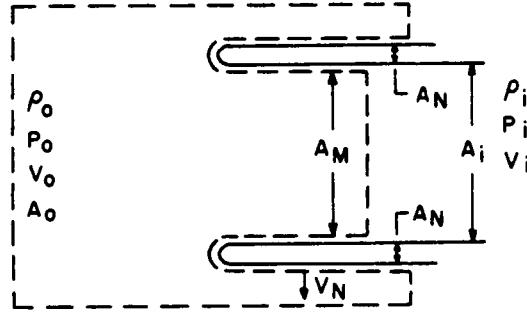
APPENDIX II

OPTIMUM INTAKE CONTOURS (Ref. 37)

APPENDIX II

OPTIMUM INTAKE CONTOURS (Ref. 37)

A control volume is defined as follows:



Integrating over the control volume surface

$$\rho_0 V_0 A_0 V_0 + P_0 A_0 - \rho_i V_i A_i V_i - P_i A_i - \int_{A_N} P_N dA_N - P_0 (A_0 - A_M) - (\rho_0 V_0 A_0 - \rho_i V_i A_i) V_0 = 0$$

Gathering terms and rearranging yields:

$$\int_{A_N} (P_N - P_0) dA_N = \rho_i V_i A_i (V_0 - V_i) - (P_i - P_0) A_i$$

Assume incompressible flow:

Then

$$\rho_0 = \rho_i = \rho$$

Using Bernoulli's equation

$$P_i - P_0 = \frac{\rho V_0^2}{2} - \frac{\rho V_i^2}{2}$$

The thrust force is given by:

$$F_N = - \int_{A_N} (P_N - P_O) dA_N = 1/2 \rho V_O^2 A_i \left(1 - \frac{V_i}{V_O} \right)^2$$

$$\frac{F_N}{\frac{\rho V_O^2 A_i}{2}} = \left(1 - \frac{V_i}{V_O} \right)^2$$

Since the inlet velocity, at least for normal operations is never less than zero, there is a positive thrust force on the intake for all mass flow conditions.

The thrust force on intakes is due entirely to the change in momentum of the inflow. The thrust force is independent of where the change takes place, i.e., in the free stream, inside the duct, or inbetween. Only the forces are displaced. The force diagram for various intakes is shown in Figure 36a,b,c.

The pressure coefficient (assuming incompressible flow is given by:

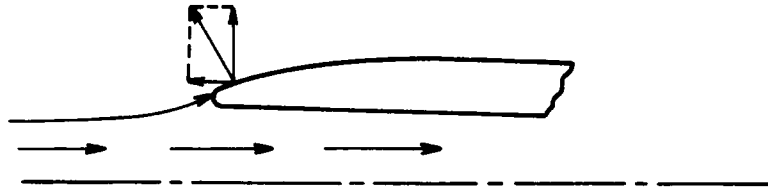
$$C_P = \frac{P_N - P_O}{\frac{\rho V_O^2}{2}}$$

Using Bernoulli's equation and substituting yields:

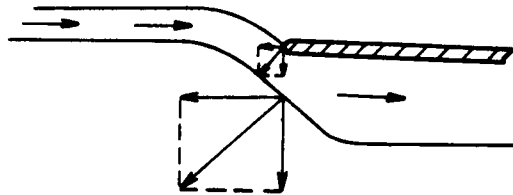
$$C_P = 1 - \left(V_N/V_O \right)^2$$

From this equation, it is seen that for negative pressure coefficients to exist, i.e., positive thrust, the velocity v_n on the outer surface of the intake will be locally greater than the free-stream velocity.

EXTERNAL TO THE DUCT



FLUSH INTAKES



PARTIAL INTAKES

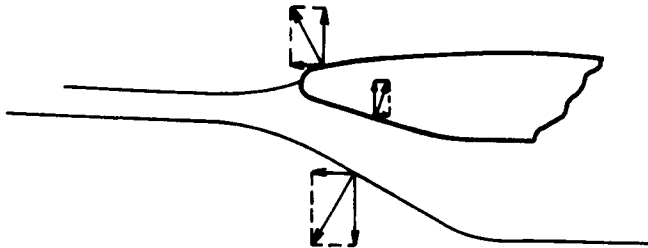


Figure 36. Thrust forces on intakes.

The idea for the optimum intake is to describe a shape so that the velocity V_n is uniform over the entire outer surface.

Contour Equations

The outer surface is calculated according to:

$$r/R_m = \frac{P_o}{R_m} + (1 - R_o/R_m) \sqrt{1 - (1 - R_m/L \cdot X/R_m)^2}$$

The inner surface shape is calculated according to:

$$r/R_m = \frac{R_o}{R_m} - \left(\frac{R_o}{R_i} - 1 \right) R_i/R_m \sqrt{1 - (1 - R_i/L_i \cdot X/R_i)^2}$$

Series Constants

Class	k_1	k_2	k_3	k_4
A	1.15	0.2	12.5	1.5
B	1.10	0.2	12.5	1.3
C	1.03	0.2	7.5	1.0

The inner and outer thicknesses and lengths of inner and outer curved portions are given in terms of the area ratio A_i/A_m .

The inner thickness is given by:

$$t_{inner} = R_o - R_i$$

$$R_o/R_i = k_1$$

The outer thickness is given by:

$$t_{outer} = R_m - R_o$$

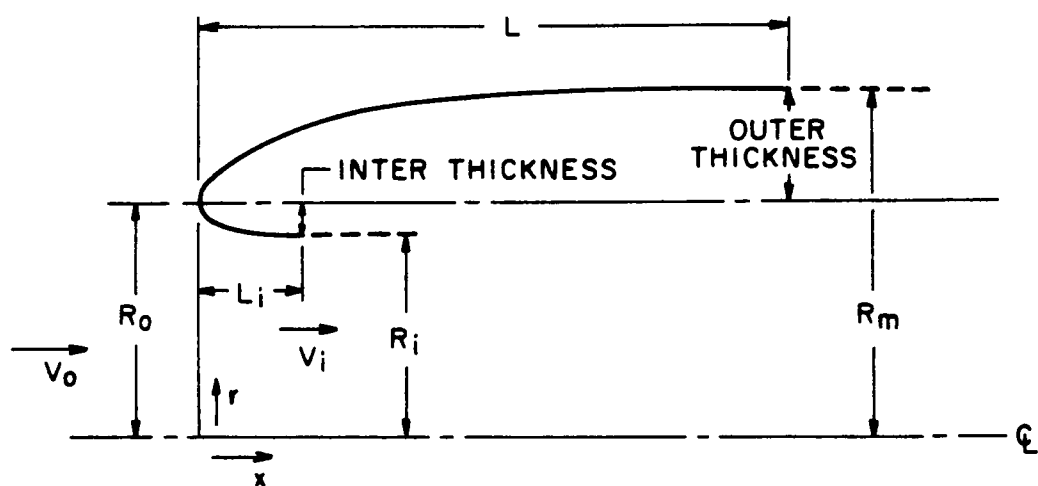


Figure 37. Kuchemann series inlet geometry.

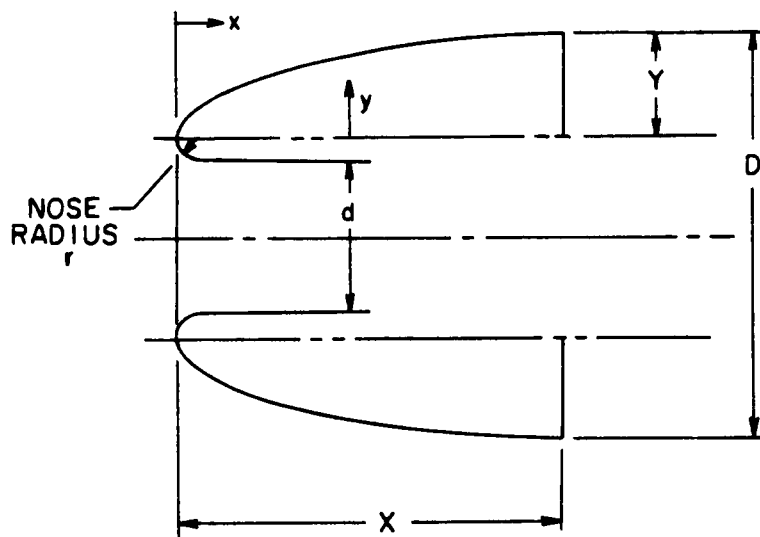
$$R_o/R_m = k_1 \sqrt{\frac{A_i}{A_m}}$$

The length of the outer surface is given by:

$$R_m/L = k_2 + k_3 (A_i/A_m)^3$$

The length of the inner surface is given by:

$$R_i/L_i = \frac{1}{k_4(k_1-1)}$$



$$X = \left(\frac{X}{D}\right) D \quad Y = \frac{D-d}{2} - r \quad \text{FOR } r = 0.025Y$$

$$Y = \frac{D-d}{0.05}$$

Figure 38. NACA-1 nomenclature.

