GARTEUR LIMITED

A STUDY OF THE AERODYNAMICS OF AN INTAKE BUMP COMPRESSION SURFACE AT SUBSONIC AND SUPERSONIC SPEEDS USING CFD METHODS

P G Martin, J Hodges, A Gaiddon, P Curtis,
N C Bissinger, K Bradbrook, J E J Maseland,

The present report is distributed on a limited basis, for the information of the listed organisations only; consequently filing in any central library open to others, and citation in accession lists, or as literature reference, are prohibited.

GARTEUR aims at stimulating co-operation between Research Establishments in the areas of Aerodynamics, Flight Mechanics, Structures & Materials and Propulsion.
Distribution

1 copy for each member of Action Group AD AG-34

Mr. P. G. Martin          Dstl          UK
Mr. J. Hodges             QinetiQ        UK
Mr. P Curtis              BAE SYSTEMS   UK
Mr. K Bradbrook           BAE SYSTEMS   UK
Mr. A. Gaiddon            MBDA          France
Dr. N. C. Bissinger       EADS-MAS      Germany
Mr. J. E. J. Maseland     NLR           The Netherlands

Members of the Group of Responsables for Aerodynamics

Mr. P. Champigny           ONERA         France
Dr. R. Rudnik              DLR           Germany
Mr. G. Mingione            CIRA          Italy
Mr. K. de Cock             NLR           The Netherlands
Dr. K. Petterson           DSTL          UK

IPoC members of the Group of Responsables for Aerodynamics

Mr. D. Pagan               MBDA France   France
Mr. T. Pinar               EADS Deutschland Germany
Mr. C. Newbold             QinetiQ        UK

Heads of the National Delegation

ICA P. Ramette            DGA/CEP       France
Mr. F.-J. Mathy            BMWA          Germany
Capt R Dekker              MoD           The Netherlands
Mr. T Birch                Dstl          UK

1 copy for the Secretary for the GARTEUR central file

N. Ewinger                 DLR           Germany
# Contents

1. **Introduction** ................................................................................................................. 1
   1.1 Background .................................................................................................................. 1
   1.2 Form of the Collaboration ............................................................................................ 2
   1.3 References .................................................................................................................. 3

2. **Objectives** ..................................................................................................................... 4
   2.1 Outline of the Study Case ............................................................................................. 4
   2.2 Onset Flow Conditions and Allocation of the Calculations ........................................ 5
      2.2.1 Matrix .................................................................................................................... 5
      2.2.2 CFD Codes .............................................................................................................. 5
      2.2.3 Resources ............................................................................................................... 6

3. **CFD Codes Used** .......................................................................................................... 7
   3.1 Code Descriptions ......................................................................................................... 7
   3.2 The Calculation of Boundary Layer Displacement Thickness, ($\delta^*$) ...................... 8

4. **Geometry and Flow Conditions** .................................................................................... 10
   4.1 Bump Geometry .......................................................................................................... 10
      4.1.1 Stream-Surface Definition ..................................................................................... 10
      4.1.2 Rear Geometry ...................................................................................................... 11
      4.1.3 Smoothing .............................................................................................................. 11
      4.1.4 Flat Plate ................................................................................................................. 12
      4.1.5 Geometry for CFD Calculations .............................................................................. 12
   4.2 Flow Conditions ........................................................................................................... 13
      4.2.1 Mach Numbers ....................................................................................................... 13
      4.2.2 Cross-Flow Angles ............................................................................................... 13
      4.2.3 Boundary-layer Thicknesses ................................................................................ 13
   4.3 References .................................................................................................................... 14

5. **Results of CFD Code Comparison** .............................................................................. 15
   5.1 $M=1.8 \delta^*/k=0.07$ .................................................................................................... 15
      5.1.1 Centreline Data ...................................................................................................... 15
      5.1.2 Lateral Cuts at Constant X ................................................................................... 17
   5.2 $M=0.8 \delta^*/k=0.7$ ..................................................................................................... 20
      5.2.1 Centreline Data ...................................................................................................... 20
      5.2.2 Lateral Cuts at Constant X ................................................................................... 21
   5.3 Discussion ..................................................................................................................... 24

6. **Presentation and Analysis of Effect of Mach Number** .................................................. 25
   6.1 Low Mach Range ($M = 0.6 – 1.8$) ............................................................................ 25
      6.1.1 Mesh Used .............................................................................................................. 25
      6.1.2 Boundary Conditions ............................................................................................ 25
      6.1.3 Results .................................................................................................................... 25
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1.4</td>
<td>Flow Physics</td>
<td>27</td>
</tr>
<tr>
<td>6.1.5</td>
<td>References:</td>
<td>27</td>
</tr>
<tr>
<td>6.2</td>
<td>High Mach Range (M = 1.8 to 3.5)</td>
<td>33</td>
</tr>
<tr>
<td>7.</td>
<td>Presentation and Analysis of Effect of Sideslip</td>
<td>36</td>
</tr>
<tr>
<td>7.1</td>
<td>Subsonic (M=0.8)</td>
<td>36</td>
</tr>
<tr>
<td>7.1.1</td>
<td>Mesh Used</td>
<td>36</td>
</tr>
<tr>
<td>7.1.2</td>
<td>Boundary Conditions</td>
<td>36</td>
</tr>
<tr>
<td>7.1.3</td>
<td>Results</td>
<td>36</td>
</tr>
<tr>
<td>7.2</td>
<td>Effect of Sideslip (M = 1.8)</td>
<td>41</td>
</tr>
<tr>
<td>7.2.1</td>
<td>Mesh and Boundary Conditions</td>
<td>41</td>
</tr>
<tr>
<td>7.2.2</td>
<td>Results</td>
<td>41</td>
</tr>
<tr>
<td>7.3</td>
<td>Mach=3.5</td>
<td>45</td>
</tr>
<tr>
<td>8.</td>
<td>Presentation and Analysis of Effect of Reynolds Number (Flat Plate Length)</td>
<td>47</td>
</tr>
<tr>
<td>8.1</td>
<td>Subsonic (M=0.8)</td>
<td>47</td>
</tr>
<tr>
<td>8.1.1</td>
<td>Mesh Used</td>
<td>47</td>
</tr>
<tr>
<td>8.1.2</td>
<td>Boundary Conditions</td>
<td>47</td>
</tr>
<tr>
<td>8.1.3</td>
<td>Results</td>
<td>47</td>
</tr>
<tr>
<td>8.1.4</td>
<td>References</td>
<td>48</td>
</tr>
<tr>
<td>8.2</td>
<td>Supersonic (M=1.8)</td>
<td>59</td>
</tr>
<tr>
<td>8.2.1</td>
<td>Meshes</td>
<td>59</td>
</tr>
<tr>
<td>8.2.2</td>
<td>Boundary conditions</td>
<td>60</td>
</tr>
<tr>
<td>8.2.3</td>
<td>Results</td>
<td>61</td>
</tr>
<tr>
<td>8.2.4</td>
<td>Discussion</td>
<td>63</td>
</tr>
<tr>
<td>8.2.5</td>
<td>References</td>
<td>65</td>
</tr>
<tr>
<td>9.</td>
<td>Presentation and Analysis of Effect of Turbulence Modelling</td>
<td>82</td>
</tr>
<tr>
<td>9.1</td>
<td>Mesh</td>
<td>82</td>
</tr>
<tr>
<td>9.2</td>
<td>Boundary conditions</td>
<td>82</td>
</tr>
<tr>
<td>9.3</td>
<td>Results</td>
<td>82</td>
</tr>
<tr>
<td>9.4</td>
<td>Discussion</td>
<td>83</td>
</tr>
<tr>
<td>10.</td>
<td>Discussion</td>
<td>95</td>
</tr>
<tr>
<td>10.1</td>
<td>General Comments</td>
<td>95</td>
</tr>
<tr>
<td>10.2</td>
<td>Interpretation of the CFD Code Comparison Cases</td>
<td>95</td>
</tr>
<tr>
<td>10.3</td>
<td>Effect of Mach Number</td>
<td>97</td>
</tr>
<tr>
<td>10.4</td>
<td>Effect of Sideslip</td>
<td>100</td>
</tr>
<tr>
<td>10.5</td>
<td>Effect of Reynolds Number</td>
<td>101</td>
</tr>
<tr>
<td>10.6</td>
<td>Effect of Turbulence Modelling</td>
<td>101</td>
</tr>
<tr>
<td>10.7</td>
<td>Thoughts on the Application of the Work</td>
<td>102</td>
</tr>
<tr>
<td>10.7.1</td>
<td>Use as an Intake Boundary Layer Diverter</td>
<td>102</td>
</tr>
<tr>
<td>10.7.2</td>
<td>Use in Combination with a Conventional Pitot-Type Boundary Layer Diverter</td>
<td>102</td>
</tr>
<tr>
<td>10.8</td>
<td>Collaborative Working</td>
<td>102</td>
</tr>
<tr>
<td>11.</td>
<td>Conclusions</td>
<td>104</td>
</tr>
<tr>
<td>Section</td>
<td>Title</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>11.1</td>
<td>General Conclusions</td>
<td>104</td>
</tr>
<tr>
<td>11.2</td>
<td>Bump Flow Physics</td>
<td>104</td>
</tr>
<tr>
<td>12.</td>
<td>Recommendations</td>
<td>104</td>
</tr>
<tr>
<td>13.</td>
<td>Notation</td>
<td>105</td>
</tr>
<tr>
<td>14.1</td>
<td>Design of the Bump</td>
<td>106</td>
</tr>
<tr>
<td>14.1.1</td>
<td>Geometries Studied</td>
<td>106</td>
</tr>
<tr>
<td>14.2</td>
<td>Performance of a Supersonic Inlet</td>
<td>107</td>
</tr>
<tr>
<td>14.3</td>
<td>Numerical Procedure</td>
<td>107</td>
</tr>
<tr>
<td>14.3.1</td>
<td>Results of the Calculations</td>
<td>107</td>
</tr>
<tr>
<td>14.4</td>
<td>Comparison of Performance</td>
<td>109</td>
</tr>
<tr>
<td>14.5</td>
<td>Conclusion</td>
<td>109</td>
</tr>
</tbody>
</table>
1. Introduction

1.1 Background

Action Group 34 ‘Aerodynamics of Supersonic Air Intakes’ was formed in early 1998. The Group consisted of members from the UK (BAE SYSTEMS and DERA), France (Aerospatiale and ONERA), Germany (Daimler Benz Aerospace) and the Netherlands (NLR). An earlier Exploratory Group (EG33) had shown that these nations had a common interest in improving their understanding of two classes of flow relevant to the design of supersonic intakes. The balance between Government and Industry organisations within AG34, together with interest in both aircraft and weapons applications, led to a healthy range of perspectives on the work required. An aspiration of the Group was that the work would find wide applicability through the resulting choices of cases and onset flow conditions. This report documents a CFD-based study of the aerodynamics of a class of ‘bump’ compression surface for aircraft or weapon intake applications. There is also some relevance to semi-buried intake installations for civil applications. Another study performed by AG34 on the control of shock-wave boundary-layer interaction in intakes through the use of active bleed was the subject of a separate report.

