Numerical Aspects of Transition Prediction for Three-Dimensional Configurations

N. Krimmelbein*, R. Radespiel†
Institute of Fluid Mechanics, Braunschweig Technical University, 38106 Braunschweig, Germany

and

C. Nebel‡
CeBeNetwork, 28199 Bremen, Germany

The feasibility to predict transition for three dimensional configurations is presented by means of a coupled program system consisting of a 3D Navier-Stokes solver, a transition module and a stability code. The assumption to use inviscid streamlines as integration paths for the \( N \)-factor calculation makes it possible to use linear stability theory in a straightforward way for three dimensional flows. The developed transition module has been adapted to be used with sequential and parallel computations to account for the increased computational demand for three dimensional configurations. Detailed investigations have been carried out, to show the ability of the Navier-Stokes code to provide data of three dimensional boundary layers of high accuracy needed for the stability analyses. Applications of the transition prediction method using the \( 2N \)-factor method for the case of an inclined prolate spheroid shows reasonably good results compared to experiments. First application of the transition prediction tool on a generic transport aircraft show promising results for the ability to predict transition on complex geometries.

Nomenclature

\[
\begin{align*}
\alpha & = \text{angle of attack} \\
\alpha_i, \beta_i & = \text{spatial amplification numbers} \\
\alpha_r, \beta_r & = \text{wave numbers} \\
\beta & = \text{twist angle of the velocity profile} \\
c_p & = \text{local pressure coefficient} \\
c & = \text{chord length} \\
\delta & = \text{boundary layer thickness} \\
M & = \text{Mach number} \\
N & = \text{N-factor} \\
N_{\text{crit}} & = \text{critical N-factor} \\
\hat{r} & = \text{vector of amplitude functions} \\
Re & = \text{Reynolds number} \\
Tu & = \text{turbulence level} \\
\hat{u}_e & = \text{free stream velocity} \\
u_e & = \text{boundary layer edge velocity}
\end{align*}
\]
THE prediction of laminar-turbulent transition in Reynolds-averaged Navier-Stokes solvers (RANS) plays a growing role for the computation of complex flows in aircraft industries. For the flow past transport aircrafts not only the transition from laminar to turbulent flow on airfoils with high aspect ratio is of importance, but also the transition location on bodies, tailplanes, nacelles, and around intersections of these components has to be known in order to accurately predict the aerodynamic performance.

State of the art for transition prediction on airfoil, tailplane and nacelle surfaces is the linear local stability theory applied in form of the $e^N$-method. The assumption behind this theory is that the breakdown of wave-like disturbances in the laminar boundary layer leads to the transition to turbulent flow. The amplification of these disturbances is defined by the N-factor. Transition is expected to occur where the amplitude of the most unstable disturbance has reached a factor $e^N$ compared to the initial amplitude at the beginning of the unstable region in the boundary layer.

Generally, boundary layer transition can be caused by many different mechanisms. In fully three dimensional boundary layers, there may exist a significant crossflow velocity component that leads to crossflow (CF) instability, whereas in more two-dimensional flow regions the flow is unstable due to Tollmien-Schlichting (TS) waves. Both, crossflow and Tollmien-Schlichting instabilities can be described by the $e^N$-method.

At the Institute for Fluid Mechanics at Braunschweig Technical University a transition prediction module was previously integrated into the Reynolds-averaged Navier-Stokes code TAU. This module uses the 3D RANS flow field to define integration paths for the calculation of the N-factor and takes the RANS boundary layer data as mean flow for the stability analysis. Crossflow and Tollmien-Schlichting instabilities are analyzed with database methods of Casalis and Arnal and Stock and Degenhart respectively. The module is capable of predicting transition in 2D and 2.5D flows on airfoils and infinite swept wings and on simple geometries for 3D flows.

This module has been used as a base to develop an enhanced version for the analysis of flows past more complex configurations. To improve the accuracy of the stability analysis, the database methods are replaced by the linear stability code COAST. Especially for prediction of transition over 2D laminar separation bubbles an advancement was already achieved with this new code version. The previous transition module could only be run in sequential mode and had to be converted, with regard to decent turn around times, to a parallel version. For complex 3D cases with grids with suitable resolution of the boundary layers the supply of a parallel code version is absolutely necessary. The present paper describes various numerical aspects in the development of the current advanced transition prediction method. Implementation issues of assembling non-local flow data needed for stability analysis are discussed. Results for code verification and validation are presented along with first results for a complex aircraft configuration that demonstrate the potential of the method.