Table III. NACA-1 ordinates.

X/X	X/Y	X/X	X/Y
0	0	34.0	69.08
0.2	4.80	35.0	70.08
0.4	6.63	36.0	71.05
0.6	8.12	37.0	72.00
0.8	9.33	38.0	72.94
1.0	10.38	39.0	73.85
1.5	12.72	40.0	74.75
2.0	14.72	41.0	75.63
2.5	16.57	42.0	76.48
3.0	18.31	43.0	77.32
3.5	19.94	44.0	78.15
4.0	21.48	45.0	78.95
4.5	22.96	46.0	79.74
5.0	24.36	47.0	80.50
6.0	27.01	48.0	81.25
7.0	29.47	49.0	81.99
8.0	31.81	50.0	82.69
9.0	34.03	52.0	84.10
10.0	36.13	54.0	85.45
11.0	38.15	56.0	86.73
12.0	40.09	58.0	87.95
13.0	41.94	60.0	89.11
14.0	43.66	62.0	90.20
15.0	45.30	64.0	91.23
16.0	46.88	66.0	92.20
17.0	48.40	68.0	93.11
18.0	49.88	70.0	93.95
19.0	51.31	72.0	94.75
20.0	52.70	74.0	95.48
21.0	54.05	76.0	96.16
22.0	55.37	78.0	96.79
23.0	56.66	80.0	97.35
24.0	57.92	82.0	97.87
25.0	59.15	84.0	98.33
26.0	60.35	86.0	98.74
27.0	61.52	88.0	99.09
28.0	62.67	90.0	99.40
29.0	63.79	92.0	99.65
30.0	64.89	94.0	98.85
31.0	65.97	96.0	99.93
32.0	67.03	98.0	99.98
33.0	68.07	100.0	100.0

APPENDIX III
FORTRAN PROGRAM

\$DEBUG

```

REAL LDR
DIMENSION ALTP(11,4),CFP(11,4),WA(11),RNA(11),ETAU(11),PRSDRP(11)
DIMENSION VEXIT(11),VFR(11),EXAREA(11),TITLE(20),PCI(11),S(13,5)
DIMENSION QO(11),DIFFP(11),AREAIN(11)
MVALUE1=0
MVALUE2=0
K=0
WRITE(*,*) 'INPUT ANALYSIS TITLE'
READ(*,90) TITLE
90 FORMAT(20A4)

```

C
C
C

THIS SECTION IS INPUT INFORMATION FOR OPERATING REQUIREMENTS

```

WRITE(*,*) 'ENTER THE ENGINE MAX OPERATING HORSEPOWER'
READ(*,*) HPR
WRITE(*,*) 'ENTER THE ENGINE OPERATING TEMPERATURE (deg F)'
READ(*,*) EOT
WRITE(*,*) 'ENTER THE FLIGHT VELOCITY (MPH)'
READ(*,*) AVEL
HRR=2545.0*HPR*0.40
CFR=0.4*HPR
WRITE(*,*) 'ENTER THE ASSUMED DIFFUSER EFFICIENCY'
READ(*,*) DIFFEFF
WRITE(*,*) 'ENTER THE MAXIMUM OPERATING ALTITUDE'
READ(*,*) CALTM

```

C
C
C
C

CALCULATE AIR PROPERTIES FOR EACH ALTITUDE INCREMENT
LIMITED TO 36000 FT

```

DIV=OALTM/10
ALT=0.0
DO 100 I=1,11
THETA=1.0-(ALT*6.875E-6)
DELTA=THETA**5.2561
SIGMA=THETA**4.2561
PO=14.696
DENSO=0.07648/32.2
TEMPO=99.0
ALTP(I,2)=(TEMPO-((ALT/1000.0)*3.57))+460.0
ALTP(I,3)=PO*DELTA*144.0
ALTP(I,4)=DENSO*SIGMA
ALTP(I,1)=ALT
ALT=ALT+DIV
100 CONTINUE

```

C
C
C
C

CALCULATE AIR PROPERTIES AT THE CORE ENTRANCE
ASSUME VELOCITY AT THE CORE TO BE CLOSE TO ZERO

```

VO=AVEL*1.46
DO 200 I=1,11
FC=1.0+1.035*((VO/100)**2/ALTP(I,2))+0.422*((VO/100)**2/ALTP(I,2))
1**2
QC(I)=(ALTP(I,4)/2.0)*(VO**2)*FC
DIFFP(I)=(1.0-DIFFEFF)*QC(I)
CFP(I,1)=ALTP(I,3)+QC(I)-DIFFP(I)
DELTAT=0.833*((VO/100.0)**2)
CFP(I,2)=ALTP(I,2)+DELTAT
CFP(I,3)=0.002378*(CFP(I,1)/2115.0)*(518.4/CFP(I,2))
200

```

```

      CFF(I,4)=1/(CFF(I,3)*32.2)
200 CONTINUE
C
C CALCULATE THE AVERAGE TEMPERATURE RISE THROUGH THE ENGINE
C
      DELCOL=HRR/(CFR*8.33*60.0)
C
C SPECIFY THE RADIATOR SURFACE VARIABLES
C
      WRITE(*,*) 'CHOOSE EITHER THE PROGRAMED SURFACE VARIABLES OR USER
1 INPUT'
      WRITE(*,*) '1-ENTER USER VARIABLES'
      WRITE(*,*) '2-SURFACE TYPE 9.68-0.87 PLAIN INLINE'
      WRITE(*,*) '3-SURFACE TYPE 9.1-0.737S PLAIN STAGGERED'
      WRITE(*,*) '4-SURFACE TYPE 9.68-0.87R RUFFLED INLINE'
      WRITE(*,*) '5-SURFACE TYPE 9.21-0.737SR RUFFLED STAGGERED'
      WRITE(*,*) '6-SURFACE TYPE 11.32-0.737SR RUFFLED STAGGERED'
300 READ(*,*) NVALUE
      IF (NVALUE .EQ. 1) GOTO 301
      IF (NVALUE .EQ. 2) GOTO 302
      IF (NVALUE .EQ. 3) GOTO 302
      IF (NVALUE .EQ. 4) GOTO 302
      IF (NVALUE .EQ. 5) GOTO 302
      IF (NVALUE .EQ. 6) GOTO 302
      IF (NVALUE .EQ. 7) GOTO 1281
      WRITE(*,*) 'CHOOSE 1 THRU 6 TO CONTINUE, 7 TO EXIT'
      GOTO 300
C
      DATA S(1,1)/0.0725/,S(2,1)/0.0100/,S(3,1)/0.01180/
      DATA S(4,1)/0.00033/,S(5,1)/0.697/,S(6,1)/229.0/
      DATA S(7,1)/0.795/
      DATA S(8,1)/0.00321/,S(9,1)/0.00058/,S(10,1)/0.179/
      DATA S(11,1)/45.945/,S(12,1)/0.01522/,S(13,1)/0.01317/
C
      DATA S(1,2)/0.06142/,S(2,2)/0.00833/,S(3,2)/0.01380/
      DATA S(4,2)/0.00033/,S(5,2)/0.788/,S(6,2)/224.0/
      DATA S(7,2)/0.813/
      DATA S(8,2)/0.00301/,S(9,2)/0.00039/,S(10,2)/0.129/
      DATA S(11,2)/42.1/,S(12,2)/0.01224/,S(13,2)/0.01875/
C
      DATA S(1,3)/0.0725/,S(2,3)/0.010/,S(3,3)/0.01180/
      DATA S(4,3)/0.00033/,S(5,3)/0.697/,S(6,3)/229.0/
      DATA S(7,3)/0.795/
      DATA S(8,3)/0.00321/,S(9,3)/0.00058/,S(10,3)/0.179/
      DATA S(11,3)/45.945/,S(12,3)/0.01522/,S(13,3)/0.01317/
C
      DATA S(1,4)/0.06142/,S(2,4)/0.00833/,S(3,4)/0.01352/
      DATA S(4,4)/0.00033/,S(5,4)/0.788/,S(6,4)/228.0/
      DATA S(7,4)/0.814/
      DATA S(8,4)/0.00301/,S(9,4)/0.00039/,S(10,4)/0.129/
      DATA S(11,4)/42.1/,S(12,4)/0.01224/,S(13,4)/0.01875/
C
      DATA S(1,5)/0.06142/,S(2,5)/0.00833/,S(3,5)/0.01152/
      DATA S(4,5)/0.00033/,S(5,5)/0.780/,S(6,5)/270/
      DATA S(7,5)/0.345/
      DATA S(8,5)/0.00301/,S(9,5)/0.00039/,S(10,5)/0.129/
      DATA S(11,5)/42.1/,S(12,5)/0.01224/,S(13,5)/0.01875/
C
302 NS=NVALUE-1
      TUBLEN=S(1,NS)