The Group decided not to adopt the traditional CFD study approach of selecting a set of experimental data for a particular flow and comparing the CFD results obtained by the participants. Instead, a theoretically derived bump shape was selected and each contributor computed a different part of a matrix of flow conditions. The emphasis was on characterising the flow and gaining an understanding of its main features. The reason for this was in part due to a lack of suitable data, but also because there was interest in exploring the feasibility of undertaking a CFD based investigation of a new flow collaboratively. An experimental investigation would still be available as a fall-back should this approach prove to be unsatisfactory.

The shaping of military air intakes can be heavily constrained by factors that are not related to aerodynamic performance. A recent trend towards a primary operational need for reduced radar signature has, for some types of vehicle, introduced wide-ranging shaping constraints. Synergistic intake concepts, that is those where the shape fits both sets of design rules reasonably well, are thus particularly sought after. The use of a ‘bump’-type compression surface as a means of providing a supersonic intake with external compression coupled with a degree of boundary layer diversion was first examined in the late 1950s [refs.1-1,1-2]. This work showed that an intake so equipped could potentially offer an improvement in installed thrust-minus-drag and reduced structural complexity compared to an intake with a conventional compression wedge and diverter. This concept was used in the US on the Grumman Super-Tiger aircraft but not subsequently, presumably due to risk through the high degree of three-dimensionality of the flow and consequent installation design difficulty for highly manoeuvrable aircraft. Renewed interest in this concept has been awakened by its potential for eliminating compression surface edges, the classical intake diverter gap and boundary layer bleed features, all of which can lead to difficulty in designing for low radar signature. The bump compression surface is thus a truly synergistic concept in that for an aircraft application it potentially offers improved aerodynamic performance, low radar signature and in addition the possibility of reduced structural complexity and weight. Even for a weapon application where the radar signature aspect may not be important, the concept may still be highly attractive on the basis of the other factors alone. Recent examples of the use of the bump intake concept are the Boeing and Lockheed Martin (LM) prototype Joint Strike Fighter aircraft (Figure 1.1-1).
An LM F-16 aircraft (Figure 1.1-2) was used as a test bed in the development of the LM JSF intake. This is a good indicator of the high level of technical risk attributed to the integration of this type of intake. Each of these intake designs features a highly forward-swept intake lip that will greatly influence the development of the flow on the bump surface. At both subsonic speeds and supersonic speeds, flow spillage will be concentrated into the lateral extremes of the bump surface. At subsonic speeds the lip will have a substantial forward influence and at supersonic speeds the influence will be primarily on the region aft of the intake normal shock.

A generic compression bump case was selected by AG34 for detailed study using CFD methods alone. A thorough understanding of the aerodynamics of such a bump was considered necessary before moving on to consider ways in which this feature might be used in a full intake design for either an aircraft or weapon.

### 1.2 Form of the Collaboration

Each organisation examined the effects of one or more parameters using its own CFD method, thereby spreading the workload. Further multiplication of the effort was obtained in the form of shared experience of problems encountered in the application of several different CFD codes to one problem. No calibration data existed for this case so it was a useful exercise in the ‘blind’ application of CFD methods. An element of the Study looked at code-to-code comparison, and this, together with engineering judgement, would be relied upon as the main way of testing the results, but the underlying aim was to see how well CFD methods alone could be used in such a collaboration to study unknown flows. It was considered that the flow should be attached everywhere on the bump and therefore
should be amenable to calculation with most Navier-Stokes codes. It was also thus considered unnecessary to carry out preliminary calculations on other geometries to validate the codes first. Depending upon the outcome of the work, the Group had the option to recommend that some experimental work should be done under a further Action Group. The purpose of this would be to fill gaps in the study where confidence in the CFD was poor or perhaps to verify a particularly controversial result. An experiment that replicated the precise geometry and conditions used here would be required to obtain absolute verification of the results and some care was taken to define a case where at least some elements could be readily examined in this way.

1.3 References

1-1 Bower, R E, et al. ‘Design and development of pre-compression bump surfaces for use with supersonic inlets’, Grumman Aircraft Engineering Corporation, Research Department Reports 122 and 129, 1959

2. **Objectives**

1. To study, as a team, the flow around a bump compression-surface at on and off-design conditions using CFD methods alone.

2. To improve the understanding of the physics of this class of flow. The emphasis was upon using, within their known limitations, the available CFD tools to improve the understanding of the aerodynamics of this Study Case as far as possible rather than on the testing of the CFD codes themselves.

3. To characterise the main flow features present across a range of flow conditions. The examination of several different variables created a need for a large number of calculations. The degree to which the bump thinned and/or diverted the approach boundary layer around where an intake entry would be located was the area of principal interest. The way in which this behaviour was affected by the onset Mach number and sideslip angle were also important factors.

4. Each contributing organisation carried out calculations using a different set of variables (onset flow conditions, turbulence model), though two conditions were calculated with all of the codes used. An underlying objective was therefore to test the viability of working in this way.

5. From the results of these studies, to decide whether experimental work was needed and, if it was, to define the experiments required.

### 2.1 Outline of the Study Case

![Sketch of bump compression surface](image)

*Fig 2.1-1 Sketch of bump compression surface*

The basic geometry of the case (Fig 2.1-1) was an exact stream surface of the inviscid, Mach 1.8 flow past a 23° cone at zero incidence and sideslip. The origin of the stream surface was a flat plane located at a fixed non-dimensional distance away from the axis of the cone. A matrix of on and off-design onset flow conditions was selected to represent typical wind tunnel model and flight scales for both aircraft and weapon applications. Mach number, sideslip and boundary layer thickness were the parameters varied. Different turbulence models were compared using one code at a limited number of conditions. All participants performed calculations at two standard conditions to allow confidence-building code-to-code comparisons to be made for each of the codes used.
2.2 Onset Flow Conditions and Allocation of the Calculations

2.2.1 Matrix

The higher end of the Mach number range was chosen to accommodate weapon applications. The high sideslip angles were chosen to reflect side intake mounting positions where aircraft incidence would appear as sideslip by the definition used here. The shaded areas indicate the main lines across the matrix on which calculations would be performed. Actual calculations performed are indicated.

\[
\begin{array}{c|c|c|c|c|c|c|c|c|c}
\hline
\beta /^\circ & 0 & 0.6 & 0.8 & 1.2 & 1.5 & 1.8 & 2 & 2.5 & 3 & 3.5 & 4 \\
\hline
0 & 0.03 & DA & BA & & & & & & & & \\
 & 0.07 & DE & ( ) & DE & \{\}^* & AS & AS & & & & \\
 & 0.12 & DA & BA & & & & & & & & \\
5 & 0.03 & & & & & & & & & & \\
 & 0.07 & & & DE & & & & & & & \\
 & 0.12 & & & & & & & & & & \\
10 & 0.03 & & & & & & & & & & \\
 & 0.07 & DE & & DE & AS & & & & & & \\
 & 0.12 & & & & & & & & & & \\
15 & 0.03 & & & & & & & & & & \\
 & 0.07 & & & & & & & & & & \\
 & 0.12 & & & & & & & & & & \\
20 & 0.03 & & & & & & & & & & \\
 & 0.07 & DE & & & AS & & & & & & \\
 & 0.12 & & & & & & & & & & \\
\hline
\end{array}
\]

\{\} = All to compute
( ) = All except AS to compute
* = Turbulence model study (BA)

\[
\begin{align*}
DE &= \text{DERA} \\
AS &= \text{Aerospatiale} \\
NL &= \text{NLR} \\
DA &= \text{DASA} \\
BA &= \text{BAe}
\end{align*}
\]

2.2.2 CFD Codes

Codes selected for use were as follows:

<table>
<thead>
<tr>
<th>Organisation</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale (Fr)</td>
<td>Matilda</td>
</tr>
<tr>
<td>DASA (Ge)</td>
<td>FLOWer</td>
</tr>
<tr>
<td>NLR (Ne)</td>
<td>ENSOLV</td>
</tr>
<tr>
<td>BAE SYSTEMS (UK)</td>
<td>FLUENT</td>
</tr>
<tr>
<td>DERA (UK)</td>
<td>SAUNA</td>
</tr>
</tbody>
</table>
2.2.3 Resources

Man months committed by each organisation were:

<table>
<thead>
<tr>
<th>Country</th>
<th>Months</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale (France)</td>
<td>3.1</td>
</tr>
<tr>
<td>DASA (Germany)</td>
<td>2.2</td>
</tr>
<tr>
<td>NLR</td>
<td>2.5</td>
</tr>
<tr>
<td>BAE (UK)</td>
<td>2.5</td>
</tr>
<tr>
<td>DERA (UK)</td>
<td>2.5</td>
</tr>
<tr>
<td><strong>Group Total</strong></td>
<td><strong>12.8</strong></td>
</tr>
</tbody>
</table>

Computer and computer hours committed by each organisation were as follows:

<table>
<thead>
<tr>
<th>Organisation</th>
<th>Computer(s)</th>
<th>Hours</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale (Fr)</td>
<td>SGI Origin 2000</td>
<td>400</td>
</tr>
<tr>
<td>DASA (Ge)</td>
<td>SGI Power Challenge</td>
<td>As required</td>
</tr>
<tr>
<td>NLR (Ne)</td>
<td>NEC SX-4</td>
<td>100</td>
</tr>
<tr>
<td>BAE (UK)</td>
<td>SGI Origin 2000</td>
<td>As required</td>
</tr>
<tr>
<td>DERA (UK)</td>
<td>CrayC90/SGI Octane</td>
<td>100 / As required</td>
</tr>
</tbody>
</table>
3. CFD Codes Used

3.1 Code Descriptions

The participants employed flow solvers to solve the Reynolds-Averaged Navier-Stokes equations on block-structured meshes as well as on unstructured meshes. The main differences that could affect the flow solutions concerned the selected turbulence models and artificial dissipation details. General information about the flow solvers can be found in the list of references.

