

W. G. Thomson, Research Assistant,
W. H. Wentz, Jr., Professor,
and
C. Ostowari, Research Associate
Wichita State University
Wichita, Kansas.

Abstract

The results of an analytical and experimental study of aerodynamic interference effects for a light twin aircraft are presented. Both the influence of a body (either fuselage or nacelle) on a wing and the influence of a wing on a body are studied. The wing studied uses a new natural laminar flow airfoil with variable camber movable trailing edge. A three-dimensional panel method program utilizing surface source and surface doublet singularities was used to design wing-nacelle and wing-fuselage fairings. Experiments were conducted using a 1/6 scale reflection plane model. Forces, pressures, and surface flow visualization results are presented. Results indicate that potential flow analysis is useful to guide the design of intersection fairings, but experimental tuning is still required. While the study specifically addressed a light twin aircraft, the methods are applicable to a wide variety of aircraft.

Introduction

As the needs for energy conservation and aerodynamic efficiency become more critical, sources of drag that have been in past years regarded as tolerable are being viewed as possible sources of improvement. One source of drag which is often appreciable but is not well understood is wing-body aerodynamic interference, both between a wing and a fuselage and between a wing and a nacelle of a multi-engine aircraft.

Experimental methods, such as wind tunnel or flight tests, have become increasingly expensive, especially for parametric studies where several configurations need to be tested, while the advent of higher speed computers and improved software has made computational analysis more attractive. Even though potential flow methods cannot predict drag levels, these methods can be used as a guideline to reduce adverse pressure gradients, which should reduce boundary layer growth and separation, thus reducing drag. An example of a potential flow program that can be used would be a panel method program, such as the Hess program used in this study. This program is documented in reference 1.

In general, when a body is placed in a moving fluid, the fluid will accelerate as it flows around the body (There will, of course, be local regions where the fluid experiences a deceleration, such as near a stagnation region). A lifting wing, in particular, will induce local regions of accelerated flow. If another body is placed in the vicinity of the wing, the accelerated flow velocities will be induced on the body. On the wing, the increased velocity region occurs shortly after the stagnation region, where the boundary layer is thin and the flow can recover from the acceleration, while the increased velocity region on the body occurs far behind the stagnation region, where the boundary layer is well developed and thick, and cannot recover as easily.

As well as the wing affecting the flow about the body, the body affects the flow about the wing. A long, cylindrical shaped body, such as a fuselage, will not induce a change in velocity along the major axis but will induce a change in velocity for flow perpendicular to the axis. These perturbations in velocity will create local changes in angle of attack, resulting in a change in loading on a wing near the intersection with a body. This change in loading can be in the chordwise direction as well as the spanwise direction.

Description of Model

The configuration chosen to be analyzed in this study was a circular fuselage, low wing, unswept, twin aircraft, typical of a large general aviation aircraft, with an aspect ratio of 10.95. This configuration could also apply to a small commuter aircraft (about 19 passenger). The wind tunnel model used for these tests was a modified version of a Beech Aircraft Corporation research model, donated to Wichita State University for this study. A photograph of the model is shown in figure 1, along with pertinent dimensions. For the tests at Wichita State University, only half the model was used as a reflection plane model. The airfoil used on the original model was a 6 series NACA laminar flow airfoil, but most

* This research was supported by NASA Langley Research Center, and Beech Aircraft Company.

of the present testing was done with a new natural laminar flow variable camber airfoil designed by NASA. This airfoil is documented in reference 2. A 25% flap was included with the wing with flap deflections possible ranging from -10° to 40° . The flap extended for the entire span of the wing except for the portion buried beneath the nacelle and fuselage. The planform of the new wing was identical to the planform of the old wing. Portions of the constant area section aft and forward of the wing were removed and plugs were manufactured so short and long versions could be tested. The short configuration retained the constant area circular cross section only in the region of the wing-fuselage intersection, typical of a light twin configuration, while the long version is more representative of the larger business aircraft or a commuter configuration. The computational analysis concentrated on the long version.