```

```

TUBWID=S(2,NS)
HYDA=S(3,NS)
FINTHK=S(4,NS)
SIGMAA=S(5,NS)
ALPHA=S(6,NS)
FATA=S(7,NS)
FATUBE=S(8,NS)
FREARTUBE=S(9,NS)
SIGMAW=S(10,NS)
ALPHAW=S(11,NS)
HYDW=S(12,NS)
FINLEN=S(13,NS)
GOTO 307
301 WRITE(*,*) 'ENTER THE FOLLOWING SURFACE VARIABLES'
WRITE(*,*) 'TUBE LENGTH (FT)'
READ(*,*) TULEN
WRITE(*,*) 'TUBE WIDTH (FT)'
READ(*,*) TUBWID
WRITE(*,*) 'AIRSIDE HYDRAULIC DIAMETER (FT)'
READ(*,*) HYDA
WRITE(*,*) 'FIN THICKNESS (FT)'
READ(*,*) FINTHK
WRITE(*,*) 'FREE FLOW/FRONTAL AREA'
READ(*,*) SIGMAA
WRITE(*,*) 'HEAT TRANSFER AREA/TOTAL VOLUME (SQ FT/CU FT)'
READ(*,*) ALPHA
WRITE(*,*) 'FIN AREA/TOTAL AREA'
READ(*,*) FATA
WRITE(*,*) 'ENTER THE FOLLOWING VARIABLES FOR THE WATER SIDE'
WRITE(*,*) 'FRONTAL AREA OF ONE TUBE (SQ FT)'
READ(*,*) FATUBE
WRITE(*,*) 'FREE FLOW AREA OF ONE TUBE (SQ FT)'
READ(*,*) FREARTUB
WRITE(*,*) 'FREE-FLOW/FRONTAL AREA'
READ(*,*) SIGMAW
WRITE(*,*) 'COOLANT SIDE TRANSFER AREA/TOTAL VOLUME (SQ FT/CU FT)'
READ(*,*) ALPHAW
WRITE(*,*) 'COOLANT SIDE HYDRAULIC DIAMETER (FT)'
READ(*,*) HYDW
WRITE(*,*) 'ENTER THE FIN LENGTH (FT)'
READ(*,*) FINLEN
307 WRITE(*,*) 'ENTER THE THERMAL CONDUCTIVITY OF CORE MATERIAL'
WRITE(*,*) '      IN (BTU/HR-SQ-FT-F/FT)'
READ(*,*) CONCR
799 DO 701 J=1,11
IF(J .GT. 1) GOTO 80
IF(K .EQ. 1)GOTO 601
IF(K .GT. 1)GOTO 80
C
C   CALCULATE THE FIRST ESTIMATE OF THE FRONTAL AREA
C
T2F=CFP(J,2)-460.0
TWALL=EOT-(DELCO/2.0)
CP=0.240
G=32.2
HRRS=HRR/3600.0
ABO=(HRRS/(CP*G*CFP(1,3)*(TWALL-T2F)*VO))*(2.22/SIGMAA)
AFR=1.44*ABO*144.0
WRITE(*,*) HRRS,CFP(1,3),TWALL,T2F,VO,SIGMAA
C

```

```

C      THIS SECTION CALCULATES FLUID PROPERTIES FOR AVERAGE COOLANT TEMPERATURE
C      ASSUMES A 70-30 MIX
C
C      CONCO=0.25382+(0.46661E-3*TWALL)-(0.17179E-5*TWALL**2)+(0.16964E-
18*TWALL**3)
C      CMU=13.658-(0.16843*TWALL)+(0.75967E-3*TWALL**2)-(0.11410E-5*TWALL
1**3)
C      PRNOC=45.614-(0.57836*TWALL)+(0.26547E-2*TWALL**2)-(0.40249E-5*TW
1ALL**3)
C      SPVOLC=0.15253E-1+(0.16708E-5*TWALL)+(0.13250E-7*TWALL**2)-(0.3083
13E-11*TWALL**3)
C      FIRST ESTIMATE OF THE DEPTH BASED ON COOLANT SIDE RN
C
C      GW=(8000.0*CMU)/HYDW
C      AFRWR=(CFR+8.33*60.0)/(GW*SIGMAW)
C      WRITE(*,*) 'THE FIRST ESTIMATE OF FRONTAL AREA IS',AFR,'SQ IN'
C      WRITE(*,*) 'THE FIRST ESTIMATE OF THE COOLANT SIDE AREA IS',
1AFRWR,'SQ IN'
601 WRITE(*,*) 'ENTER THE CORE VARIABLES WIDTH,HEIGHT,AND DEPTH (IN)'
C      READ(*,*) W,H,D
C      W=W/12.0
C      H=H/12.0
C      D=D/12.0
C      AFR=W*H
C      AFRW=W*D
C      VTOT=W*D*H
C      WT=VTOT*.5*.289
80 CONTINUE
C
C      THIS ROUTINE ASSUMES A REYNOLDS NO CALCULATES A DELTA T THEN
C      ESTABLISHES A HEAT BALANCE RECALCULATES A REYNOLDS NO. UNTILL
C      CONVERGENCE IS REACHED
C
C      LDR=D/HYDA
C      IF(J .GT. 1)GOTO 750
C      RN=50000.0
C      IF(J .EQ. 1)GOTO 652
750 RN=RNA(J-1)
652 PEREF=1.0-EXP(-0.1*(RN**(-0.2))*LDR)
C      DELTI=TWALL-T2F
C      WA(J)=HRR/(0.244*DELT I*PEREF)
C
C      HEAT BALANCE TO DETERMINE THE AIR OUTLET TEMPERATURE
C
C      DELTAR=HRR/(WA(J)*.244)
C      TOA=T2F+DELTAR
C      AIRAVG=(TOA+T2F)/2.0
C
C      CALCULATE AIR PROPERTIES
C
C      AMU=0.39898E-1+(0.72099E-4*AIRAVG)-(0.14879E-6*AIRAVG**2)+(0.60370
1E-9*AIRAVG**3)-(0.80323E-12*AIRAVG**4)
C      FRNOA=0.72
C      SPVOLA=11.619+(0.22316E-1*AIRAVG)+(0.30130E-4*AIRAVG**2)-(0.96572E
1-7*AIRAVG**3)+(0.99763E-10*AIRAVG**4)
C
C      CALCULATE REYNOLDS NUMBERS
C
C      AIRSIDE

```

```

GA=WA(J)/(AFR*SIGMAA)
RNA(J)=(HYDA+GA)/AMU
RNDIFF=ABS(RN-RNA(J))
PERCENT=RNDIFF/RN
IF (PERCENT .GT. 0.05) GOTO 650
GOTO 654
650 CONTINUE
IF (RNA(J) .LT. RN) GOTO 651
GOTO 653
651 RN=RN-(RNDIFF/2.0)
GOTO 652
653 RN=RN+(RNDIFF/2.0)
GOTO 652
654 CONTINUE
IF (RNA(J) .LT. 10000) GO TO 150
RYN=10000.0
GA1=(RYN*AMU)/HYDA
AFR1=(WA(J)/(GA1*SIGMAA))*144.0
WRITE(*,*) 'THE CALCULATED RYNOLDS NO.',RNA(J), ' EXCEEDS 10000'
WRITE(*,*) 'THIS INDICATES THAT THE FRONTAL AREA IS TOO SMALL'
WRITE(*,*) 'THE REQUIRED FRONTAL AREA IS',AFR1,'SQ-IN'
WRITE(*,*) 'MORE AREA SHOULD BE PROVIDED '
GOTO 601
C
C WATER SIDE
C
150 GW=(CFR*8.33*60.0)/(AFRW*SIGMAW)
RNC=(HYDW*GW)/CMU
C
C CALCULATION OF HST AND F
C
IF (NVALUE .EQ. 1) GOTO 1281
IF (NVALUE .EQ. 2) GOTO 802
IF (NVALUE .EQ. 3) GOTO 803
IF (NVALUE .EQ. 4) GOTO 804
IF (NVALUE .EQ. 5) GOTO 805
IF (NVALUE .EQ. 6) GOTO 806
802 CONTINUE
STNNPR=0.16077E-1-(0.15838E-4*RNA(J))+(0.75194E-8*RNA(J)**2)-(0.161
133E-11*RNA(J)**3)+(0.15743E-15*RNA(J)**4)-(0.56737E-20*RNA(J)**5)
FRIC=0.63942E-1-(0.65770E-4*RNA(J))+(0.31317E-7*RNA(J)**2)-(0.6769
1E-11*RNA(J)**3)+(0.66589E-15*RNA(J)**4)-(0.24163E-19*RNA(J)**5)
GOTO 807
803 CONTINUE
STNNPR=0.2158E-1-(0.16689E-4*RNA(J))+(0.70442E-8*RNA(J)**2)-(0.143
122E-11*RNA(J)**3)+(0.13584E-15*RNA(J)**4)-(0.48146-20*RNA(J)**5)
FRIC=0.75431E-1-(0.59465E-4*RNA(J))+(0.25235E-7*RNA(J)**2)-(0.5144
13E-11*RNA(J)**3)+(0.48878E-15*RNA(J)**4)-(0.17344E-19*RNA(J)**5)
GOTO 807
804 CONTINUE
STNNPR=0.13786E-1-(0.91411E-5*RNA(J))+(0.39626E-8*RNA(J)**2)-(0.82
1727E-12*RNA(J)**3)+(0.800009E-16*RNA(J)**4)-(0.2875E-20*RNA(J)**5)
FRIC=0.70523E-1-(0.57435E-4*RNA(J))+(0.26147E-7*RNA(J)**2)-(0.5520
12E-11*RNA(J)**3)+(0.5350E-15*RNA(J)**4)-(0.19225E-19*RNA(J)**5)
GOTO 807
805 CONTINUE
STNNPR=0.20077E-1-(0.14243E-4*RNA(J))+(0.60904E-8*RNA(J)**2)-(0.12
1444E-11*RNA(J)**3)+(0.11815E-15*RNA(J)**4)-(0.41858E-20*RNA(J)**5)
FRIC=0.84360E-1-(0.63841E-4*RNA(J))+(0.27751E-7*RNA(J)**2)-(0.5698