<table>
<thead>
<tr>
<th>Participant</th>
<th>Acronym</th>
<th>Discretisation</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale</td>
<td>GASpex</td>
<td>Cell-centred upwind differencing</td>
<td>3-1</td>
</tr>
<tr>
<td>BAe</td>
<td>RAMPANT, FLUENT</td>
<td>Cell-vertex upwind differencing</td>
<td>3-2</td>
</tr>
<tr>
<td>DASA</td>
<td>FLOWer</td>
<td>Cell-vertex central differencing</td>
<td>3-3</td>
</tr>
<tr>
<td>DERA</td>
<td>SAUNA</td>
<td>Cell-centred central differencing</td>
<td>See below</td>
</tr>
<tr>
<td>NLR</td>
<td>ENSOLV</td>
<td>Cell-centred central differencing</td>
<td>3-4</td>
</tr>
</tbody>
</table>

Aerospatiale applied implicit Block Jacobi decomposition with inner iterations to advance the solution towards the steady state. The remainder of the participants applied the explicit Runge-Kutta time stepping scheme. Furthermore, each partner utilised convergence acceleration techniques like local time stepping, implicit residual smoothing and multi-grid. The turbulence models applied included the following 1 and 2-equation transport models:

<table>
<thead>
<tr>
<th>Participant</th>
<th>Turbulence model</th>
<th>Reference</th>
<th>Transition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale</td>
<td>k-ε (Chien)</td>
<td>3-5</td>
<td>Fully turbulent</td>
</tr>
<tr>
<td>BAe</td>
<td>k-ε and SA</td>
<td>3-6, 3-7</td>
<td>Fully turbulent</td>
</tr>
<tr>
<td>DASA</td>
<td>Baldwin-Lomax</td>
<td>3-8</td>
<td>Fully turbulent</td>
</tr>
<tr>
<td>DERA</td>
<td>k-ε (Chien)</td>
<td>3-5</td>
<td>Fully turbulent</td>
</tr>
<tr>
<td>NLR</td>
<td>k-ω (Wilcox)</td>
<td>3-9</td>
<td>Fixed transition</td>
</tr>
</tbody>
</table>

The references indicate whether the low or high Reynolds number variant is implemented. The following artificial dissipation models for stability were applied:

<table>
<thead>
<tr>
<th>Participant</th>
<th>Artificial dissipation model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aerospatiale</td>
<td>Flux-vector splitting (Van-Leer, Ref. 3-10)</td>
</tr>
<tr>
<td>BAe</td>
<td>Matrix dissipation (Roe)</td>
</tr>
</tbody>
</table>
SAUNA, (Structured And Unstructured Numerical Analysis), is a multiblock CFD code allowing solution of the Euler or Reynolds Averaged Navier-Stokes (RANS) equations. The algorithm employed in SAUNA is based on the work of Jameson. The time-dependent form of the governing equations is considered, in conservative form. These equations are numerically integrated to a steady state solution by balancing the mass, momenta and energy fluxes for control volumes within the flow field. The dependent variables for each control volume are stored and computed at each cell vertex in the flow field. The time integration is performed using an explicit Runge-Kutta scheme. In order to accelerate the convergence to a steady state, local time stepping and multigrid can be used.

For the RANS option, several turbulence models are available, including Baldwin-Lomax, Johnson-King, the 2-equation models, k-e and k-w, and the multiscale differential Reynolds stress model. Various options, including wall functions, Menter's shear stress limiter (SST) and non-linear eddy viscosity are available for some of these models.

References


3.2 The Calculation of Boundary Layer Displacement Thickness, ($\delta^*$)

For a detailed description of the method used to calculate boundary layer displacement thickness, the reader is referred to AIAA 92-0026, by D A Johnson (Non-equilibrium Algebraic Turbulence Modelling Considerations for Transonic Airfoils and Wings). This describes an approach suitable for the calculation of the location of the edge of the boundary layer ($\delta$). Having calculated $\delta$, this is used as
the upper limit in the usual integral definition of \( \delta^* \). The integral is evaluated numerically using quantities calculated through the boundary layer.
4. Geometry and Flow Conditions

4.1 Bump Geometry

4.1.1 Stream-Surface Definition

Conical flow is advantageous for slowing a supersonic flow ahead of an intake because of the element of isentropic compression involved, together with the structural advantage afforded by the inherently circular cross section of the required intake cowl. Conical flows can be produced directly by cones or indirectly by stream-surfaces derived from conical flows. The present bump is based on a stream-surface of the supersonic flow past a cone with a semi-angle, \( \alpha \), of 23 degrees. A design Mach number of 1.8 was chosen as this is within the ranges of interest of both the aircraft and the missile aerodynamicists contributing to this work. This cone and Mach number combination gives rise to a shock angle, \( \beta \), of 43.92 degrees.

Across the width of such a stream surface (span wise) from the centre line outwards, there will be a fall in pressure at the surface. This will give rise to a cross-flow component in the surface boundary layer which, for a suitably positioned intake, has the desirable effect of reducing the amount of boundary layer air that is ingested. Used in this way, the bump can be seen as a type of intake boundary layer diverter device, or 'bump diverter'.

We are interested in the basic performance of a bump diverter and therefore we wish to calculate the shape of a bump on a flat plate. It is therefore necessary to calculate the streamlines of the conical flow from points on the conical shock which also lie on a plane parallel to the cone axis (see fig 4.1-1). Because of the conical nature of the flow, the distance of the plane from the cone axis is irrelevant. Different plane positions will give rise to the same bump shape at different scales. For the calculation of the present stream surface, the plane was chosen to be a distance of 1.0 unit from the cone axis. Unit length in the bump geometry definition is therefore the distance of the plane from the cone axis in the derivation of the stream surface. The intersection of the plane with the shock is given by

\[
x^2 \tan^2 \beta - y^2 = h^2 \quad (4.1-1)
\]

where \( h \) is the height of the plane above the axis. (See fig 4.1-1). So in the present case, the starting points for the streamline calculations are given by

\[
0.92719x^2 - y^2 = 1.0 \quad (4.1-2)
\]

The present stream surface height is defined at a uniform array of points on the plane, 101 points in the stream-wise direction, between the cone apex (\( x = 0.0 \)) and the downstream limit (chosen arbitrarily as \( x = 4.0 \)), and 101 points in the span-wise direction, in the range -5.0 < \( y < 5.0 \). For points outside the cone shock the height above the plane is obviously zero, and for points within the cone shock, the method of reference 4-1 is used, which involves a hyperbolic approximation for the cone flow streamlines:

\[
r^2 = z^2 \sec^2 \theta = x^2 \tan^2 \alpha + c \quad (4.1-3)
\]

where \( c \) is constant for a particular streamline. We obtain \( c \) by combining equations 4.1-1 and 4.1-3 and noting that \( z = h \) and \( y = h \tan \theta \) on the plane. We then obtain:

\[
c = h^2 \sec^2 \theta \left( 1 - \tan^2 \alpha / \tan^2 \beta \right) \quad (4.1-4)
\]

Substituting in equation 4.1-3, we obtain:

\[
z^2 = x^2 \tan^2 \alpha \cos^2 \theta + h^2 \left( 1 - \tan^2 \alpha / \tan^2 \beta \right) \quad (4.1-5)
\]

The \( y \) value is given by

\[
y = z \tan \theta \quad (4.1-6)
\]
Different values of $\theta$ correspond to different members of the family of streamlines making up the stream surface. For the present case, equation 4.1-5 gives the following:

$$z^2 = 0.18018x^2 \cos^2 \theta + 0.80567 \quad (4.1-7)$$

For each combination of $x$ and $y$ from the uniform array mentioned above, equations 4.1-6 and 4.1-7 can be solved to give a corresponding value of $z$.

4.1.2 Rear Geometry

Having produced a definition of the stream-surface geometry, it was necessary to add further geometry at the rear, smoothly reducing the stream-wise slope at the end of the calculated stream surface to zero. The resultant cross section shape could then be extended downstream in a parallel fashion to the boundary of the computational domain.

The simplest way of adding such a smooth surface was to add a circular arc at each span-wise station which matched the position and slope at the end of the stream surface and had zero slope at a specified stream-wise location. These three conditions were sufficient to define the radius and centre of the required circular arc at each span-wise station.

As mentioned above, the end of the stream surface is at $x = 4.0$ and, also somewhat arbitrarily, the end of the circular arc section was chosen to be at $x = 5.0$. The stream-wise slope of the geometry at $x = 5.0$ is thus zero and the cross section shape at this location can be taken downstream as far as necessary for the CFD calculations.

4.1.3 Smoothing

The height of the bump is defined at a uniform array of points in the $x,y$ plane. If we consider the most down-stream station on the stream surface ($x = 4.0$ in this case), points outside the conical shock (i.e., off the bump) will have a height of zero and a stream-wise slope of zero will be used for the circular
arc extension. ie the circular arc will default to a straight line and the extension will continue at zero height. The first point inside the shock will be on the bump and have a small positive height and slope associated with it. The circular arc at this station will therefore give the geometry a further height increase beyond the end of the stream surface. This span-wise step in height onto the bump increases over the length of the circular arc and can give rise to an unwanted small geometric discontinuity in the span-wise sense.