Description of Wind Tunnel Facilities

The wind tunnel tests were conducted in the WSU 2.13 x 3.05 meter (7 x 10 foot) low speed wind tunnel. Measurements included 6-component forces, surface pressures on the baseline model, and surface flow patterns using a fluorescent oil mixture and tufts. The model was constructed so that the wing could be tested alone or with the fuselage and/or the nacelle, or the fuselage could be tested alone. Since the nacelle mounted on the wing, no provisions were made for nacelle alone tests.

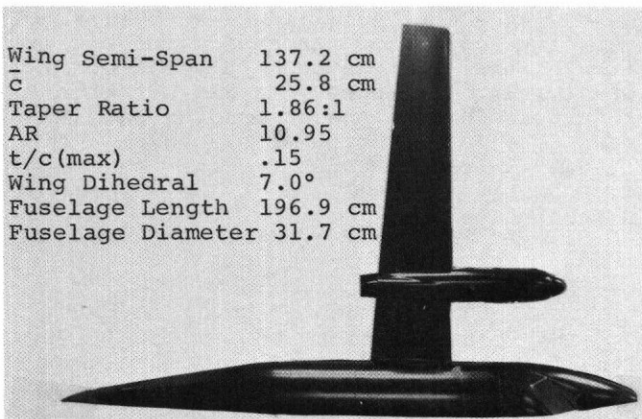


Figure 1- Model Geometry

Wind tunnel tests of the baseline model indicated there were regions of separated flow originating in the intersection region of the wing and body and the wing and nacelle. These separated regions were visible in the fluorescent oil flow studies, as observed in figure 2. Since any separation is an indication of increased drag, it was decided that the goal of the computational design would be to reduce or eliminate these regions, either through altering the shape of the airfoil locally or altering the shape of the body.

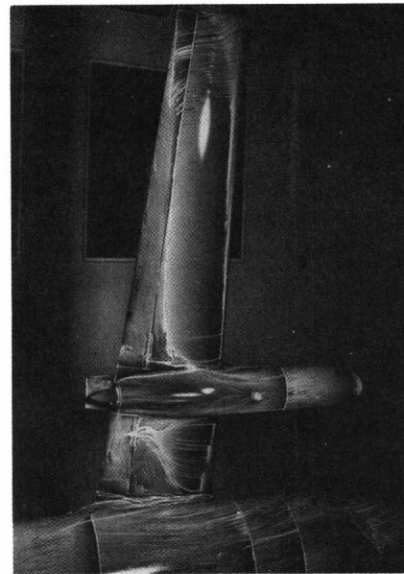


Figure 2- Fluorescent Oil Flow Pattern, $\alpha=10^{\circ}$.

Influence of a Body on the Wing

As mentioned in the introduction, the body induces a change in the vertical velocity in the vicinity of the body. This change in vertical velocity results in a change in the local angle of attack on the wing. An indication of the magnitude of change in vertical velocity (and therefore change in local angle of attack) can be obtained from two dimensional analysis. For the analysis, the Y and Z axis system can be used. This would correlate to the circular cylinder (body) lying on the X axis, with flow in the Z (vertical) direction. Figure 3(a) shows the velocity in the Z direction along a line extending from the side of a circular cylinder resulting from a free stream flow in the Z direction. The local velocity component is twice the free stream component at the edge of the cylinder, and decays to free stream magnitude far from the cylinder. Figure 3(b) is a similar plot, but in this case, the line extends from near the bottom of the cylinder. The velocity is nearly zero at the cylinder (since it is near a stagnation point), and accelerates to a velocity greater than free stream magnitude some distance from the cylinder, then decays to a velocity equal to free stream magnitude far from the cylinder. Note that only the Z component of velocity is shown in these figures. In figure 3(a), there is no Y component of velocity, but in the condition shown in figure 3(b), there would be a Y component of velocity that is not included. Figure 3(b) is approximately representative of a wing-fuselage intersection for a low wing aircraft, while figure 3(a) is more representative of a wing-nacelle intersection or a mid wing configuration.