```

```

16E 11*RNA(J)**3)+(0.54196E-15*RNA(J)**4)-(0.19215E-19*RNA(J)**5)
      GOTO 807
300 CONTINUE
      STNNPR=0.21399E-1-(0.18035E-4*RNA(J)**2)-(0.17
1568E-11*RNA(J)**3)+(0.17048E-15*RNA(J)**4)-(0.61157E-20*RNA(J)**5)
      FRIC=0.85576E-1-(0.72508E-4*RNA(J)**2)+(0.33174E-7*RNA(J)**3)-(0.7063
15E-11*RNA(J)**3)+(0.68934E-15*RNA(J)**4)-(0.24899E-19*RNA(J)**5)
807 CONTINUE
C
C   CALCULATE STANTON NUMBER
C
      STN=STNNPR/(PRNOA**0.66666)
      RNU=0.023*(RNC**0.8)*(PRNOC**0.3333)
C
C   CALCULATE THE UNIT FILM CONDUCTANCES
C
C   AIRSIDE
C
      HA=STN*GA*0.244
C
C   COOLANT SIDE
C
      HC=(RNU*CONCO)/HYDW
C
C   CALCULATE FIN EFFECTIVENESS AIRSIDE ONLY
C
      FNEFF=SQRT((2.0*HA)/(CONCR*FINLTHK))
      EFFLEN=FNEFF*FINLEN
      ETAF=1.0001E+1+(0.13632E-2*EFFLEN)-(0.40456*EFFLEN**2)+(0.19442*EF
1FLEN**3)-(0.28666E-1*EFFLEN**4)
C
C   CALCULATE OVERALL SURFACE EFFECTIVENESS (AIRSIDE)
C
      ETAO=1.0-FATA*(1.0-ETAF)
C
C   CALCULATE THE OVERALL COEFFICIENT OF HEAT TRANSFER U (BASED ON
AIRSIDE AREA)
C
      UINV=(1.0/(ETAO*HA))+1.0/((SIGMAW/SIGMAA)*HC)
      UA=1.0/UINV
C
C   CALCULATE NTU AND EXCHANGER EFFECTIVENESS
C
C   AIRSIDE CAPACITY RATE
C
      CA=WA(J)*CP
C
C   COOLANT SIDE CAPACITY RATE
C
      CC=CFR*8.33*60.0
C
C   NUMBER OF HEAT TRANSFER UNITS
C
      UNT=(ALPHA*VTOT*UA)/CA
      R=CA/CC
      SUM3=(1-EXP(-UNT))*(1-EXP(-R*UNT))
      DO 904 N=1,4
      SUM1=1.0
      SUM2=1.0
      DO 903 M=1,N

```

```

FLOWACC=(1.0+(SIGMAA**2))*RV1-1.0)
COREFRIC=FRIC*((ALPHA*VTOT)/(SIGMAA*AFR))*RV2
ADD=FLOWACC+COREFRIC
PRSDRP(J)=((ELMU**2)*V1*ADD)/(64.4*144.0)
WRITE(*,*) 'THE CORE PRESSURE DROP =',PRSDRP(J),'PSI'
PRSDRP(J)=PRSDRP(J)*144.0
PINLET=CFP(J,1)
FREESTRM=ALTP(J,3)
IF(J.GT.1.AND.MVALUE.EQ.1)GOTO 796
WRITE(*,*) 'IS THE HEAT EXCHANGER INLET LOCATED IN THE PROPELLER
1SLIPSTREAM,1=YES,2=NO'
READ(*,*) MVALUE1
IF(MVALUE1.EQ.2)GOTO 795
796 PINLET=ALTP(J,3)+(QO(J)*1.3)-DIFFP(J)
IF(J.GT.1.AND.MVALUE2.EQ.1)GOTO 797
C DEFLECT FLAP 15 DEG
795 WRITE(*,*) 'IS THE OUTLET FLAP DEFLECTED INTO THE SLIPSTREAM ENTER
11 FOR YES 2 FOR NO'
READ(*,*) MVALUE2
IF(MVALUE2.EQ.2)GOTO 780
797 FREESTRM=ALTP(J,3)-(0.5*QO(J))
C
C CALCULATE THE STATIC PRESSURE BEHIND THE EXCHANGE
C
780 P3=PINLET-PRSDRP(J)
T3=CFP(J,2)+DELTAR
C
C CALCULATE THE VALUES AT THE EXIT
C ASSUME 2% OF QO FOR DELTAP4
C
DELP4=0.02*QO(J)
P4=P3-DELP4
WRITE(*,*) PINLET,FREESTRM,P3,T3,P4,ALTP(J,3),QO(J)
IF(FREESTRM.LT.P4)GOTO 800
WRITE(*,*) 'THE FLAP DEFLECTION REQUIRED EXCEEDS 15 DEG'
WRITE(*,*) 'THE HEAT EXCHANGER PRESSURE DROP EXCEEDS THE PRESSURE
1AVAILABLE, A LARGER CORE IS NEEDED'
WRITE(*,*) 'INPUT 1 TO CONTINUE, 2 TO EXIT'
READ(*,*) KVALUE
IF(KVALUE.EQ.2)GOTO 1281
K=K+1
GOTO 799
800 T4=T3*((FREESTRM/P4)**0.286)
VEXIT(J)=100.0*SQR((T3-T4)/0.832)
RHO4=0.002378*(P4/2116.0*(518.4/T4))
RHO4=RHO4*32.2
WA(J)=WA(J)/3600.0
VFR(J)=WA(J)/RHO4
EXAREA(J)=VFR(J)/VEXIT(J)
C
C INTERNAL POWER CONSUMED (HP)
C
PCI(J)=(WA(J)/32.2)*(VO*(VO-VEXIT(J)))/550.
701 CONTINUE
OPEN(4,FILE='LPT1')
WRITE(4,1000) TITLE
1000 FORMAT('1',5X,20A4,///)
WRITE(4,1010) HPR
1010 FORMAT('0',5X,'ENGINE HORSEPOWER.....',F6.1,' HP')
WRITE(4,1020) EOT