In order to remove this discontinuity, the geometry was smoothed. The smoothing routine used works on radius of curvature and the user supplies it with a minimum radius. The routine calculates the radius of a circle through three consecutive points and if the radius is less than the specified minimum, the central point is shifted so that the three points lie on a circle of the specified radius. The routine then passes on to consider the position of the next point and so on. This method uses several passes until the radius is everywhere greater than or equal to the minimum specified. An advantage of this approach to smoothing is that the geometry is unaffected in areas that are already smooth. The geometry was smoothed in both the span-wise and stream-wise directions using a minimum radius of 2.0. This gave rise to the required removal of the discontinuity and a smoothing of the slope discontinuity at the forward edge of the bump.

4.1.4 Flat Plate

The bump is mounted on a flat plate which extends forward to x = -50.0. The plate provides a surface for the growth of a boundary layer, simulating an aircraft or missile forebody. The length of the plate ahead of the bump was chosen to be approximately fifty times the bump height (1.102), corresponding to, say, 5m of forebody ahead of a bump of height 0.1m on a full scale aircraft, or 2m of forebody ahead of a bump of height 0.04m on a missile. The length of the flat plate was varied to provide different boundary layer heights in the region of the bump (see paragraph 4.2.3 below).

4.1.5 Geometry for CFD Calculations

Having produced the geometry described above, 25.0 was added to all values of y and 1.0 subtracted from all values of z before distributing it to the participants. This was so that y had positive values in the area of interest and z was relative to the plane on which the bump was generated. The centreline of the bump is thus at y = 25.0 and the flat plate is at z = 0 for the remainder of this document.
4.2 Flow Conditions

4.2.1 Mach Numbers

Because the group consisted of both missile and aircraft aerodynamicists, the Mach number range of interest was quite large, from 0.6 to 4.0. The design Mach number of 1.8 was chosen as a datum as this was of interest to all contributors, being at the top end of the range for aircraft aerodynamics and at the bottom end for missiles.

Lower Mach numbers, (0.6, 0.8, 1.2, and 1.5), were chosen for the aircraft aerodynamicists to use for the investigation of the performance of the bump, and higher Mach numbers, (2.0, 2.5, 3.0, 3.5 and 4.0), were chosen for the missile aerodynamicists.

4.2.2 Cross-Flow Angles

Three Mach numbers, (0.8, 1.8, and 3.5), were chosen to study the effect of cross-flow angle. These were chosen to include the datum Mach number and one lower (subsonic) and one higher Mach number. The cross-flow angles chosen for M = 0.8 were 10 and 20 degrees, and those for M = 1.8 were 5 and 10 degrees. These correspond to typical ranges of aircraft incidence. For M = 3.5, the higher angles expected to be encountered by a missile were represented by cross-flow angles of 10 and 20 degrees.

4.2.3 Boundary-layer Thicknesses

It was agreed that calculations would be made for three values of boundary-layer displacement thickness over bump height, \( \frac{\delta}{k} \). A full scale aircraft with a 0.1m bump height and a 5m forebody, flying at 11Km altitude at Mach 0.8 gives rise to a value of \( \frac{\delta}{k} \) of 0.074 using flat plate boundary layer theory (ref. 4-2):

\[
\frac{\delta}{\ell} = 0.37 R^{-0.2} \text{ and } \frac{\delta}{\ell} = 1/8
\]  (4.2-1)

lead to

\[
\frac{\delta}{k} = 0.0463 \left( \frac{\ell}{k} \right) R^{-0.2}
\]  (4.2-2)

Using

\[ R = \left( \frac{\ell}{k} \right) R_k \]  (4.2-3)

gives

\[
\frac{\delta}{k} = 0.0463 \left( \frac{\ell}{k} \right)^{0.8} R_k^{-0.2}
\]  (4.2-4)

The corresponding value at a Mach number of 1.8 is 0.057. Nominal values of \( \frac{\delta}{k} \) of 0.03, 0.07, and 0.12 were chosen to encompass this notional case, and provide boundary layers of roughly half and double the height. The most appropriate way to achieve different boundary layer heights was to vary the length of the flat plate ahead of the bump, and maintain the same Reynolds number based on bump height, \( R_k \), of 7.87e05, which was derived from the flight conditions mentioned above.

Using a value of \( k \) of 1.102 units (the height of the bump at \( x>5 \)), plate lengths were calculated from equation 4.2-4. The table below shows the plate lengths used and the corresponding values of \( \frac{\delta}{k} \).

<table>
<thead>
<tr>
<th>( \ell )</th>
<th>( \ell/k )</th>
<th>( \delta/k )</th>
</tr>
</thead>
<tbody>
<tr>
<td>17.0</td>
<td>15.43</td>
<td>0.028</td>
</tr>
<tr>
<td>50.0</td>
<td>45.37</td>
<td>0.067</td>
</tr>
<tr>
<td>98.0</td>
<td>88.93</td>
<td>0.115</td>
</tr>
</tbody>
</table>
4.3 References

4-1 Seddon, J and Goldsmith, E L.

4-2 Duncan W J, Thom A S and Young A D
5. Results of CFD Code Comparison

Two cases were chosen as comparisons for all the contributors to analyse. The intention was to gain confidence in the different codes’ abilities to model the flow correctly. While comparing results, it also allowed the contributors to learn from each other the best way to grid and set up boundary conditions for the various cases. Several of the contributors undertook mesh refinement studies at the start of the collaboration such that consensus on appropriate mesh characteristics was reached early in the programme.

The main case chosen was \( M=1.8, \delta^*/k=0.07, \beta=0 \), with a subsidiary off-design case of \( M=0.8, \delta^*/k=0.07, \beta=0 \). Aerospatiale did not compute the subsidiary case as their interest was in much higher Mach numbers. The parameters compared were \( C_p, C_f \) and \( \delta^* \). Profiles of these parameters have been prepared at values of \( y=25 \) (centreline), and \( x=2, 4 \) & \( 6 \). For the constant \( x \) plots the data has been plotted assuming symmetry on the bump, so only data greater than \( y=25 \) is shown.

5.1 \( M=1.8 \delta^*/k=0.07 \)

5.1.1 Centreline Data

The following plots show comparisons for the data from the various participants. Figure 5.1-1 shows the basic \( C_p \) along the centreline in the region of the bump. The shape of the traces matches very well for all the codes.

![Figure 5.1-1](image)

Zooming in closer to the bump, as in Figure 5.1-2 the detailed differences in the codes can be seen more clearly. No centreline data was received from NLR, although it is available for the lateral cuts.
Looking at friction coefficient (Figure 5.1- 3) the previous good agreement between codes is not so apparent. The codes show different positions and values for their maxima and minima. The Aerospatiale and DASA data shows the flow very nearly separating, with $C_f$ very close to 0.
The plot of $\delta^*$ (Figure 5.1-4) has data from three participants. This reflects the difficulty in actually producing values of $\delta^*$. The DERA data shows a distinct decrease in boundary layer thickness along the bump. The BAe data appears to show similar levels of value to the DERA data, which is about all that can be said. The discontinuity in the DERA data at $x=5$, and probably the discontinuities in the DASA data, are explained in section 6.1.3.

![Graph of $\delta^*$ vs x (y=25)](image)

**Figure 5.1-4**

5.1.2 Lateral Cuts at Constant X

Figure 5.1-5 to Figure 5.1-7 show values of $C_p$ at $x=2$, 4 and 6 along the bump. All the codes show similar characteristics with the BAe, DASA and NLR data showing very close agreement.

![Graph of $C_p$ vs y (x=2)](image)

**Figure 5.1-5 to Figure 5.1-7**
The Cf data (Figure 5.1- 8) is shown only at x=4, as this shows similar differences to those at x=2 and 6. The Aerospatiale data reaches its minimum at the same location as the other four, but the maximum levels on the bump are lower than on the flat plate, which is different to the other solutions.
The $\delta^*$ values (Figure 5.1-9) are only available at $x=4$ from NLR and BAe. The BAe, DASA and NLR data agree extremely well, while the DERA data follows a very similar pattern, but appears to be offset from the other three.
5.2 $M=0.8 \, \delta^*/k=0.7$

5.2.1 Centreline Data

The centreline values of $C_p$, $C_f$ and, where available, $\delta^*$, are shown.

**Figure 5.2-1**

**Figure 5.2-2**
5.2.2 Lateral Cuts at Constant X
The same selection of plots is shown as for the M1.8 data. The BAe and NLR data can again be seen to be very similar to each other. The DERA data follows very similar trends, but is slightly offset.
The Cf data (Figure 5.2- 7) shows more difference between the solutions, with the NLR code giving higher values on the bump, although similar on the plate.
No BAe $\delta^*$ values were not available, so the comparison is between NLR, DASA and DERA (two different grids). The NLR code gave considerably lower values on the bump (up to half the DERA values), but the difference was fairly constant all the way out to the free stream. All codes gave very similar patterns, including the small bump at $y=28.75$. The flow here is in an area of high stream-wise pressure gradient, thus accentuating any small differences between the codes. The discontinuity in the NLR data is unexplained.
5.3 Discussion

In general the Cp data supplied by the participants shows a good level of agreement. The poorest agreement seems to come from Aerospatiale at $M=1.8$, particularly looking at the lateral cuts, e.g. figure 5.1-6. This was puzzling, but may be resolved by studying section 8.2.3, the effect of Reynolds number, figure 8.2-18. This shows similar deviations for the different flat plate lengths, and with all the data crossing at a constant value at $y=29.2$. It could well be that the Aerospatiale solution was set up with an incorrect value of Reynolds number.

The Cf data shows more variation, but this parameter is notoriously sensitive to grid quality.
6. Presentation and Analysis of Effect of Mach Number

6.1 Low Mach Range (M = 0.6 – 1.8)

These RANS calculations were carried out by John Hodges at DERA Bedford, using the SAUNA CFD code with the k-ε turbulence model and wall functions.

6.1.1 Mesh Used

Flow symmetry was assumed so that only half the flow field needed to be modelled. A structured multi-block mesh consisting of 30 blocks (5 x 2 x 3) and 614400 cells (160 x 80 x 48) was used.