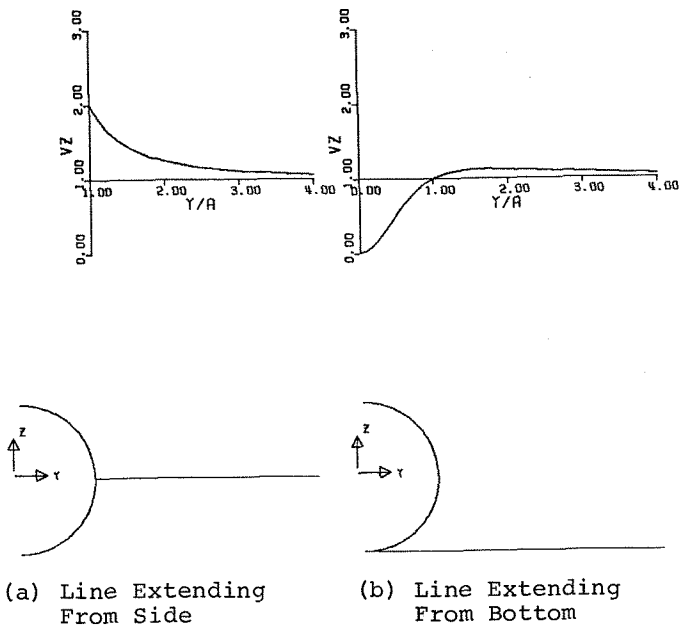


Figure 3- Flow about a Circular Cylinder.

The increased angle of attack near the intersection has the effect of increasing the local loading at the intersection. In addition, the wing induces a velocity that is felt by the body and this effect is mirrored back to the wing. Around a lifting airfoil, there is an upflow near the leading edge of the airfoil and a downflow near the trailing edge. Near the body, the effects of vertical velocities are altered, as described earlier. Therefore, for a midwing intersection, there is a net increase in the upflow at the leading edge and a net decrease in the upflow at the trailing edge. This is equivalent to a change in camber.

In order to counteract this change in loading on the wing, it was decided to determine the change in velocity due to the interference and then design an airfoil to produce an opposite change in velocity (delta velocity) of the same magnitude, so that the resulting combined pressure would be the same as the wing alone. For a two dimensional airfoil, the velocity at any point on the airfoil is a linear function of the circulation. If the flow is assumed to be two dimensional, at any constant percentage (chordwise) location point on the airfoil, the velocity should be a linear function of the circulation only. From the value of the doublet singularity, the circulation is known at each wing station, at least to within a constant of proportionality. Therefore, a plot of the velocities from all the wing stations at a given chord location versus the doublet singularity at that wing station should collapse to a straight line. Figure 4 is such a plot for a wing alone, and does indeed collapse

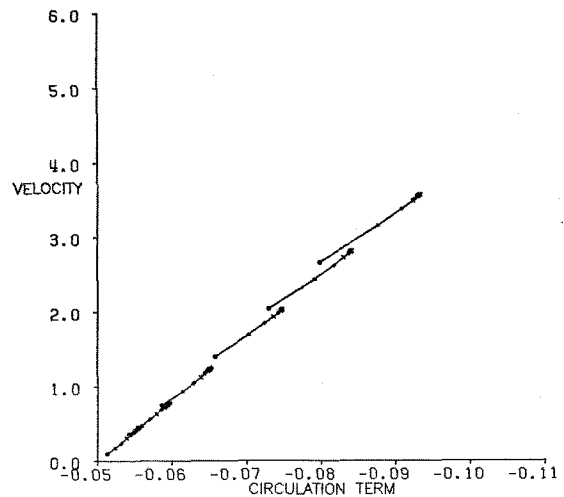


Figure 4- Velocity Versus Circulation for Wing alone.

to a straight line. This is a plot of only one of the points on the airfoil. In the model tested, there were 28 chordwise stations, and a similar plot was generated for each. Figure 5 is a plot for a wing-nacelle, and shows the influence of interference. From figure 5, a delta velocity due to interference can be determined. Therefore, the next step is to design an airfoil that has this same delta velocity with an opposite sign. Note that this entire procedure is dependent on angle of attack, therefore a design angle of attack must be chosen. For this study, the design angle of attack was chosen to be 7.6 degrees. This corresponds to the local angle of attack just outboard of the nacelle when the fuselage is at an angle of attack of 5 degrees, a typical climb condition.