```

```

      CALL FACT(M,X)
      SUM1=SUM1+(UNT**M)/X
      SUM2=SUM2+(R*UNT)**M)*X
903  CONTINUE
      SUM3=SUM3+((1-EXP(-UNT)*SUM1)*(1-EXP(-R*UNT)*SUM2))
904  CONTINUE
      EXEFF=(1/(R*UNT))*SUM3
C
C      CALCULATE THE REQUIRED EFFECTIVENESS
C      AIR OUTLET TEMPERATURE
      TOA=T2F+DELTAR
      EXEFFR=DELTAR/(EOT-T2F)
      IF (EXEFF.GT. EXEFFR) GOTO 502
      WRITE(*,*) 'THE HEAT EXCHANGER EFFECTIVENESS IS LESS THAN REQUIRED
1'
      WRITE(*,*) 'HEAT EXCHANGER EFFECTIVENESS=',EXEFF
      WRITE(*,*) 'THE REQUIRED EFFECTIVENESS=',EXEFFR
      WRITE(*,*) 'THE THERMAL DESIGN IS NOT SATISFACTORY AND REQUIRES A
1DEEPER CORE'
      IF(J.GT. 1) GOTO 710
      WRITE(*,*) 'DO YOU WANT TO CONTINUE OR EXIT THE PROGRAM'
      WRITE(*,*) 'ENTER 1 TO CONTINUE OR 2 TO EXIT'
      READ(*,*) IVALUE
      IF (IVALUE.EQ. 1) GOTO 601
      IF (IVALUE.EQ. 2) GOTO 1281
502  WRITE(*,*) 'THE HEAT EXCHANGER EFFECTIVENESS IS GREATER THAN
1REQUIRED'
      WRITE(*,*) 'HEAT EXCHANGER EFFECTIVENESS=',EXEFF
      WRITE(*,*) 'THE REQUIRED EFFECTIVENESS=',EXEFFR
      IF(J.GT. 1) GOTO 710
521  WRITE(*,*) 'THE THERMAL DESIGN IS SATISFACTORY, HOWEVER IF THE'
      WRITE(*,*) 'DIFFERENCE IS LARGE A SMALLER CORE COULD BE USED'
      WRITE(*,*) 'DO YOU WANT TO CONTINUE, INPUT NEW CORE DIMENSIONS'
      WRITE(*,*) 'OR EXIT'
      WRITE(*,*) 'ENTER 1 TO CONTINUE, 2 TO EXIT, 3 TO INPUT NEW CORE '
      READ(*,*) IVALUE
      IF (IVALUE.EQ. 1) GOTO 710
      IF (IVALUE.EQ. 2) GOTO 1281
      IF (IVALUE.EQ. 3) GOTO 601
710  CONTINUE
C
C      THIS SECTION CALCULATES THE PRESSURE DROP FOR THE AIR SIDE
C
C      SPECIFIC VOLUME AT THE CORE FACE
      V1=CFP(J,4)
C
C      CALCULATE THE SPECIFIC VOLUME AT THE OUTLET
      V2=((460.0+TOA)/39.55)*(14.7/CFP(J,1)/144.0)
C
C      CALCULATE THE SPECIFIC VOLUME AT THE MEAN TEMPERATURE
      VM=(V1+V2)/2.0
      RV1=V2/V1
      RV2=VM/V1
      ELMU=GA/3600.0
C
C      CALCULATE LOSSES

```

```

1020 FORMAT('0',5X,'ENGINE OPERATING TEMPERATURE...',F6.2,' DEG F' )
    WRITE(4,1030) OALTM
1030 FORMAT('0',5X,'MAXIMUM OPERATING ALTITUDE...',F8.2,' FT')
    WRITE(4,1040) AVEL
1040 FORMAT('0',5X,'FLIGHT VELOCITY...',F6.2,' MPH')
    WRITE(4,1050) DIFFEFF
1050 FORMAT('0',5X,'DIFFUSER EFFICIENCY...',F4.2, '//')
    WRITE(4,1060)
1060 FORMAT('0',5X,'*****HEAT EXCHANGER DETAILS*****')
    IF(NVALUE .EQ. 1) GO TO 1070
    IF(NVALUE .EQ. 2) GO TO 1080
    IF(NVALUE .EQ. 3) GO TO 1090
    IF(NVALUE .EQ. 4) GO TO 1100
    IF(NVALUE .EQ. 5) GO TO 1110
    IF(NVALUE .EQ. 6) GO TO 1120
1070 WRITE(4,1130)
1130 FORMAT('0',5X,'SURFACE TYPE IS USER DEFINED')
    GO TO 1190
1080 WRITE(4,1140)
1140 FORMAT('0',5X,'SURFACE TYPE IS 9.68-0.87 PLAIN INLINE')
    GO TO 1190
1090 WRITE(4,1150)
1150 FORMAT('0',5X,'SURFACE TYPE IS 9.1-0.73-S PLAIN STAGGERED')
    GO TO 1190
1100 WRITE(4,1160)
1160 FORMAT('0',5X,'SURFACE TYPE IS 9.68-0.87R RUFFLED-INLINE')
    GO TO 1190
1110 WRITE(4,1170)
1170 FORMAT('0',5X,'SURFACE TYPE IS 9.21-0.737-SR RUFFLED STAGGERED')
    GO TO 1190
1120 WRITE(4,1180)
1180 FORMAT('0',5X,'SURFACE TYPE IS 11.32-0.737-SR RUFFLED STAGGERED')
1190 CONTINUE
    WRITE(4,1200) AFR
1200 FORMAT('0',5X,'AIRSIDE FRONTAL AREA...',F6.2,'SQ-FT')
    WRITE(4,1210) AFRW
1210 FORMAT('0',5X,'COOLANT SIDE FRONTAL AREA...',F6.2,'SQ-FT')
    WRITE(4,1220) H,W,D
1220 FORMAT('0',5X,'CORE DIMENSIONS',/,6X,'HEIGHT=',F5.2,/,6X,'WIDTH=',F
15.2,/,6X,'DEPTH=',F5.2)
    WRITE(4,1230) VTOI
1230 FORMAT('0',5X,'HEAT EXCHANGER VOLUME=',F6.3,'CU-FT')
    WRITE(4,1240) WT
1240 FORMAT('0',5X,'HEAT EXCHANGER WEIGHT=',F6.4,'LBS')
    WRITE(4,1250)
1250 FORMAT('0',3X,'ALTITUDE',2X,'MASS FLOW',2X,'VOLUMETRIC FLOW',2X,'P
1RESSURE DROP',2X,'EXIT AREA',2X,'INTERNAL HP')
    WRITE(4,1260)
1260 FORMAT(15X,'(LB/SEC)',5X,'(CU FT/SEC)',5X,'(PSF)'10X,'(SQR FT)',
15X,'(HP)')
    ALT=0.0
    DO 1280 I=1,11
    WRITE(4,1270) ALT,WA(I),VFR(I),PRSDRP(I),EXAREA(I),PCI(I)
1270 FORMAT('0',3X,F7.1,5X,F5.2,5X,F7.2,10X,F5.2,8X,F5.2,6X,F5.2)
    ALT=ALT+DIV
1280 CONTINUE
    VAR=0.0
    DO 2000 I=1,5
    VAR=VAR+0.2
    VIN=VO*VAR

```

```

      DO 2020 N=1,11
      AREAIN(N)=WA(N)/((ALTP(N,4)*32.2)*VIN)
2020 CONTINUE
      WRITE(4,2030)
2030 FORMAT('0','ALTITUDE',2X,'INLET AREA',2X,'VELOCITY RATIO')
      ALT=0.0
      DO 2040 N=1,11
      WRITE(4,2050) ALT,AREAIN(N),VAR
2050 FORMAT('0',3X,F7.1,5X,F6.3,5X,F3.1)
      ALT=ALT+DIV
2040 CONTINUE
2000 CONTINUE
1281 STOP
      END
      SUBROUTINE FACT(M,FA)
      FA=1.0
      DO 910 I=1,M
      FA=FA*I
910 CONTINUE
      RETURN
      END

```

APPENDIX IV

BOUNDARY LAYER CONTROL FOR VARIOUS INLET TYPES

APPENDIX IV

BOUNDARY LAYER CONTROL FOR VARIOUS INLET TYPES

The first attempts (Ref. 70) provided for more containment of the boundary layer rather than controlling the boundary layer. This containment was provided by providing horizontal and vertical inlet and outlet guide vanes in the duct (Figure 39) which essentially contained the low energy air next to the surface and did not allow it to mix with the higher energy air. A major drawback to this method is the testing required to ensure proper operation and the complication of the outlet flap design.

The next step (Ref. 70) provided a bypass channel for the low energy air (Figure 40). In this design the boundary layer air was channeled around the heat exchanger. This method provides for good overall pressure recoveries. The bypass sizing, however, is critical and sufficient flow area must be provided to ensure good duct operation.

An extension of this idea is boundary layer diversion. This method requires that the inlet be located above the surface and a splitter plate and channel be provided for the boundary layer air. This is probably the most common and best operating of the systems.

In all cases sufficient exit area for the boundary layer air should be provided. If this area is not provided, the blockage will propagate upstream and result in a local thickening of the boundary layer ahead of the inlet (Ref. 11).

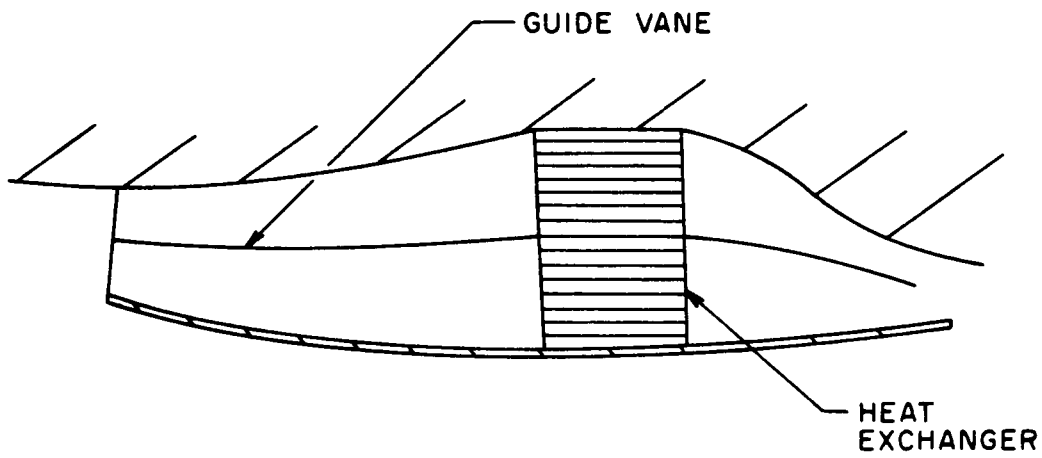


Figure 39. Guide vane installation.