Because of instabilities arising in SAUNA when a boundary layer starts at the upstream boundary of the flow domain, one layer of mesh blocks was used ahead of the flat plate (-100 < x < -50). The other four layers of blocks in the x-direction were used as follows: One block was used on the flat plate (-50 < x < 0), two blocks were used in the bump region (0 < x < 5) and one block was used downstream of the bump (5 < x < 50). In the y-direction, one block layer was used in the bump region (25 < y < 30), and one layer outboard (30 < y < 65).

The Navier-Stokes mesh was created with an initial cell size of 0.0025 units adjacent to solid surfaces to provide a suitable distribution of y+. Ahead of the flat plate, the closely spaced mesh was splayed out in the z-direction.

6.1.2 Boundary Conditions

At the upstream boundary of the flow domain, a Riemann invariant boundary condition was used. At the downstream boundary, the Rudy-Strikwerda condition was used. This drives the static pressure to its free stream value for subsonic flows and all other variables are extrapolated from the interior. For supersonic flows, all variables are extrapolated.

As mentioned above, flow symmetry was assumed on the centreline of the configuration (y = 25) and so flow tangency was applied there. On the opposite side of the flow domain, (y = 65), the Rudy-Strikwerda condition was used. On the lower boundary, the no slip boundary condition was applied to the solid surfaces, while upstream of the flat plate, flow tangency was applied. On the upper boundary of the flow domain, the Riemann invariant boundary condition was used.

6.1.3 Results

Converged flow solutions were obtained at Mach numbers of 0.6, 0.8, 1.2, 1.5 and, after some initial convergence difficulties, 1.8. Figure 6.1-1 shows the distribution of y+ along the centreline of the configuration for the various Mach numbers. Because the same grid and Reynolds number were used for all the Mach numbers, there is a spread in the values of y+, but they are centred around 50, and therefore considered adequate for the k-ε turbulence model with wall functions.

Figure 6.1-2 shows the effect of Mach number on pressure coefficient at x=4, the nominal location of an intake entry plane. At the centreline, positive values of Cp are indicated for all the supersonic free streams, showing that the bump is working as a compression surface. However, in the subsonic case, negative values of Cp are seen, presumably because the circular arc closure downstream of x=4 is having an upstream effect and accelerating the flow here.

The subsonic calculations show a small pressure “hump” near y=28.5 which is due to the upstream effect of insufficient smoothing being applied at the edge of the closure (see section 4.1.3).

The effect of the circular arc closure is seen clearly in Figure 6.1-3, which shows the Cp distributions along the centreline. For the subsonic free stream, positive values of Cp are seen ahead of and on the initial part of the bump, but a large negative Cp peak is observed near x=4.5 on the circular arc section.

At the design condition, M=1.8, the pressure rises through an attached shock near x=1 and then continues to rise isentropically as far as about x=4. This shows that the main features of the conical flow, on which the bump was designed, have been reproduced. At a free-stream Mach number of 1.5,
the behaviour is very similar. At M=1.2, there is a pressure rise ahead of the bump, indicating a detached shock.

Figure 6.1-4 shows the effect of Mach number on the boundary layer displacement thickness, $\delta^*$, along the centreline of the configuration. There are some discontinuities in the calculated values of $\delta^*$ in the region 5<x<8 which are due to anomalies in the method for locating the edge of the boundary layer. This region is not of great interest as it is downstream of the nominal location of an intake entry plane.

Also shown is the theoretical flat-plate boundary-layer thickness given by ref. 6.1-1:

$$\delta^*/x = 0.046 R_x^{-0.2}$$

6.1-1

This equation assumes incompressible flow, which explains why the theoretical boundary layer growth does not coincide with the SAUNA predictions of boundary layer thickness in the area ahead of the bump. The discrepancy, which increases with Mach number, is an indication of the effect of compressibility on the growth of the boundary layer. This was confirmed by a separate incompressible calculation of the flat plate boundary layer using SAUNA.

Figure 6.1-5 shows the effect of Mach number on boundary layer displacement thickness at x=4, the nominal intake entry plane. Table 6.1-1 summarizes the predicted values on the centreline at x=3.5 and 4 (denoted by $\delta^*_c$) and quantifies the reduction in boundary layer thickness from the undisturbed flat-plate predictions at x=0. The values of thickness used for the undisturbed boundary-layer, (denoted by $\delta^*_u$), were those at (x,y) = (0,45), which is sufficiently far from the centreline to be unaffected by the bump.

<table>
<thead>
<tr>
<th>M</th>
<th>$\delta^*_u$</th>
<th>$\delta^*_b$</th>
<th>$1.0 - \delta^<em>_u/\delta^</em>_b$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3.5</td>
<td>4.0</td>
<td>3.5</td>
</tr>
<tr>
<td>0.6</td>
<td>0.097</td>
<td>0.059</td>
<td>0.392</td>
</tr>
<tr>
<td>0.8</td>
<td>0.103</td>
<td>0.062</td>
<td>0.398</td>
</tr>
<tr>
<td>1.2</td>
<td>0.114</td>
<td>0.095</td>
<td>0.167</td>
</tr>
<tr>
<td>1.5</td>
<td>0.118</td>
<td>0.099</td>
<td>0.161</td>
</tr>
<tr>
<td>1.8</td>
<td>0.151</td>
<td>0.122</td>
<td>0.192</td>
</tr>
</tbody>
</table>

Table 6.1-1 Effect of M on $\delta^*$ at x=3.5 and 4.0

It is seen that as Mach number is increased, the bump causes a lower fraction of the boundary layer to be removed. However, when used in conjunction with a forward swept intake, it is expected that higher spillage at supersonic speeds will tend to flush more of the boundary layer away. Also, the thinning of the boundary layer at this location at subsonic speeds has probably been exaggerated by the local flow acceleration due to the upstream effect of the circular arc closure mentioned above.

Figure 6.1-5 shows that the reduction of boundary layer thickness at the centreline is achieved by redistributing the viscous layer and increasing the boundary layer thickness near the edge of the bump (y=28.7). The span-wise location of the maximum thickness of the boundary layer is almost insensitive to Mach number. At Mach numbers below 1.2 the maximum thickness occurs at y=28.7, but at higher Mach numbers it is slightly nearer the centreline. (eg at M=1.8, y=28.0). If a forward swept intake is positioned near x=4 with a span-wise extent roughly equal to the width of the bump, it is expected that the position of the thickened boundary layer is such that flow spillage will prevent it being ingested.

Figure 6.1-6 shows a comparison of the results of two methods for calculating the boundary layer displacement thickness, $\delta^*$, in the same flow field at M=1.8. Both methods use the same approach for the integration, but differ in the derivation of the upper limit for the integration, i.e. the boundary layer edge location.
Figure 6.1-4 and Figure 6.1-5, based on moment of vorticity, (ref 6.1-2), is compared to a method programmed at ARA, (ref 6.1-3), based on reduction in turbulence viscosity.

In general, the results are very similar. However, the discontinuities produced by the method of ref 5.2-2 and referred to above are not present in the results of ref 6.1-3. The results of ref 6.1-3 appear to be less peaky, but also less smooth than those of the method of ref 6.1-2.

The effect of Mach number on the effectiveness of the bump is summarised in Figure 6.1-7, which shows a plot of \(1.0 - \frac{\delta^* b}{\delta^* u}\) against Mach number.

### 6.1.4 Flow Physics

The bump redistributes the boundary layer so that the thickness increases at the edges of the bump and reduces on the bump itself. By suitable positioning of an intake, spillage will prevent the thicker layer at the sides of the bump from being ingested and thus the bump can be used as a boundary layer diverter.

It appears that the bump is more effective at reducing the boundary layer thickness at the lower Mach numbers, but this effect may have been exaggerated by the upstream effect of the circular arc closure of the bump.

The main features of the conical flow on which the design of the bump was based have been reproduced at the design Mach number of 1.8 and at a Mach number of 1.5.

### 6.1.5 References:


6-2 Johnson, D A, ‘Nonequilibrium Algebraic Turbulence Modelling Considerations for Transonic Airfoils and Wings’, AIAA 92-0026

6-3 May, N E, Subroutine DSTAR – private communication.
Figure 6.1- 1

Figure 6.1- 2
Figure 6.1-3

$C_p \times \gamma = 25$

$\delta^* \times x$ on centreline ($\gamma = 25$)
Figure 6.1- 4
Figure 6.1- 5
Comparison of $\delta^*$ methods  ($M=1.8$ centreline)

Figure 6.1-6
\( \delta^* \) reduction v M

\[
(1 - \delta^* b/\delta^*)
\]

Figure 6.1- 7
6.2 High Mach Range (M = 1.8 to 3.5)

The following calculations use the Matilda code described in section 3. The mesh is shown in the next figure: its size is about 250,000 nodes. A free-stream and a supersonic outflow (first order extrapolation) conditions are used at the boundaries of the domain. The bump wall is considered as adiabatic.

The inviscid fluxes are computed with a Roe type scheme associated with an isentropic correction (Harten) and a Van Leer limiter. The turbulence model is k-L with laws of the wall.

Let us define the deviation rate as $1.0 - \frac{\delta^*}{\delta_{u}^*}$, the ratio between the boundary layer height at the top of the bump (here defined as $x=6$) and the boundary layer height before the bump ($x=0$). The following table presents the several boundary layer heights and deviation rates for each Mach number.

<table>
<thead>
<tr>
<th>Mach</th>
<th>Before Bump ($10^4$ m)</th>
<th>After Bump ($10^4$ m)</th>
<th>Deviation Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.8</td>
<td>220</td>
<td>50</td>
<td>77 %</td>
</tr>
<tr>
<td>2.5</td>
<td>210</td>
<td>55</td>
<td>74 %</td>
</tr>
<tr>
<td>3.5</td>
<td>200</td>
<td>60</td>
<td>70 %</td>
</tr>
</tbody>
</table>

The highest deviation rate is obtained for the design Mach number of the bump, that is $M=1.8$. However, the loss of performance, even at $M=3.5$, is very weak.

The following figure shows in each case the non-dimensional Mach numbers, that is the fields of the ratio:
These fields are plotted on a front view slice located at the end of the bump.