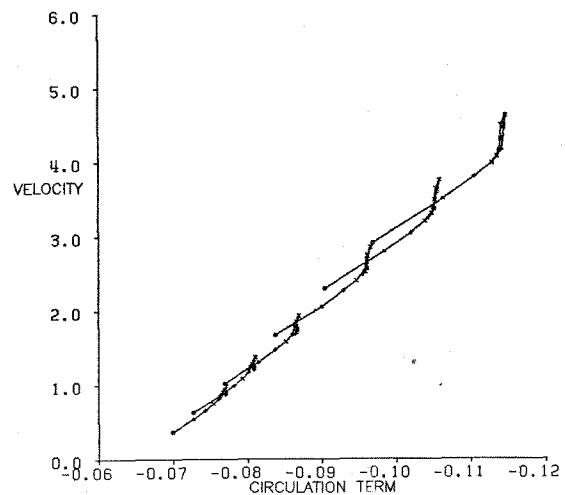


Figure 5- Velocity Versus Circulation for Baseline Wing-Nacelle.

For the design of the airfoil, the Mixed Analysis and Design code written by Bristow was chosen. This program is described in reference 3. In this code, the input can either be coordinates (for analysis) or velocity (for design). Also, the two options can be combined, so part of the airfoil can be input as geometry coordinates and part of the airfoil can be input as velocity, so the desired shape is generated in some regions but the present shape is retained in other regions. Also, the circulation can be solved (satisfying the Kutta condition) or can be input, so airfoils can be modified with constant circulation.

The first step in the procedure used here was to run a baseline analysis of the airfoil used at the design angle of attack, satisfying the Kutta condition with a variable circulation. From this run, baseline velocities and the baseline circulation were determined. The circulation was then fixed for all remaining runs. The delta velocities determined from the three dimensional runs were subtracted from the baseline velocities in the region of the airfoil to be redesigned (the entire airfoil was not redesigned, only the leading 10 percent of the upper surface and the leading 57 percent of the lower surface) and the program was run in the mixed mode. The airfoils generated by this procedure were then modeled in the three dimensional program, where the fully modified airfoils were applied at the intersections of the wing and nacelle and decreasing modifications were applied linearly from the nacelle inboard and outboard for a distance of about twice the maximum nacelle width. The inboard and outboard sections are different, as the wing intersects the nacelle differently inboard and outboard due to dihedral. The shapes of the resulting airfoils are shown in figures 6(a) and 6(b). A plot of the velocities versus circulation, similar to the earlier plots, is shown in figure 7. This data indicates that, at least at this point on the airfoil, the velocity is less than the desired velocity. In retrospect, the modification is probably too large.

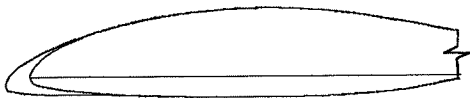


Figure 6a- Modified Wing Outboard of Nacelle.

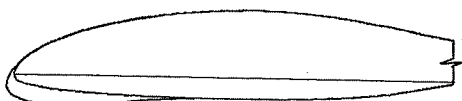


Figure 6b- Modified Wing Inboard of Nacelle.