II. Numerical methods

A. Reynolds-averaged Navier-Stokes Solver

The DLR TAU code used for the present work is a three-dimensional Reynolds-averaged Navier-Stokes (RANS) solver based on a finite volume scheme and using hybrid meshes. The code can be run with implicit or explicit integration schemes and includes several acceleration techniques such as multigrid and preconditioning. Central schemes or various upwind schemes are available for spatial discretisation. Turbulent flows are modeled using Spalart-Allmaras or Wilcox k-omega turbulence models or modifications of these. Transition modeling is done by
limiting the production terms of the turbulence equation in laminar regions to be smaller than their destruction counterparts.

B. Transition prediction method

Transition prediction is performed by using linear stability theory in form of the well known \( e^{i\omega t} \)-method. For this purpose, a coupled system consisting of the RANS solver TAU with a newly included transition prediction module and the stability solver COAST\(^3\) has been developed. The transition prediction module extracts the boundary layer data from the RANS solution and preprocesses the data in an appropriate way to COAST\(^3\) format. The output of the stability program is then used by the prediction module to determine new transition locations.

The classical linear stability theory evaluates the stability of a laminar boundary layer by examining the development of small disturbances. The principal approach of the theory is to introduce harmonic disturbances into the linearised Navier-Stokes equations of the form

\[
\frac{d^2}{dy^2} \hat{r} + A \frac{d}{dy} \hat{r} + B \hat{r} = \omega C \hat{r}
\]  

with circular frequency \( \omega \) and wave numbers \( \alpha \) and \( \beta \). \( \alpha, \beta \) are generally complex numbers.

Solutions can be obtained by solving the resulting system of second order differential equations for the amplitude function \( \hat{r} \):

\[
\frac{d^2}{dy^2} \hat{r} + A \frac{d}{dy} \hat{r} + B \hat{r} = \omega C \hat{r}
\]  

The \((5 \times 5)\) matrices \( A, B, C \), depend on Reynolds number, Mach number and on the local velocity and temperature profiles and their first and second derivatives. The vector \( \hat{r} \) contains the amplitude functions for velocities, temperature and pressure. The five equations form a generalized linear eigenvalue problem for the complex eigenvalue \( \omega \) when prescribing \( \alpha \) and \( \beta \). The stability code COAST\(^3\) solves this eigenvalue problem by the use of a QR-decomposition or a generalized inverse Rayleigh iteration\(^9\).

However, the local eigenvalue problem contains the six parameters \( \omega, \alpha, \beta, \alpha_i, \beta_i \) and \( \beta \) and only determines two of them. Hence, four conditions are needed to connect the eigenvalue problem: 1. constant frequency along the group velocity trajectories. 2. use of either spatial or temporal theory. 3. direction of the group velocity is the amplification direction. The remaining fourth condition leads to four different integration strategies:\(^9\):

- \( N_{TS} \)-factor: Frequency and propagation direction prescribed
- \( N_{CF} \)-factor: Frequency and wave length prescribed
- \( N_{b*} \)-factor: Frequency and dimensional spanwise wavenumber prescribed
- Envelope method

In the present work, the \( 2N \)-factor strategy with use of the \( N_{TS} \)-factor and the \( N_{CF} \)-factor is considered, treating the N-factors for crossflow and Tollmien-Schlichting instabilities independently. The \( N_{TS} \)-factor is calculated by assuming that the propagation direction coincides with the inviscid flow direction. For the \( N_{CF} \)-factor only stationary crossflow waves are considered, hence prescribing a frequency of 0Hz. An envelope enclosing the N-factor curves for a certain frequency range, or a wavelength range respectively, is then used to determine the transition location.