Figure 6.2.2: Comparison of non-dimensional Mach contours for Mach 1.8, 2.5 and 3.5 (with no sideslip angle)

These views clearly show the effect of the Mach number on both angles and sizes of the conical shock wave it creates. The effect on the height of the boundary layer, with a light loss of performance with an increasing Mach number can also be seen.

The three next figures present the values of $C_p$ at the top of the bump in the transverse direction. The shock also appears to be less wide with an increasing Mach number.
Figure 6.2.3: Evolution of Cp value with Mach number
7. Presentation and Analysis of Effect of Sideslip

7.1 Subsonic (M=0.8)

These RANS calculations were carried out at DERA Bedford, using the SAUNA CFD code with the $k-\varepsilon$ turbulence model and wall functions.

7.1.1 Mesh Used

The half flow field calculation assuming flow symmetry, (see section 6.1), was used for the zero sideslip case. For the non zero sideslip cases (Beta = 10, 20 degrees), a mirror image was added to the mesh for $y<25$. Also, for the same reason as outlined in section 6.1.1, a further layer of blocks was added on the windward side to provide some free air between the boundary of the domain and the edge of the plate. The number of cells in this mesh was 1257984.

7.1.2 Boundary Conditions

At the upstream boundary of the flow domain and on the inflow sidewall, a Riemann invariant boundary condition was used. At the downstream boundary and on the outflow sidewall, the Rudy-Strikwerda condition, (see section 6.1), was used.

On the lower boundary, the no slip boundary condition is applied to the solid surfaces, while upstream of the flat plate, flow tangency is applied. On the upper boundary of the flow domain, the Riemann invariant boundary condition was used.

7.1.3 Results

At $x=2$, the $C_p$ distribution shows that as sideslip is increased, the pressure increases on the windward side of the bump and decreases on the leeward side so that the pressure peak moves towards the windward (Figure 7.1-1). At a sideslip of 20 degrees, a suction peak appears on the leeward side of the bump.

The effect of sideslip on the boundary layer is seen in Figure 7.1-2 as an increase in the $\delta^*$ peak on the windward side of the bump and a decrease in the peak on the leeward side.

At $x=4$, (Figure 7.1-3), the changes in the $C_p$ distribution are in similar directions to those at $x=2$, but the magnitudes of the changes are greater. The negative $C_p$ peak on the windward side disappears and becomes a positive $C_p$ peak with increasing sideslip, while the suction peak to leeward increases.

The pressure “humps” near $y=21.5$ and $y=28.5$ are caused by the upstream influence of insufficient smoothing at the edge of the circular arc closure (described in section 4.1.3).

Interestingly, this increased suction on the leeward side does not give rise to a reduction in $\delta^*$ as at $x=2$. On the contrary, with the advent of 10 degrees of sideslip, $\delta^*$ increases on both windward and leeward sides (Figure 7.1-3).

Figure 7.1-4). However, a further increase of sideslip to 20 degrees hardly changes $\delta^*$ on the leeward side.

On the centreline, the $C_p$ distribution, (Figure 7.1-5), shows that generally, the pressure on the bump reduces with sideslip, except in the area between $x=4$ and $x=5$, where the suction peak is reduced slightly.

The effect of sideslip on $\delta^*$ along the centreline is shown in Figure 7.1-6. It is interesting to note that on most of the conical stream-surface part of the bump, (0.5<$x$<4.0), $\delta^*$ is almost unaffected by sideslip. However, there is a small increase in $\delta^*$ due to sideslip ahead of the...
bump and a huge increase downstream of $x=5$. It is the upstream effect of this huge increase which produced the unexpected results at $x=4$ mentioned above. Plots of $C_f$ do not indicate any flow separation and the cause of this large effect is not yet known.

There is a small area near $x=5$ where $\delta^*$ appears to be discontinuous. This is almost certainly caused by an anomalous location being found in the routine for finding the edge of the boundary layer.
Figure 7.1-1

Figure 7.1-2
Figure 7.1-3

Figure 7.1-4
Figure 7.1-5

Cp v x on centreline (y=25)

Figure 7.1-6

δ* v x on centreline (y=25)
7.2 Effect of Sideslip (M = 1.8)

These RANS calculations were intended to be carried out by NLR, but because of a resource issue, they were eventually performed by John Hodges at DERA Bedford, using the SAUNA CFD code with the k-e turbulence model and wall functions.

7.2.1 Mesh and Boundary Conditions

The same mesh and boundary conditions were used as those for the M=0.8 calculations (see sections 7.1.1 and 7.1.2).

7.2.2 Results

As at M=0.8, the x=2 pressure distribution shows the pressure peak moving to windward as the sideslip increases. Unlike the M=0.8 case, there are no suction (negative Cp values) on the leeward side when sideslip is introduced.

At x=4, the trends with sideslip are similar to those at x=2, but slightly greater. Unlike M=0.8, there is no large suction peak.

The y=25 (centreline) pressure distribution shows only a small drop in pressure in the isentropic recompression region (1<x<4) with increasing sideslip.

As at M=0.8, sideslip increases the boundary layer thickness on the bump at x=4, both at the maxima, and at the minimum on the centreline. However, the results at 5 and 10 degrees sideslip are very similar. The effect at x=2 appears much smaller.

On the centreline, the boundary layer thickness is very similar for 5 and 10 degrees sideslip, but they are very different from the zero sideslip case. Compared to the zero sideslip case, there is a much more dramatic thinning of the layer near x=1, albeit from a higher value at x=0. Also the thinning achieved in the zero sideslip case near x=4 reduced with sideslip. The shapes of the boundary layer thickness curves are not yet fully understood.
Figure 7.2-1; Effect of sideslip angle on surface static pressure at a cross flow plane (x = 2)

Figure 7.2-2; Effect of sideslip angle on boundary layer displacement thickness at a cross flow plane (x = 2)
Figure 7.2-3; Effect of sideslip angle on surface static pressure at a cross flow plane (x = 4)

Figure 7.2-4; Effect of sideslip angle on boundary layer displacement thickness at a cross flow plane (x = 4)
Figure 7.2-5; Effect of sideslip angle on surface static pressure at the symmetry plane

Figure 7.2-6; Effect of sideslip angle on boundary layer displacement thickness at the symmetry plane
7.3 Mach=3.5

The following calculations use the Matilda code described in section 3. The mesh size is about 250,000 nodes. Free-stream conditions and supersonic outflow (first order extrapolation) are used at the boundaries of the domain. The bump wall is treated as adiabatic.

The boundary layer heights before and after the bump are presented in the following table:

<table>
<thead>
<tr>
<th>Sideslip Angle (deg.)</th>
<th>Before Bump (10^{-4} m)</th>
<th>After Bump (10^{-4} m)</th>
<th>Deviation Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>200</td>
<td>60</td>
<td>70 %</td>
</tr>
<tr>
<td>10</td>
<td>220</td>
<td>65</td>
<td>70 %</td>
</tr>
<tr>
<td>20</td>
<td>230</td>
<td>70</td>
<td>69.5 %</td>
</tr>
</tbody>
</table>

The main effect is that, at this Mach number, the deviation rate seems to be independent of the sideslip angle.

However, the sideslip angle introduces asymmetry. The following figure shows the Mach number fields for each side-slip angle on a slice located at the end of the bump (in a front view). The boundary layer height on the side facing the wind is increased, since more compression occurs, whereas the opposite effect is present on the other side. The deformation of the conical shock wave and its displacement are also clearly shown on this view.

Figure 7.3.1: Comparison of Mach contours for sideslip angles 0, 10 and 20 degrees (M=3.5)

The following curves are the velocity profiles on the symmetry line at the top of the bump for the three computed sideslip angles. They are compared with the velocity profile before the bump. The character of the boundary layer appears to be almost independent of the sideslip angle.
Figure 7.3.2: influence of sideslip angle on the velocity profiles
8. Presentation and Analysis of Effect of Reynolds Number (Flat Plate Length)

8.1 Subsonic (M=0.8)

These calculations were performed with the FLOWer code using the Baldwin-Lomax turbulence model. The flow for three different lengths of the flat plate upstream of the bump has been calculated at the fixed Mach number $Ma = 0.8$.

8.1.1 Mesh Used

A structured multiblock mesh consisting of 12 blocks was used for all configurations. Grids for three different configurations have been constructed with the grid generation tool ICEM-CFD-Hexa (Ref. [8.1]). The difference between these three configurations lies in the length of the flat plate upstream of the bump. The flat plate lengths were 1700 mm, 5000 mm, and 9800 mm. (These numbers have been used in the name for the configurations: bump1105-1700, bump1202-5000, and bump1004-9800). The numbers of grid points were 3.3, 3.5 and 4.0 million for the full bump.

The boundaries of the blocks can be seen in Figure 8.1-1 for one of the flat plate lengths (bump1202-5000). One side of all blocks represented the solid walls of the flat plate and/or bump. In order to minimise numerical leading edge disturbances the blocks on the entrance side of the flow regime have been extended upstream of the tip of the flat plate (without extension of the solid body boundary conditions) by 25 meshpoints. The cell heights adjacent to solid surfaces varied between 0.003 mm and 0.005 mm.

8.1.2 Boundary Conditions

At the upstream boundary of the flow domain the free-stream conditions were set and held fixed for all calculations. At the side-walls and top-walls of the flow domain the static pressure of the free-stream was prescribed and held fixed. At the exit of the flow domain the static free-stream pressure has been fixed for subsonic Mach number calculations whereas for supersonic Mach numbers zero order extrapolation of the respective flow variables has been used. On the solid walls of the lower domain boundary no slip boundary conditions were applied, while upstream of the flat plate leading edge flow symmetry (i.e. inviscid flow tangency) conditions ensured disturbance free on-set flow of the plate.