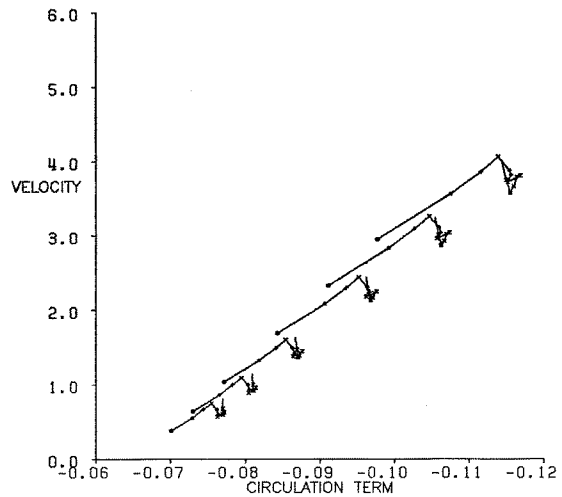


Figure 7- Velocity Versus Circulation for Modified Wing-Nacelle.

The modification designed by this method was tested in the wind tunnel. Plots of lift versus drag are shown in figure 8 for both the baseline and modified wing-body-nacelle combinations. As seen, there was no decrease in drag at any given lift, which implies no reduction in flow separation. Since this method yielded poor results, it was decided to attack the separated region on the fuselage in a different manner, as described in the following sections.

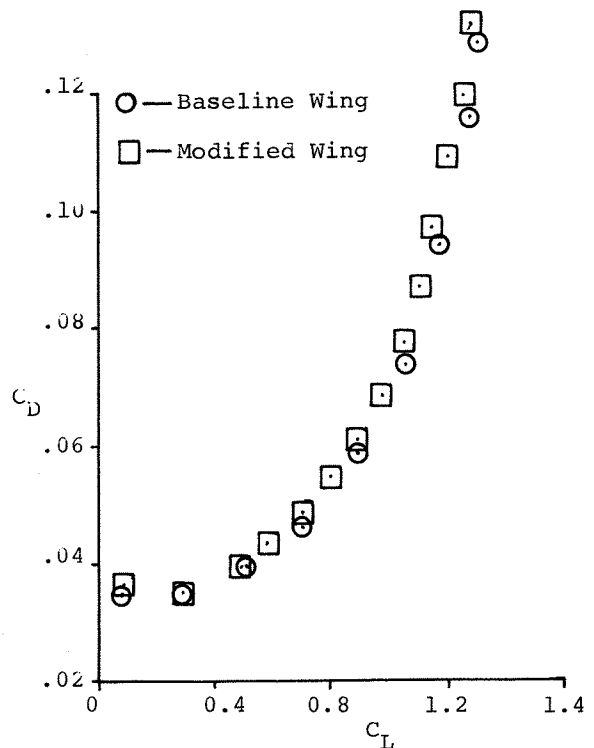


Figure 8- Effects of Wing Modification on Drag Polar.

Influence of the wing on a body

Figure 9 depicts the pressure along a row of panels on the fuselage starting at the leading edge of the fuselage, running just above the wing, and back to the trailing edge of the fuselage. The location of the wing is indicated by the wing plotted on the horizontal axis. The increased velocity induced by the wing is easily seen in this plot. This increased velocity is preceded by a long run along the fuselage, so the boundary layer is mature before the increase in velocity, and recovery is not accomplished as easily as on the wing.

The next logical step would be to model several fillets to see if the magnitude of the pressure spike could be reduced. Before this step could be conducted, a problem with the Hess code modeling had to be resolved.