One basic assumption within this work is to approximate the group velocity trajectory with an inviscid streamline at the edge of the boundary layer. The direction of the group velocity is considered as amplification direction and N-factors would be integrated along the trajectory. By using the inviscid streamline assumption a simple way is found to determine transition locations with the \( e^{i\omega t} \)-method in three dimensional flow. According to experiences with infinite swept wings, the difference between the inviscid flow direction and the direction of the group velocity is only a few degrees\(^2, 4, 10\). Hence, the transition module is used to calculate inviscid streamlines, extract the boundary layer data and velocity profiles from these streamlines and provide the data as input for the stability solver COAST\(^3\). The stability code calculates the local amplification rates which are then integrated by the transition module along the streamlines to give the N-factor curves. New transition locations on the streamlines are found by comparing the N-factors with a critical value generally deduced from Mack’s formula\(^11\).
C. Parallelization issues

The DLR TAU code uses domain decomposition for parallel computing. For a given number $n$ of processors the computational domain (the grid) is divided into $n$ subdomains. Each of the processors computes on one of the subdomains. Since the solutions on the subdomains depend on each other, a continuous communication between the processes is required. The TAU code uses MPI 1.0\textsuperscript{12} for these communication purposes between the different processes.

For acceptable turnaround times for the calculation of flows around complex geometries, the transition prediction procedure had to be adapted to parallel computations. Whereas the computation of the basic flow solution requires only local data (only data at the domain boundaries is communicated), for the transition prediction process non-local data is needed as well.

Basically, parallelization in means of transition prediction is needed for the determination of wall normals, the search for the boundary layer edge, the extraction of whole velocity profiles and the calculation of the inviscid streamlines. To limit the amount of memory usage, velocity profiles are generally only assembled on parts of the solution domain where they actually will be needed, i.e. along the previously determined streamlines in the vicinity of the domain edges. On the other hand, the components of the boundary layer edge velocity are needed over the whole solution domain before the calculation of the streamlines is performed as these velocities are used for the integration. Hence, the assembly of the velocity profiles and the determination of the edge velocity are considered as different parallelization tasks. To calculate boundary layer data over the whole solution domain for output and visualization purposes an option is included to assemble velocity profiles at any boundary of the subdomains.

The basis of the whole boundary layer calculation process is the knowledge of the wall normals and the grid points associated with these. The calculation of the wall normals is done in a preprocessing step and their length is limited by a maximal number of points on the normal and a maximum distance. The determination of the wall normals is performed for each surface point of the computational domain. Starting from these points, the next points on the wall are determined by using a visible cone method until the domain boundary is reached. The end points of the uncompleted normals will be communicated to the neighbour domains to give the start points of the next part of the normal line. This process is repeated until all wall normals have been completed. From the knowledge of the normals the search for the boundary layer edge can be started. Along the normals the mean velocity is compared to a scaled value of a theoretical edge velocity deduced from the surface pressure. Generally, the boundary layer edge is assumed to be found when the local mean velocity exceeds 98% of the theoretical boundary layer edge velocity. The velocity components of the so found boundary layer edge velocity have then to be communicated to the domain owning the base point of the wall normal. Fortunately, the domain decomposition procedure implemented in TAU generally creates domain boundaries perpendicular to the surfaces, which keeps the number of splitted wall normals to a minimum. Nevertheless, divided normals can not be completely avoided for grids around complex geometries and even for very simple geometries steps at the domain edge in the structured part of the grid may occur. An example is given in Fig. 1, where a typical discontinuity on a 2D grid, decomposed into seven subdomains, is highlighted. As reference the boundary layer edge is depicted as well.

The assembly of the velocity profiles is straightforward. For each normal part the velocities will be communicated to the domain containing the first part of the considered wall normal. From the completed profiles boundary layer parameters may be deduced.

The calculation process of the streamlines resembles the one for the wall normals. Starting from certain initial stations, the streamlines will be integrated into two directions until a domain boundary is reached. The end points give then the next startpoints in the neighbouring domain. The process of streamline integration is continued until all streamlines have reached their end coordinates, given by a trailing edge, an attachment line or a user specified geometrical constraint. Attachment lines calculated prior to the streamline calculation. Whereas the parallelization strategies are the same as for the streamlines, the wall friction vector is used for integration instead of the boundary layer edge velocity. This is done since the wall friction vectors will be provided as part of the solution process of the Navier-Stokes solver and lead to smoother attachment lines when used for integration. Calculated streamlines and a typical decomposed calculation domain for a prolate spheroid are presented in Fig. 2.