8.1.3 Results

The $C_p$ distribution at $Ma = 0.8$ along the solid walls in the centre plane is shown in Figure 8.1-2 for all plate lengths. All calculations show a $C_p$ peak just on the bump and a $C_p$ minimum just downstream of the bump. The magnitude at the $C_p$ peak is highest for the shortest flat plate, i.e. for the thinnest boundary layer approaching the bump. For the other flat plates the peak values are reduced due to smearing of the bump shape by the boundary layers. For the same reason the minimum $C_p$ value just downstream of the bump is smallest for the shortest flat plate and highest for the longest flat plate. Figures 8.1-3 to 8.1-6 depict the $C_p$ distribution versus $y$ at different $x$-stations along the bump ($x=0.0$ is at the start of the bump). It is interesting to note that at $x = 2.0$ the $C_p$ graphs show a sudden change in slope at $y\approx 23.3 / 26.7$. At the same stations the $C_p$ graphs at $x = 4.0$ (Figure 8.1-5) have a minimum (and a higher value on the centreline.) The difference between the $C_p$ of the bump1105-1700 flat plate and the others at $x = 6.0$ (Figure 8.1-6) in this region is due to a separation on the bump for the bump1105-1700 flat plate configuration (Figure 8.1-18). The influence on $C_p$ due to the insufficient smoothing at the edge of the bump closure (see section 4.1.3) can only be recognised in the expansion area, i.e. at $x = 4.0$ (Figure 8.1-5).

Figure 8.1-7 shows the effect of plate length on the boundary layer displacement thickness $\delta^*$ along the centreline of the configurations. These graphs are not smooth everywhere. This is due to the method applied to calculate this variable and does not give any indication concerning
the quality of the flow calculations. The increase in $\delta^*$ in front of the bump is largest for the longest flat plate and smallest for the shortest flat plate.

The displacement thickness is plotted versus $y$ for different $x$-stations in Figures 8.1-8 to 8.1-11. It can be clearly recognized that boundary layer air is removed from the centreline to the side of the bump. The $y$-station where this boundary layer air is collected is independent of the boundary layer displacement thickness, i.e. flat plate length. It probably depends on the geometry of the bump extension only. The largest reduction in displacement thickness is gained by the longest flat plate (Figure 8.1-10). To visualize this effect the same variable $(1.0 - \delta^*_b/\delta^*_u)$ as defined in Chapter 6.1 has been calculated and is given in Table 8.1-1. These data have also been plotted in Figure 8.1-12. The values of the undisturbed displacement thickness $\delta^*_u$ were those taken from a point $x = 0.0$ and $y = 45.0$. With increasing length of the flat plate the bump becomes slightly less effective in reducing boundary layer thickness. At station $x = 4.0$ its effectiveness is somewhat superior to that at $x = 3.5$ (Figure 8.1-12).

<table>
<thead>
<tr>
<th>Ma = 0.8</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Plate length</th>
<th>$\delta^*_u$</th>
<th>$\delta^*_b$</th>
<th>$1.0 - \delta^<em>_b/\delta^</em>_u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>x=3.5</td>
<td>x=4.0</td>
<td>x=3.5</td>
<td>x=4.0</td>
</tr>
<tr>
<td>1700</td>
<td>0.03393</td>
<td>0.01800</td>
<td>0.01121</td>
</tr>
<tr>
<td>5000</td>
<td>0.08925</td>
<td>0.04913</td>
<td>0.02928</td>
</tr>
<tr>
<td>9800</td>
<td>0.15121</td>
<td>0.09072</td>
<td>0.05751</td>
</tr>
</tbody>
</table>

Table 8.1-1 Effect of flat plate length on $\delta^*$ at $x = 3.5$ and $x = 4.0$

Friction coefficients were plotted in Figures 8.1-13 to 8.1-17. Figure 8.1-13 shows that there is a reduction in friction in the deceleration region at the “tip” of the bump. The friction increases again in the acceleration region, reaches a peak and drops down to another minimum. The maximum value and the second minimum value of the configuration with the shortest flat plate (bump1105-1700) are highest and lowest, respectively. The shape of the cfinf graphs at different $x$-stations (Figures 8.1-14 to 8.1-17) are similar to the Cp graphs. I.e., in the area in which boundary layer air is accumulated there exist maxima or minima in friction coefficient. These figures also depict that the magnitudes in friction of the two configurations with the longest plates are very close.

8.1.4 References

Figure 8.1-1 Block boundaries of bump1202-5000

Ma = 0.8
Cp versus x at y = 25

Figure 8.1-2
Figure 8.1-3

Figure 8.1-4
Ma = 0.8
Cp versus y at x = 4.0

Figure 8.1-5

Ma = 0.8
Cp versus y at x = 6.0

Figure 8.1-6
**Figure 8.1-7**

Ma = 0.8

$\delta^*$ versus x at y = 25

**Figure 8.1-8**

Ma = 0.8

$\delta^*$ versus y at x = 0.0

- 52 -
Figure 8.1-9

Ma = 0.8
\( \delta^* \) versus \( y \) at \( x = 2.0 \)

Figure 8.1-10

Ma = 0.8
\( \delta^* \) versus \( y \) at \( x = 4.0 \)
Ma = 0.8

\( \delta^* \) versus \( y \) at \( x = 6.0 \)

Figure 8.1-11

MA = 0.8

\( \delta^* \) reduction versus plate length

Figure 8.1-12
Ma = 0.8
cfinf versus x at y = 25

Figure 8.1-13

Ma = 0.8
cfinf versus y at x = 0.0

Figure 8.1-14

- 55 -
Ma = 0.8

$c_{\infty}$ versus $y$ at $x = 2.0$

Figure 8.1-15

Ma = 0.8

$c_{\infty}$ versus $y$ at $x = 4.0$

Figure 8.1-16

- 56 -
Ma = 0.8
cfinf versus y at x = 6.0

Figure 8.1-17
Figure 8.1-18 $C_p$ and surface streamlines on bump1105-1700
8.2 Supersonic (M=1.8)

8.2.1 Meshes

Details of the meshes used in the study of the effect of flat plate length are listed in Table 8.2-1. In all, three meshes were used.

<table>
<thead>
<tr>
<th>Description</th>
<th>Flat plate length</th>
<th>First cell height</th>
<th>Total number of cells</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Datum mesh</td>
<td>50</td>
<td>0.01</td>
<td>194,266</td>
<td>Datum geometry computed by all AG34 members to provide a measure of the consistency achieved across the group.</td>
</tr>
<tr>
<td>Short mesh</td>
<td>17</td>
<td>0.005</td>
<td>244,644</td>
<td>With datum and long meshes, used to investigate the effect of flat plate length</td>
</tr>
<tr>
<td>Long mesh</td>
<td>98</td>
<td>0.005</td>
<td>395,817</td>
<td>With datum and short meshes, used to investigate the effect of flat plate length</td>
</tr>
</tbody>
</table>

Table 8.2-1 Meshes used for the effect of flat plate length

The computational domain was a cube with nominal side length of 500 units. Since the effects of sideslip were to be investigated by other members of AG34, a half model was used to keep the mesh sizes down. The leading edge of the flat plate was coincident with the front face of the computational domain.

The bump geometry was prepared by DERA, Bedford (section 4.1). In the axes system chosen by DERA, the leading edge of the conical stream surface was approximately at x=1. The stream surface was terminated, arbitrarily, at x=4 and the surface stream-wise slopes gradually reduced to zero at x=5 using circular arc blending segments. The bump height at x=5 was 1.102 units. Figure 8.2-3 shows sectional views of the bump. The junction of the bump with the flat plate was smoothed to remove discontinuous changes of curvature. With the stream-wise surface slopes zero at x=5, the section shape there could then be extended aft as far as required. Most of the other contributors to this study extended the constant cross-section portion of the bump to the downstream limit of the computational domain. The bump used for the BAE SYSTEMS studies described here was continued aft as far as x=20, approximately, and then closed off with an aft fairing of arbitrary shape (Figure 8.2-4). Since the area of interest was confined to x≤6, this departure from the otherwise common practice of continuing the bump to the back face of the computational domain was not expected to have any effect on the results. No convergence problems were experienced due to the aft fairing.

The computational domain was meshed using the BAE SYSTEMS proprietary ‘Autogrid’ program. There are three stages to this process:

1) The cube-shaped domain is divided recursively based on 3 criteria:
   a) The user-defined maximum cell size in the mesh.
   b) User defined grid control sources.
   c) The user defined maximum cell size of any cell intersecting the geometry.

This subdivision process creates a Cartesian mesh with ‘hanging’ faces and nodes.
2) All the cells intersecting the geometry, contained within the geometry and within a user-defined distance of the geometry surface are removed from the Cartesian mesh, leaving a hole. The boundary of the hole is termed the 'shell'.

3) The shell nodes are repositioned to align the shell faces better with the surface of the geometry and the nodes are projected onto the surface. The connections between the shell and the surface are then used to create body fitted cells close to the surface. The first cell height is specified by the user.

The resulting mesh satisfies the requirement for a body-fitted layer of prismatic cells for Navier-Stokes calculations while allowing the remainder of the domain to be meshed economically.

8.2.2 Boundary conditions

The procedure adopted for the pressure inlet face of the domain (Error! Reference source not found.) was to fix the "gauge static pressure" at a level of 101500 Pa, relative to an operating (or reference) pressure set at 0. The "gauge total pressure" and free stream total temperature was set using the standard isentropic flow relationships. Choosing M=1.8, static temperature=273.11K and $\gamma = 1.4$ led to values of $P_T = 583198.7 \text{ Pa}$ and $T = 450.1 \text{ K}$. For the cases run with the k-ε turbulence model, the turbulence of the incoming stream was specified by an initial turbulence intensity of 1% and a viscosity ratio of 10. This means that the turbulent or 'eddy' viscosity was set to be 10 times the laminar (or molecular) viscosity level determined using Sutherland's law. The Spalart-Allmaras turbulence model required only the specification of the initial turbulent viscosity ratio i.e. 10.