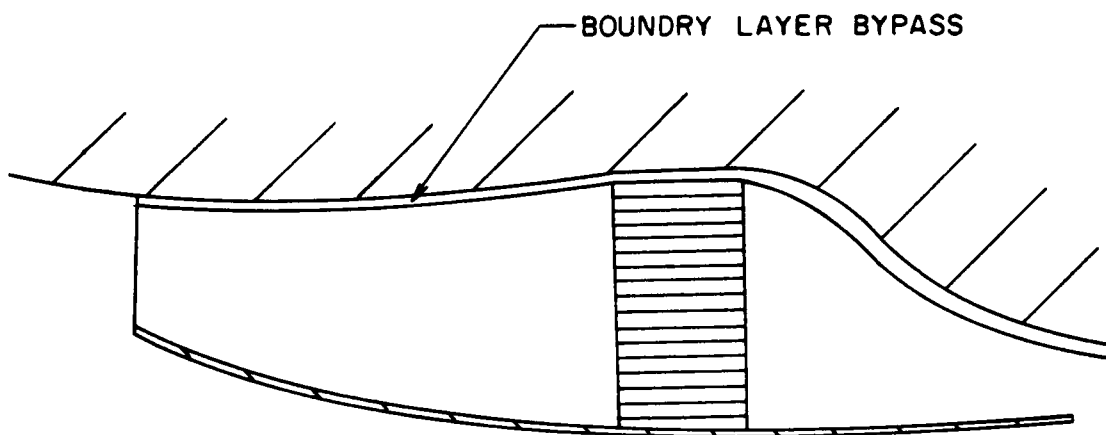


Figure 40. Boundary layer bypass.

Boundary Layer Diverter Design

Good pressure recovery of scoops requires that the flow into the inlet be smooth and as uniform in velocity profile as possible. Locating the scoop above the surface and providing for the diversion of the boundary layer prevents separation of the flow ahead of the inlet and thereby improves the pressure recovery. The basic configuration is shown in Figure 41.

To size the diverter for proper operation, it is first necessary to determine the boundary layer thickness at the location of the inlet. Although exact equations exist for the calculation of the boundary layer thickness, for design purposes it is conservative to assume the boundary layer is turbulent from the foremost part of the body.

The boundary layer thickness is then given by:

$$\delta_t = 0.37 x / Re_x^{0.2}$$

and

$$Re_x = \frac{\rho_\infty V_\infty x}{\mu}$$

The height of the diverter (h) is somewhat arbitrary. Ref. 70 indicates that large gains in duct performance can be achieved with the duct entrance raised to at least an amount equal to the boundary layer thickness. Other references, Ref. 13 and Ref. 49, indicate that due to uncertainties, it is prudent to locate the duct entrance at twice the boundary layer thickness. In any case it appears that the most conservative value to use is twice the boundary layer thickness. Therefore:

$$h = 2\delta_t$$

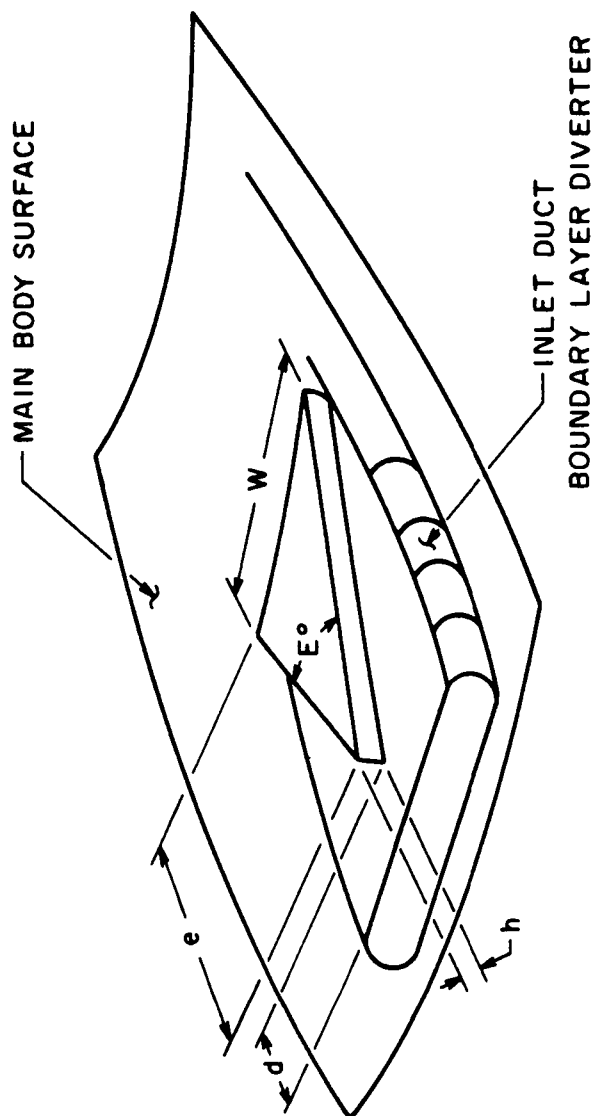


Figure 41. Boundary layer diverter.

Although most of the design rules indicated in Ref. 13 and Ref. 49 are for high speed (high Mach number inlets), the basics still apply to low speed (subsonic) inlets. These are summarized as follows:

1. The diverter apex should be located at least one diverter height back from the inlet plane, i.e., $d = h$.
2. The reduction of pressure and friction drag can be minimized by minimizing the reflection angles; this minimizes the lateral velocity. The included angle E should be 20° , or less.
3. Vortex formation in the channel should be prevented by providing generous radii at the corners.
4. The passage height should diverge both longitudinally and laterally to minimize flow resistance and prevent choking.

Flush Inlets

Flush inlets usually have the form of flush holes or slots, inclined holes or slots, and NACA submerged inlets. These inlets are considered in design because they permit inflow without increasing the frontal area. This is not accomplished without disadvantages, however. The main drawback to flush inlets is that since the openings are more or less perpendicular to the flow, the inflow then must be turned through some angle. Inlets that provide a minimum of turning will provide better performance.

Width to depth ratios also have an effect on performance of flush inlets. At the low Mach number operation of most General Aviation aircraft, wide shallow inlets can be expected to perform better than deep narrow inlets. In most cases, the NACA inlet will

provide better pressure recovery than other types of flush inlets. However, for all flush inlets the pressure recovery can be expected to be less than a well designed forward facing inlet.

Circular Holes in a Flat Surface

Flow for this type of inlet will only occur when the internal (duct) static pressure is lower than the free stream static pressure. This inlet has a very poor pressure recovery and high internal losses, and therefore should not be used for heat exchanger inlets. It may, however, be used for a boundary layer bleed ahead of a forward facing inlet.

Slots

The basic parameter for flush slots is the inclination of the inlet axis with the surface (Figure 42). The smaller this angle, the better the performance. Other parameters should also be considered in the design. If the ramp radius is too small, the flow will separate causing high pressure loss and the duct flow will be highly non-uniform.

The lip radius also is important. Smaller lip radii tend to improve the duct performance to a point; however, too small a radii may again cause separation and degradation in performance. Too large a radii tends to provide poor guidance of the flow into the inlet, also resulting in degradation of the duct performance.