From the above described strategies it is clear that parallelization of the transition module does not mean that the whole calculation process is performed parallel. Instead, the emphasis lies on the aspect, that the transition module is capable of dealing with decomposed grids and solutions. However, the amount of parallel performed routines in the transition module is kept to a maximum.
III. Results

A. Infinite swept wing

Tests have been performed to analyze the influence of grid density on the calculation of N-factors with the stability code COAST3. For this study, an infinite swept wing model with ONERA D profile\textsuperscript{13} has been used. The angle of attack was set to $\alpha_0 = 4^\circ$, with Mach number $M = 0.23$, Reynolds number $Re = 2.39 \times 10^6$ and sweep angle $\lambda = 60.0^\circ$. Transition was prescribed on the upper wing surface at $x/c = 0.03$ and on the lower wing surface at $x/c = 0.85$. For the Navier-Stokes computations different hybrid meshes have been generated using the grid generator Centaur\textsuperscript{4} with different grid resolutions in the structured part in circumferential and normal direction. The chosen range was from 128x32 cells to 512x128 cells in the structured part of the hybrid grid. An a priori estimation of the laminar boundary layer thickness has been used to ensure that the laminar boundary layer is covered by the structured grid part. The mesh with 256 cells along the surface and 64 cells normal to the surface is shown in Fig. 3. To improve accuracy and convergence of the solution the calculations were performed using low Mach number preconditioning.

Taking the $c_p$-distribution (Fig. 4) from the converged Navier-Stokes solution as input, calculations with the boundary-layer code COCO\textsuperscript{15} have been carried out for comparison purposes as well.

To evaluate the quality of boundary layer data based on the velocity profiles within the fully three dimensional boundary layer, the momentum loss thicknesses of the streamwise velocity profile $\theta_u$ and the crossflow velocity profile $\theta_w$ have been calculated. As can be seen in Fig. 5, the calculated values from the Navier-Stokes computation are in good agreement with those values obtained from boundary layer calculation, especially for the streamwise displacement thickness $\theta_u$, but differences occur for the crossflow displacement thickness on the coarser grids.

An assessment of the boundary layer velocity profiles confirms these results. The velocity profiles and their first and second derivatives at $x/c = 0.2$ and $x/c = 0.65$ for the lower wing surface are shown in Fig. 6 and 7. The flow is three dimensional and mean velocity profiles can be decomposed into a streamwise profile $u$ in the direction of the external streamline and a crossflow profile $w$ in the direction normal to this line. Fig. 6 and 7 show the typical behaviour of the crossflow profile along the chord. In the region of negative pressure gradient in the vicinity of the leading edge the crossflow profile is directed towards the concave part of the streamline. As a region with positive pressure gradient is reached, the curvature of the streamline changes. The crossflow velocity near the wall begins to reverse and an S-shaped velocity profile develops\textsuperscript{2}. From Fig. 5-7 it is clearly visible that for the streamwise profiles a grid resolution of 48 cells normal to the wall is needed, whereas for the crossflow profiles approximately 128 cells are needed to match the results obtained from boundary layer code computations.

The accuracy of the velocity profiles has a direct influence of the computed N-factors as the profiles are the main input for the stability code. The progression of the resulting N-factor for Tollmien-Schlichting instabilities is depicted in Fig. 8. As already expected, here the use of velocity profiles from RANS solutions using a grid resolution of at least 32 cells normal to the wall gives reasonably good results, compared to values obtained with velocity profiles from boundary layer computation. To the contrary, when calculating crossflow N-factors a much higher grid resolution is needed to confirm results calculated with boundary layer code data (Fig. 9).

In conclusion, it has to be expected that for the prediction of TS-instabilities a resolution of 32-48 cells normal to the wing surface in the structured mesh part will be sufficient, while for the accuracy of calculating crossflow N-factors a much higher resolution is needed.

B. Prolate spheroid

Studies on the prediction of transition in fully three dimensional flow by the present method have been performed for the 6:1 prolate spheroid of Meier and Kreplin\textsuperscript{16} at an angle of attack of $\alpha = 10^\circ$, with Mach number $M = 0.14$ and Reynolds number $Re = 7.2 \times 10^6$. The hybrid grid used for the calculation consists of 128 points normal to the surface in the structured grid part and has 3 million points in total. According to Mack’s formula\textsuperscript{15} and a turbulence level of the experiment\textsuperscript{14} of $Tu = 0.1\% - 0.2\%$ a critical N-factor of $N_{crit} = 7.2$ has been used. To accelerate convergence and to improve the accuracy of the numerical result an implicit integration scheme along with preconditioning was applied. The turbulent flow was modelled using the standard Spalart-Allmaras turbulence model.