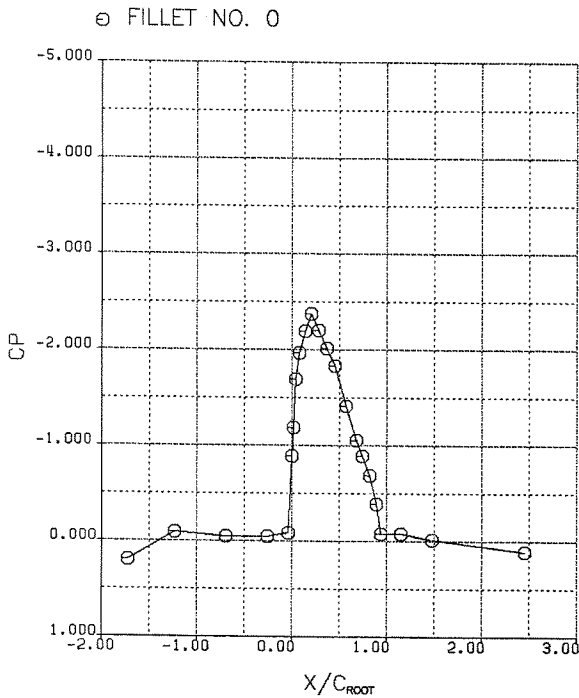


Figure 9- Pressure Distribution along Fuselage, $\alpha=5^\circ$.

A Problem of Modeling With the Hess Code

In the Hess code, panels that comprise a lifting surface are required to have parallel edges on the inboard and outboard sides. If panels are input with non-parallel sides, the program finds the closest approximation that has parallel sides. This creates one of the major problems in modeling wing-body intersections.

On the wing itself, the strips are normally input in constant y planes, so the panels have parallel edges naturally. However, the intersection line of the wing and body does not generally lie in a

plane, so the innermost strip of panels on a wing does not have parallel edges. The problem is illustrated in figure 10. In figure 10, there are three adjacent strips of panels, strip A, consisting of panels #1, #2, #3, and #4, strip B, consisting of panels #5 through #8, and strip C, consisting of panels #9 through #12. Strips A and B have parallel edges, while strip C has a curved line as the inboard edge. The input points are connected with solid lines, while the panel approximations are input as dashed lines. In strips A and B, the dashed lines are not visible, since the approximations follow the input points closely, but in strip C, the dashed lines do not follow the solid lines and are easily visible. Since the outboard edges of strip C do not align themselves with the inboard edges of strip B, the line vortices on the edges do not cancel, and spurious results can be obtained. This problem exists only on lifting surfaces, as there is no requirement of parallel edges on non-lifting surfaces.

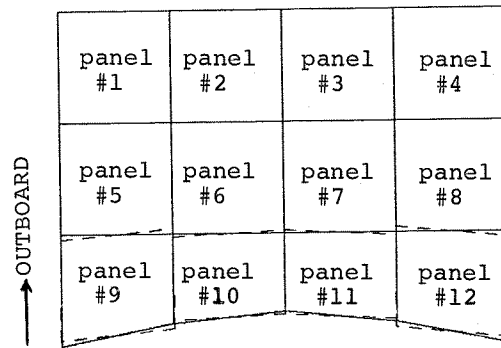


Figure 10- A problem of Modeling with the Hess Code.

First Solution To The Problem

Since non-lifting panels are not required to have parallel edges, the first solution to the problem was to panel the inboard strip as a non-lifting strip and have the carry-thru lattice go under this strip. In order to ensure that the carry-thru lattice was interior to the inboard strip, the inboard strip was panelled slightly oversized. This process of making the strip oversized must be done by moving the input points outward along the local normal. An indication of the error induced by this method by observing the pressures on the upper and lower surfaces of the non-lifting section. If these pressures are equal, the Kutta condition is satisfied, and the strength of the carry-thru lattice is such that there was no error induced. Analysis of computer runs made with this method indicate the upper surface generally had a lower pressure (higher velocity) than the lower surface, indicating the circulation was too great. The method of non-lifting strips was used in the design of fillets 1 through 14.

Second Solution To The Problem

Wind tunnel tests of the final fillet developed with the previous method did not show a major reduction in the area of separated flow on the body. However, further experimentation in the wind tunnel did result in a fillet that reduced the area of separated flow. This fillet had an upswept trailing edge on the inboard section of the wing. This involves a major change in the airfoil section, which would mean that the edges of these panels would not be parallel. Therefore, further changes in the model were called for.