Another fundamental problem in these inlets is the ingestion of the boundary layer. If the boundary layer is relatively thick ahead of the inlet, problems may be encountered in delivering smooth

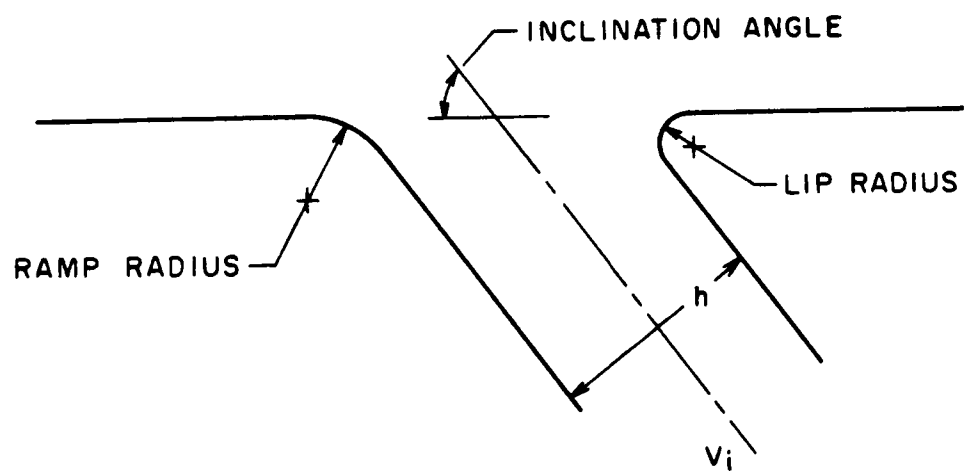


Figure 42. Flush slot nomenclature.

flow in the duct, and the size of the duct may need to be increased to achieve the desired mass flow.

Submerged Inlet

An extension of the slot inlet is the submerged inlet. A rectangular submerged inlet (Figure 43) is developed from the slot inlet by decreasing the angle of inclination, decreasing the ramp radius, and increasing the ramp length. This type of inlet can be essentially considered a two-dimensional inlet. The incoming flow then shows no divergence and flows essentially in parallel streamlines into the inlet. This inlet, however, suffers the same deficiencies as the slot inlet. These deficiencies result from the ingestion of the boundary layer. Boundary layer ingestion should be considered in sizing since for specific mass flows the inlet may have to be enlarged. The pressure recovery is somewhat better than the slot inlet due to the angle of inclination being zero.

NACA Submerged Inlet

The NACA submerged inlet is a development of the parallel wall submerged inlet. It was recognized that if the walls of the inlet are convergent or divergent (Figure 44), then the inlet can no longer be considered two-dimensional, and three-dimensional effects. The first is that since the walls are no longer parallel, the flow in the ramp surface is then either convergent or divergent (depending on the side wall angle). The second effect is that the external flow must flow over the top of the walls into the inlet. This leads to the formation of vortex sheets at the edge of the wall. Flow into

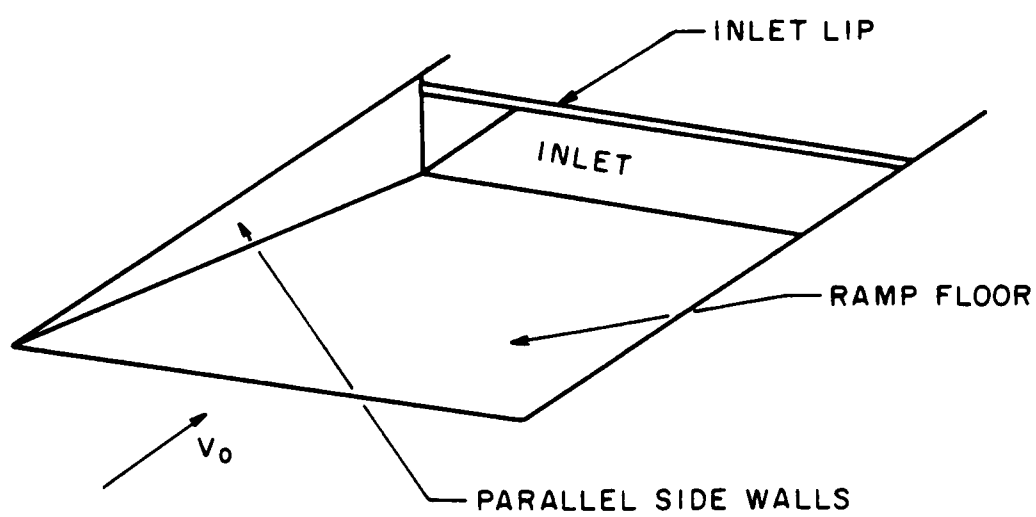


Figure 43. Submerged slot.

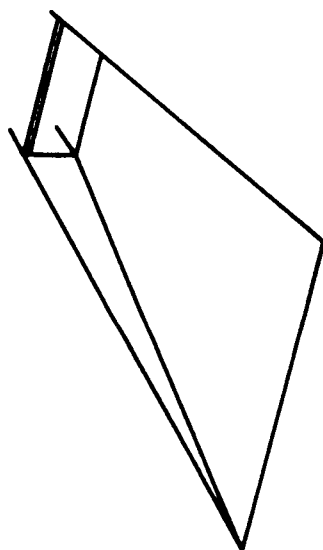
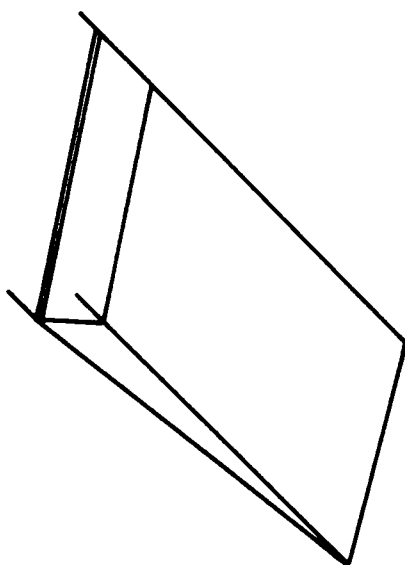


Figure 44. Convergent and divergent slot.

the convergent inlet will result in the formation and the rolling up of these vortex sheets exterior to the inlet (since the flow is up over the edge). These vortices will then essentially not affect the pressure recovery, but will significantly affect the external drag. Flow into the divergent inlet will also result in the formation and rolling up of the vortex sheets. However, these vortices will move into the ramp and flow toward the inlet.

The NACA inlet (Figure 45) makes use of the vortices to induce a sideways component of velocity. This keeps the boundary layer from entering the inlet and replaces the boundary layer flow with higher energy air from the external flow, thus providing for improved flow and better pressure recovery. The use of the NACA inlet for heat exchanger installation (at least primary cooling) appears to be somewhat limited.

These inlets are very sensitive to yaw angles. Yaw results in flow angularity that affects the vortex flow, resulting in vortices that no longer flow into the inlet but up over the surface similar to that discussed in the convergent case. This results in an increase in external drag and reduction in pressure recovery.

The flow is somewhat sensitive to mass flow ratio. The inlet can only be optimized for one mass flow ratio. Reducing the operating mass flow ratio from the design ratio results in a thickening and possible separation of the boundary layer on the ramp floor. This results in the possibility that the vortices may move entirely outside the duct and that the external flow may separate near the ramp edges. Operation at higher than designed mass flows result in entrainment of the surface boundary layer into the vortex

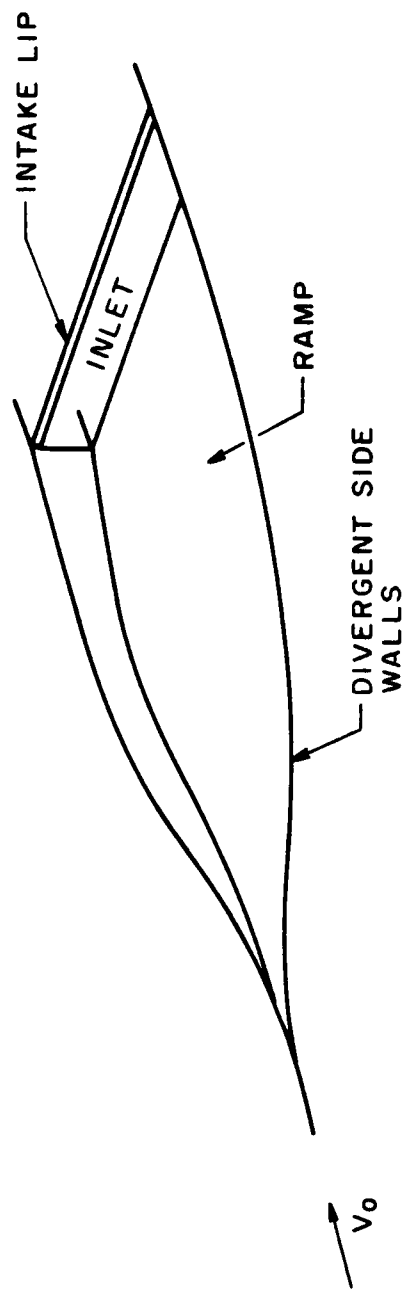


Figure 45. NACA submerged slot.

core. The net result is that if the inlet is operated at conditions different than design, there will be a loss in pressure recovery and an increase in external drag.

There is one more item that should be discussed pertaining to the use of these inlets in a heat exchanger installation. Since the proper operation of the inlet results in the vorticies being swallowed by the intake, the velocity profile in the intake is not very uniform. If efforts are not made downstream of the intake to make the flow more uniform, portions of the heat exchanger core may not be operating at the best efficiency. This lack of flow uniformity would result in a larger core being needed for the same amount of heat transfer.