Fig. 10 shows the surface pressure distribution for the prolate spheroid and 13 of overall 25 external streamlines along which the amplification rates were calculated with the stability code COAST3 and transition was determined. As a result of the use of the 2N-factor method a clear distinction between transition due to TS and CF instabilities can be made. This effect is illustrated by representing the resulting transition line with a two-coloured line in Fig. 11. To highlight the regions with strong crossflow the maximum twist angle $\beta$ of the mean velocity profile has been
used as contour plot in Fig. 11. This twist angle is defined as angle between the outer flow direction and the local flow direction inside the boundary layer and is therefore a good measure for the strength of the crossflow velocity component. As expected, the transition due to CF instabilities occurs in the region of strongly three dimensional flow, whereas the transition due to TS instabilities takes place in regions with more 2D-like flow. The effect of transition (resp. turbulent flow) on the crossflow velocity component is visible from Fig. 11 as well. As a result of the increased momentum transfer in turbulent flow the crossflow decreases which causes a much smaller twist angle right behind the line of transition.

The evaluation of the N-factor curves along the five streamlines (Fig. 11-13) highlights one major challenge when applying linear stability theory to Navier-Stokes flow solutions. In contrast to boundary layer code calculations the transition has a significant influence on the velocity profiles inside the boundary layer. Right behind the transition turbulent velocity profiles develop, for which no amplifications will be found when analysing them with COAST3. In addition, a slight upstream influence may be encountered, affecting the stability calculations right before the currently set transition position. This may lead to the effect that no N-factors reaching the critical value can be found. A solution for this problem is to extrapolate the envelopes of the N-factor curves into the (artificially) damped regions. This technique is indicated in Fig. 12 and 13.

After all, a good agreement for the line of transition compared to experimental results deduced from Ref. 16 has been achieved (Fig. 14). For comparison purposes numerical results from T. Cebeci27, which were calculated with the saddle point method of Cebeci and Stewartson18 with a limiting N-factor of 10, have been plotted as well. In contrast to the transition line calculated with boundary layer data from Navier-Stokes solution, Cebeci used a boundary layer code to create the input data for his stability code. Again, a fairly good agreement between the results is attained.

C. Generic transport aircraft

To show the feasibility of the transition prediction method for complex three-dimensional configurations a test case consisting of a generic transport aircraft model has been generated. The objective was to predict transition on each of the surfaces of the model, i.e. body, upper and lower surface of the wing and the vertical tailplane and horizontal tailplane. A very moderate point distribution for the surface normal grid fineness was chosen, so that on all surfaces 32 points have been distributed in the structured part of the grid normal to the wall, except for the tailplane surfaces, where the number of points was increased to 48. The amount of points normal to the walls guarantees a fairly accurate resolution of the boundary layer while keeping the computational amount relatively low. The overall number of points for this test case is 12 million. To improve accuracy and convergence of the RANS solution an implicit integration scheme has been used in combination with low Mach number preconditioning. The turbulent flow was modelled using the standard Spalart-Allmaras turbulence model.

The flow conditions have been chosen to provide attached flow conditions. Hence, only a moderate angle of attack of $\alpha = -4.0^\circ$ and a similarly low incidence angle for the tailplane of $i_{\text{t}} = 4.0^\circ$ have been used, along with a Mach number of $M = 0.2$ and a Reynolds number of $Re = 2.3 \times 10^6$.

To account for the results presented in chapter III.A only transition due to Tollmien-Schlichting instabilities has been considered. It can be assumed that for accurate prediction of crossflow instabilities the grid resolution on all surfaces is much too coarse.

In addition, as even for the main part of the body the grid density in flow direction was too low, transition was set on this surface a short distance after the local pressure minimum. However, amplifications were calculated along the streamlines originating at the nose of the airplane but were considered as numerically inaccurate. On the upper wing, transition has been set just in front of the laminar separation developing at the deflected flap. Although the calculated amplifications indicate transition to occur after the separation this procedure has been chosen to avoid convergence problems. On all other surfaces transition was assumed to take place when the integrated amplifications reached an arbitrarily chosen critical N-factor of 7.5 (corresponding to a turbulence level of approx. 0.13%).

The extraction of boundary layer parameters from the Navier-Stokes solution is one of the processes performed during the application of the transition module. Here a determination of the boundary layer thickness developing on the transport aircraft is presented. The result is shown in Fig. 15 and displays the trace of horse-shoe vortices developing in the junction of the body with the wing.