The original Hess code has two types of panels, lifting (consisting of both source and doublet singularities) and non-lifting (consisting of source singularities only). This procedure was modified so all panels went through the non-lifting computations and the lifting computations were done for doublet singularities only. Once the source panels were separated from the doublet panels in this manner, the source and doublet panels do not have to be exactly concurrent, although the doublet panels should be interior to the source panels, which contain the control points. The modeling of the doublet panels was done with a constant airfoil wing, so all panels had parallel edges, and the source panels were modeled to include an upswept trailing edge near the intersection. The inboard doublet panels were carried to the centerline of the aircraft instead of using a separate carry-thru lattice. In order to ensure the doublet panels were interior to the source panels, the doublet panels were input slightly undersized, in a manner similar to the oversizing of the non-lifting panels of the previous method.

Since a wake must leave from the trailing edge of the wing, an extra panel was included in the wake of the inboard section. On the outboard edge, where the source and doublet panels were nearly concurrent, this extra panel had zero length, while on the inboard edge, the extra panel went from the trailing edge of the doublet panels to the trailing edge of the source panels. The source and doublet panels of the theoretical section at the centerline of the body are shown in figure 11. As this is the theoretical section, it is exaggerated. Since the outboard edge had zero length, it was parallel to the inboard edge, regardless what the direction of the inboard edge was. In actuality, the upswept region was modeled as two rows of panels, but since the upswept trailing edge was modeled as a straight line, the middle strip was parallel to the inboard strip. Fillets 15 through 18 were modeled with the use of the second method, with fillet 18 being similar to the successful fillet in the wind tunnel.

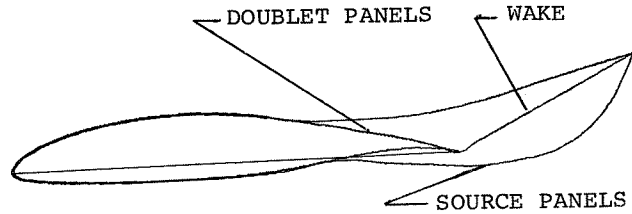


Figure 11 - Section of Fillet 18 at Centerline.

The theoretical pressures along the fuselage for the baseline fillet, fillet 14, and fillet 18 are plotted in figure 12. The baseline plot is a duplicate of figure 9. The lift versus drag polars for these three fillets is shown in figure 13. From this data, it can be seen that fillet 14 did not show a measurable improvement, but fillet 18 did show an improvement in the climb C_L region.

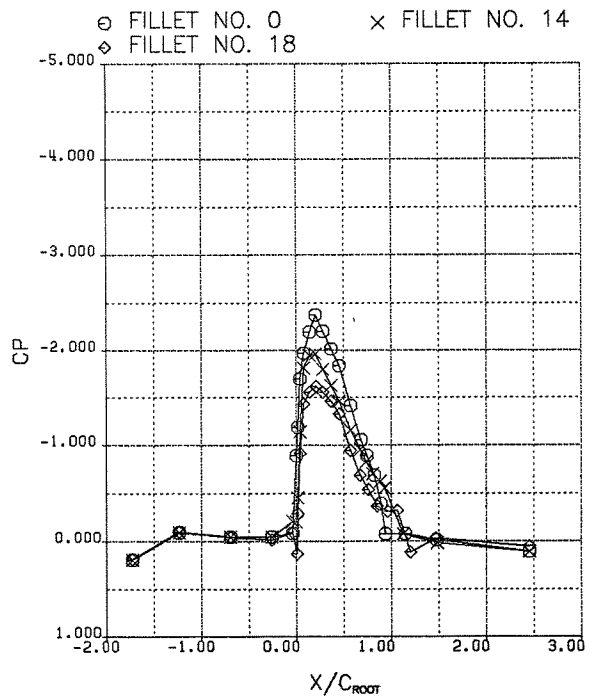


Figure 12- Pressure Distribution along Fuselage, $\alpha=5^\circ$.