APPENDIX V

BOUNDARY LAYER DIVERTER DRAG

APPENDIX V

BOUNDARY LAYER DIVERTER DRAG

Reference 49 gives the following equations for the calculation of the diverter drag: (Referenced to cowl frontal area, A_c)

$$C_{DBLD} = (C_{DBLD/2\ BLD})(2.60_{BLD}) A_{BLD}/A_c$$

where:

$C_{DBLD/2\ BLD}$ is from Figure 46.

L is shown in Figure 41.

θ_{BLD} = diverter compression ramp angle in degrees

A_{BLD} = projected surface area of the boundary layer diverter in the flow direction

An alternate method is to provide a boundary layer bypass duct which bypasses the boundary layer flow around the heat exchanger. The drag produced is composed of (1) the change in momentum of the bypass air between the bypass exit and the free stream, and (2) the pressure drag on any bypass exit door or flap.

References 11 and 31 provide test data that shows that there may be additional drag produced with the bypass arrangement, although the pressure recovery is enhanced.

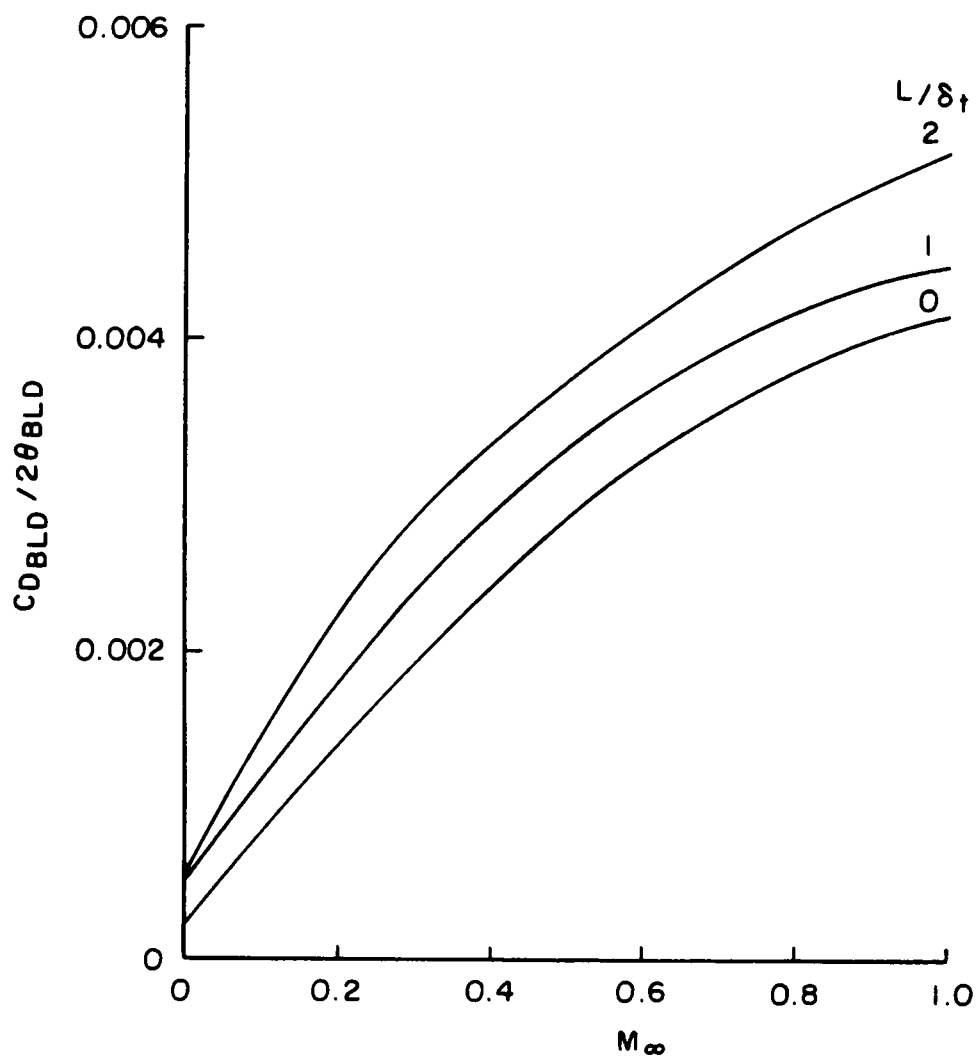


Figure 46. Boundary layer diverter drag variation with freestream Mach No.

APPENDIX VI

WING INTAKES

APPENDIX VI

Wing Intakes

The inlet is affected by the upwash due to the angle of attack in the wing; therefore the entry position for maximum efficiency is dependent upon angle of attack. The ideal inlet is one of a variable geometry, and would require the use of some type of inlet flap that efficiently changes the intake location. Care would have to be taken in this design to provide for maximum pressure recovery and not reduce any total pressure at the inlet. Reference 50 indicates that an efficient design for a fixed inlet has an upper lip of large radius (Ref. 45 recommends a value of 0.00504) conforming approximately to the original wing contour, the lower lip staggered such that the inlet plane is downward at about a 70° angle to the chord line, and the diffuser is approximately 10° to the wing chord. The staggered inlet lips (called scarf inlets in present terminology) are used to effectively control the local velocity over the intake lips. With an increase in angle of attack, the stagnation point moves down along the nose. This results in greater curvature of the flow and an increase in the local velocity, (essentially a higher angle of attack on the lips). At high wing lift coefficients, the flow on the upper wall is similar to flow on the bottom surface of an airfoil at high angles of attack and as such enters smoothly into the duct. The flow on the bottom wall, however, is similar to flow on the top surface of an airfoil, and at high local angles of attack, may separate causing high diffuser losses and non-uniform flow. Cutting back the lower lip has the effect of inclining the inlet, aligning it

more with the local flow, resulting in a lower local angle of attack for the lower duct wall. This staggering of the inlet reduces the tendency for separation, resulting in smoother flow and better pressure recovery. However, if the lower lip is cut too far back, then the resulting high negative local angle of attack at low wing lift coefficients may cause separation on the upper duct wall.

Locating the diffuser axis as a downward angle to the wing chord also has the effect of reducing the inlet flow which must turn to enter the duct. This reduces the tendency for the flow to separate and provides for a smooth duct flow and maximum pressure recovery.

When locating the inlet, three-dimensional effects should be considered. Reference 50 indicates that if the inlet is located in the root, the pressure recovery at the inboard location is low due to ingestion of the fuselage boundary layer, due to the spanwise flow on the leading edge of the wing.

Outlets

Outlets can be located on either the low or upper surface of the wing. In either case, the flow about the wing is affected and the C_{Lmax} of the wing reduced. The upper surface locations reduce C_{Lmax} less than the lower surface locations and in some instances, can actually increase C_{Lmax} .

The optimum position for the outlet position appears to be at about $0.6c$. However, locations between the range of $0.45c$ to $0.7c$ do not significantly change the flow characteristics. Outlets located further forward of $0.45c$ decrease the C_{Lmax} considerably.

Flaps located on the outlets are effective in increasing the ΔP available for cooling. The references (Ref. 45, 50, and 60) show no evidence that investigations determining the effect of cooling flap deflections on C_{Lmax} or C_D of the wing were attempted. It would seem likely that cooling flap deflections would cause more of an effect on upper cooling surface outlets than on the lower surface outlets. The lower surface cooling outlet with cooling flaps deflected may actually increase the circulation and increase C_{Lmax} .

Heat Exchanger Location

The heat exchanger could be located at any wing chord position; however, most wing sections will require that the heat exchanger be located between $0.2c$ and $0.5c$ to be completely within the wing. The forward locations ($0.2c$) are somewhat more difficult to design because the resulting short diffuser section requires large diffuser wall angles. Care must be taken in the diffuser design to prevent separation. The $0.5c$ location allows smaller diffuser wall angles, resulting in efficient diffusers and low duct losses. In general, locations further aft are more aerodynamically acceptable; however, structurally these may be more complex due to the requirement of clear passages. Most general aviation aircraft wing sections have the primary spar located at about the $0.25c$ point, thus passages into and out of heat exchanger may be required to pass through these structural members. This would lead to a more complex structure and possibly more weight.

VITA

Russell D. Groff was born in Barberton, Ohio on July 17, 1947. He graduated from Coventry High School in June 1965. He received a Bachelor of Science degree in Aerospace Engineering in September 1969 from Northrop Institute of Technology of Inglewood, California.

He has worked in the aerospace, nuclear, and energy industries in various engineering capacities. He is currently employed at the Arnold Engineering Development Center (AEDC) as a Senior Engineer/Mechanical Analysis for Calspan Corporation/AEDC Division.

He is married to the former Mary B. Biskner of Stow, Ohio and has three children. They presently reside in Winchester, Tennessee.