Figures 16 and 17 show the calculated integration paths together with the final transition lines. On all surfaces six streamlines have been determined along which the amplifications have been calculated with the stability code COAST3. For the determination of the local pressure minimum on the body eleven streamlines have been calculated. At this point it is concluded that the predicted transition positions are physically plausible and show no irregularities. To prescribe a new transition position and to create laminar zones in the calculation domain of the RANS solver, the transition lines have been extrapolated to exceed the boundaries of the single surface parts. The so
found polygon lines are introduced into the calculation process of the RANS solver and laminar regions are created as presented in Fig. 18.

**IV. Conclusion and outlook**

The present work shows that it is feasible to calculate transition for complex three dimensional configurations by applying the linear stability theory and the 2N-factor method.

A detailed analysis of the influence of grid refinement is presented. It is demonstrated that it is possible to extract boundary layer profiles and quantities from a 3D Navier-Stokes flow solver accurate enough for stability analysis. However, the limitations in form of the required high grid resolution for the prediction of crossflow instabilities have been shown.

A major achievement in predicting transition on a 6:1 prolate spheroid has been made. The accuracy of the predicted transition has been improved compared to earlier attempts. The improvement was mainly achieved by the replacement of the database methods with the stability code COAST3. It has to be mentioned, that in previous examinations the database method of Casalis and Arnal yielded similar results for CF-instabilities on the swept wing case from chapter III.A as the stability code. The differences in predicting transition positions for the prolate spheroid most likely came from malfunctions of the interface with the external database. A further investigation of this problem will be carried out in future.

First applications of the 2N-factor method to a three dimensional configuration have been performed and show physically plausible results. Comparisons to experimental data have yet to be accomplished.

**Acknowledgments**

The numerical investigations were performed as part of the MEGADESIGN programme and were partly founded by the Deutsche Forschungsgesellschaft (DFG). A main tool used in the transition module is the stability code COAST3 which was developed and provided by Geza Schrauf. These supports are thankfully acknowledged by the authors.

**References**

Figures

Figure 1: Boundary layer edge on a decomposed grid
2D test calculation, 7 domains

Figure 2: Inviscid streamlines on a prolate spheroid
parallel computation, 8 domains

Figure 3: Hybrid grid for ONERA D profile

Figure 4: Pressure distribution on ONERA D profile
\( \alpha_e = 4, M = 0.23, Re = 2.39 \times 10^6, \lambda = 60.0^\circ \)
Figure 5: Momentum loss thicknesses $\theta_u$ and $\theta_c$
ONERA D profile, results from Navier-Stokes solution on different meshes compared to boundary-layer code solution, lower profile surface

Figure 6: Velocity profiles
ONERA D profile, results from Navier-Stokes solution on different meshes compared to boundary-layer code solution; $x/c = 0.2$, lower surface
Figure 7: Velocity profiles
ONERA D profile, results from Navier-Stokes solution on different meshes compared to boundary-layer code solution; $x/c = 0.65$, lower surface

Figure 8: $N_{TS}$-factors from stability code COAST3
Input data from Navier-Stokes solution compared with input data from boundary-layer code solution, lower surface

Figure 9: $N_{CF}$-factors from stability code COAST3
Input data from Navier-Stokes solution compared with input data from boundary-layer code solution, lower surface
Figure 10: Surface pressure distribution and inviscid streamlines
prolate spheroid, $\alpha = 10^\circ$, $M = 0.14$, Reynolds number $Re = 7.2 \times 10^6$

Figure 11: Transition location on a prolate spheroid
$\alpha = 10^\circ$, $M = 0.14$, Reynolds number $Re = 7.2 \times 10^6$

Figure 12: $N_{TS}$-curves for integration paths 1, 4, 5 (see Fig. 9)
Figure 13: $N_{CF}$-curves for integration paths 2, 3 (see Fig. 9)

Figure 14: Transition locations on prolate spheroid
(Experimental data taken from Ref. 17)

Figure 15: Boundary layer thickness on generic transport aircraft
$\alpha = -4.0^\circ$, $\iota_H = 4.0^\circ$, $M = 0.2$, $Re = 2.3\times10^6$. 

American Institute of Aeronautics and Astronautics
Figure 16: Integration paths and transition locations
Generic transport aircraft, upper surfaces

Figure 17: Integration paths and transition locations
Generic transport aircraft, lower surfaces

Figure 18: Laminar (bright) and turbulent (dark) Regions
Generic transport aircraft