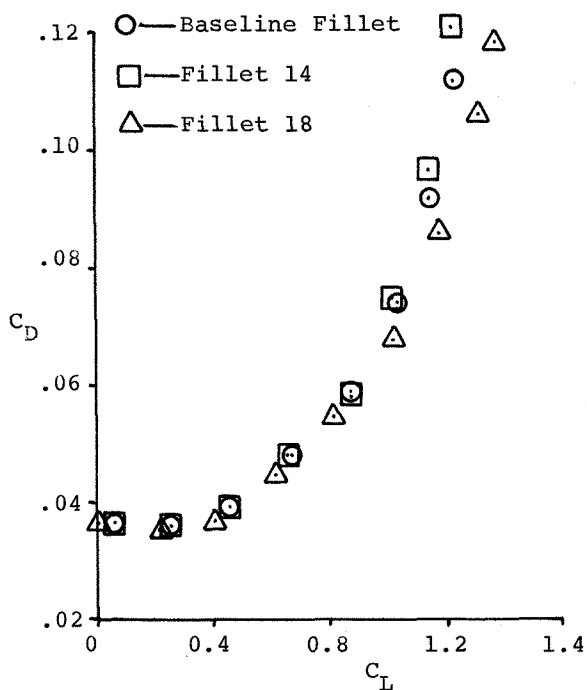


Figure 13- Effects of Fillet Modification on Drag Polar.

Discussion of Results

The results of this study indicate a potential flow program, such as a panel method potential flow program, can be useful in reducing interference effects, although no quantitative value of drag can be obtained from these programs. The parameter that needs to be minimized to reduce the interference effects seems to be the pressure spike on the body induced by the accelerated flow on the wing. Reducing the interference effects of the body on the wing does not seem to help as much. Probably the reason the body cannot recover from the pressure spike as easily as the wing is because the boundary layer on the body is mature and thick when the pressure spike occurs, while the boundary layer on the wing is new and thin when the minimum pressure occurs.

There does seem to be a minimum required fillet to attach the flow, where any fillet smaller than this does not give a significant improvement. Unfortunately, with the present level of knowledge, it is impossible to tell from potential flow when the fillet is large enough. The potential flow analysis can give direction as to how to change the fillet, but the final results have to come from experimental tests. With experience in using the analysis and correlation with experiment or a viscous solution, it may become possible to determine the level of spike suppression necessary for drag improvement.

Conclusions

1. The Hess panel method potential flow program must be modified before any analysis of a wing body intersection that is not planar can be accomplished. If the body has vertical sides where it intersects the wing, the program can handle the analysis, but most configurations have intersections that are not planar.

2. A method of modifying airfoil sections to reduce the influence of a body on a wing has been developed. For the configuration tested, this procedure did not reduce the drag, but for other configurations, this might be a useful procedure.

3. It was found that, while the first analysis of the wing-body did not provide a shape which reduced the drag, guidance was provided that would reduce the pressure spike on the side of the fuselage. After a successful fillet was obtained in the wind tunnel, the program was modified so this type of fillet could be modeled. This indicates that a potential flow solution can be useful in reducing interference drag.

Recommendations For Further Study

Although this study demonstrates that potential flow programs can be useful in reducing interference drag, there are many questions that remain to be answered on the subject. For instance, this study considered only the pressure gradient along the direction of flow, but there is a strong gradient normal to the flow on the fuselage. The study of this flow with a three-dimensional viscous program would be of interest.

The pressure spike discussed in this study probably results in separation and/or a discrete shed vortex. It would be of interest to study the region of the intersection, either using viscous analysis or experiments to determine the nature of this region, where the flow is dominated by viscous effects.

References

- Hess, J. L.: "The Problem of Three-Dimensional Lifting Potential Flow and It's Solution by Means of Surface Singularity Distributions", Computer Methods in Applied Mechanics in Engineering, Vol. 4, No. 3, Nov., 1974.
- Somers, Dan M.: "Design and Experimental Results for a Flapped Natural-Laminar-Flow Airfoil for General Aviation Applications", NASA Technical Paper 1865, June, 1981. (Date for general release, June 30, 1983.)
- Bristow, D. R.: "Development of Panel Methods for Subsonic Analysis and Design", NASA Contractor Report 3234, Sept., 1979.