For the values of density and velocity commensurate with the pressure and Mach number ($\rho = 1.29357 \text{ kg/m}^3$ and $U = 596.45 \text{ m/s}$), a reference viscosity can be defined for the desired Reynolds number

$$\mu_0 = \frac{\rho UL}{R_e}$$

In this instance, the characteristic dimension used for Reynolds number was the bump height of 1.10201 units. A typical value of Reynolds number, based on bump height was chosen to be $7.87 \times 10^5$ (8-1). Thus the reference viscosity was set to

$$\mu_0 = \frac{1.29357 \times 596.45 \times 1.10201}{7.87 \times 10^5} = 0.0010804 \text{ kg/ms}$$

These values were used for all three meshes in the study of the effect of flat plate length. Assuming a 1/7th power law velocity profile in the turbulent boundary layer on the flat plate, these conditions were calculated (section 4.2.3) to result in the following values for the ratio of the boundary layer incompressible displacement thickness to bump height:

<table>
<thead>
<tr>
<th>Flat plate length</th>
<th>$\frac{\delta}{k}_{\text{incomp}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.03</td>
</tr>
<tr>
<td>50</td>
<td>0.07</td>
</tr>
<tr>
<td>98</td>
<td>0.12</td>
</tr>
</tbody>
</table>

Table 8.2-2: Calculated variation of displacement thickness with flat plate length

Setting the flow conditions in terms of Mach number and Reynolds number made the solution independent of scale.
The pressure outlet face (Error! Reference source not found.) was set with a gauge pressure of 101500 Pa. Zero backflow was assumed so that the backflow total temperature was set equal to the free stream static temperature of 273.11K. Turbulence was specified in the same manner as for the pressure inlet face.

The walls were set to be smooth. The standard k-ε turbulence model with standard wall functions was used for all three flat plate lengths.

Fluent Version 5.3.1 was used for all the cases.

8.2.3 Results

The cases were all run for 2000 iterations with convergence being judged by the drag coefficient for the front part of the bump. Although the pressure drag terms converge relatively quickly, the viscous drag terms will only converge when the viscous effects have settled down. Since the boundary layer thickness on the bump was of prime interest, this choice of convergence criterion was appropriate. As expected, the flat plate length 98 case was the slowest to converge (presumably due to the greater thickness of the boundary layer developed in this case). The history of the forebody drag coefficient (reference area was chosen to be unity) for this case is shown in Figure 8.2-5. The corresponding normalised residuals are shown in Figure 8.2-6.

Figure 8.2-7 compares the $y^+$ values achieved on the bump centreline for the three cases. The variation in $y^+$ values reflects the variation in first cell height and wall shear stress in the three cases. Thus, flat plate lengths of 17 and 98 units had the same first cell height of 0.005 but the lower wall shear stress experienced by the 98 unit flat plate resulted in the lowest $y^+$ values. The logarithmic law for mean velocity (the 'law of the wall') is generally considered to hold for $30 < y^+ < 200$, although in Fluent it is used for $y^+$ values down to approximately 12. (For $y^+ < 12$, Fluent treats the inner layer as laminar.) Thus, although the $y^+$ values fell briefly to below 30 at the foot of the bump, $x=1$, for the flat plate lengths of 17 and 98 units, this was not expected to have a significant effect on the solution. Figure 8.2-8, Figure 8.2-9 and Figure 8.2-10 show contours of $y^+$ on the surface of the bump for flat plate lengths of 17, 50 and 98 units, respectively. It is evident that the $y^+$ values on the centreline capture the full range of $y^+$ values seen on the bump and flat plate.

Table 8.2-3 lists the boundary layer displacement thicknesses for the three flat plate lengths investigated. Note that the design values of $(\delta^*/k)_{\text{incomp}}$ of 0.03, 0.07 and 0.12 (Table 8.2-2) were largely achieved in the CFD solutions. In order to determine the boundary layer parameters tabulated in Table 8.2-3, flow data along lines normal to the surface were extracted from the CFD solution using the Line/Rake tool in Fluent 5. The edge of the boundary layer was determined using the method of 8-2. Following the approach used by NLR (8-3), the boundary layer compressible displacement thickness was determined (in a Microsoft Excel worksheet) by evaluating the following integral:

$$\delta^*_{\text{comp}} = \int_0^\delta \left(1 - \frac{\rho \mathbf{v} \cdot \mathbf{v}_{\text{edge}}}{\rho_{\text{edge}} \mathbf{v}_{\text{edge}} \cdot \mathbf{v}_{\text{edge}}} \right) \, dn$$

Here, $\rho$ is the local fluid density and $\mathbf{v}$ is the local velocity vector at a point in the boundary layer and $\rho_{\text{edge}}$ and $\mathbf{v}_{\text{edge}}$ are the density and velocity vector, respectively, at the edge of the boundary layer, at normal distance $\delta$ from the surface. The integral is evaluated with respect to normal distance from the limits of 0 and $\delta$. The integral was evaluated in a Microsoft Excel worksheet, using trapezium rule integration. The equivalent incompressible displacement thickness was then evaluated by repeating the calculation with the local density set to unity throughout the boundary layer.
The results in Table 8.2-3 show that the bump is effective at reducing the boundary layer thickness on the centreline. Values of $(\delta^*/k)_{comp}$ at $x=4$ are approximately $2/3$rds the value at $x=0$, for all cases. Figure 8.2-11 and Figure 8.2-12 show the effect of flat plate length on the boundary layer Mach number profiles at $x=0$ and $x=4$, respectively. The edge of the boundary layer may be gauged from the point where the Mach number is equal to the free stream Mach number (1.8) at $x=0$ or from where the Mach number profile runs parallel to the profile for inviscid conical flow at $x=4$.

Figure 8.2-13 shows the span-wise variation of the displacement thicknesses at $x=4$ for the flat plate length of 50 units, only. This figure also includes data from NLR (using the single block, structured mesh ENSOLV code with the k-ω turbulence model). The trends of the two sets of results (Fluent and NLR) for the compressible displacement thickness are similar. The displacement thickness is a minimum on the centreline and increases outboard, with the maximum thickness occurring at the junction of the curved surface of the bump with the flat plate. There was insufficient time to obtain this level of detail for the flat plate lengths 17 and 98 cases.

Figure 8.2-14 shows the effect of flat plate length on the local skin friction coefficient on the extended bump centreline. This figure also shows the variation of local skin friction coefficient with $x$ determined using the Prandtl-Schlichting skin friction formula for a turbulent boundary layer on a smooth flat plate at zero incidence ($8-4$). By this formula, the local skin friction coefficient in incompressible flow, $(c_f)_M=0$, at a distance $x$ from the leading edge of a flat plate is given by

$$ (c_f')_{M=0} = (2 \log_{10} \text{Re}_x - 0.65)^{-2.3} $$

where $\text{Re}_x$ is the Reynolds number based on distance, $x$, from the leading edge of the flat plate. Furthermore, albeit with scant supporting data, $8-4$ suggests that, at $M=1.8$, the ratio of the compressible to the incompressible values of local skin friction coefficient is 0.7, approximately. The lines in
Figure 8.2-14 were derived with the following equation:

\[
\left( \frac{c_f^*}{M=1.8} \right) = 0.7 \left( 2 \log_{10} \operatorname{Re}_{x} - 0.65 \right)^{2.3}
\]

The Fluent results follow the same general trend as the Prandtl-Schlichting law. Near the leading edge of the plates, where the boundary layer is initially thin, the shear stress at the wall is high, resulting in a relatively high value of local skin friction coefficient. As the boundary layer thickens with increasing distance from the leading edge, local skin friction coefficient falls as the shear stress at the wall reduces. The CFD results and the Prandtl-Schlichting prediction are both evaluated at the centres of the wall adjacent cells on the bump extended centreline. For the 17 unit long flat plate, the centre of the first cell was closer to the leading edge of the plate than for either of the other cases. Consequently, the first plotted value of skin friction for this case is higher than for the flat plates of length 50 and 98 units. Since the onset flow conditions were the same for all three cases, the boundary layer growth rates will also be the same, and the wall shear stress and thence the local skin friction coefficient will be the same at a given distance from the flat plate leading edge.

Figure 8.2-15 shows the variation of local skin friction coefficient with x on the centreline, close to and on the bump. In general, the longer the flat plate and therefore the thicker the boundary layer, the lower the values of local skin friction coefficient. This monotonic relationship is preserved on the surface of the bump. Since the local skin friction coefficient never falls to zero, the CFD results do not indicate any separated flow. However, for the 98 unit long flat plate, the local skin friction coefficient does fall to the low value of 0.0005 at the foot of the bump as the flow passes through the conical shock.

Figure 8.2-16 shows the effect of flat plate length on the variation of surface static pressure coefficient on the bump. Also shown is the pressure distribution for inviscid conical flow, using data supplied by DERA, Bedford (8-5). As expected, the shorter the flat plate in front of the bump, the thinner the boundary layer, the more rapid the pressure rise at the conical shock, and the closer the flow conforms to the inviscid ideal. The solutions all show a small “bump” in pressure ahead of the expansion at x=4, most notably for the le=17 case. This is a fairly normal numerical anomaly within most CFD codes which becomes more hidden as the boundary layer gets thicker.

Figure 8.2-17 and figure 8.2-18 show the effect of flat plate length on the span-wise \( c_{p} \) distribution on the bump at x=2 and x=4, respectively. Also shown on these plots is the span-wise \( c_{p} \) distribution for the design case of inviscid, conical flow. The junction of the curved surface of the bump with the flat plate marks the trace of the conical shock that the bump is designed to create. In both of these plots the viscous results for the flat plate length of 17 units are closest to the conical flow ideal. Considerable ‘smearing’ of the shock in the span-wise direction is evident in all the viscous solutions, presumably due to the outboard drift of the boundary layer. (Figure 8.2-13 shows that the boundary layer displacement thickness is greatest at the location of the inviscid, conical shock, i.e. at the junction of the curved surface of the bump with the flat plate.)

8.2.4 Discussion

There is a close match of the CFD-determined values of \( \left( \frac{\delta^{+}}{k} \right)_{\text{incomp}} \) (i.e. the ratio boundary layer incompressible displacement thickness to bump height) at x=0 with the design values (Table 8.2-2 and Table 8.2-3). Thus the logarithmic velocity distribution law used in the standard wall functions in the CFD solutions must closely match the \( 1/7 \)th power velocity distribution law used to derive the design values (8-1). For the 50 unit long flat plate the Reynolds number, based on bump height, of \( 7.87 \times 10^{5} \) results in a Reynolds number based on flat plate length of approximately \( 3.5 \times 10^{5} \) at x=0. 8-4 indicates this is about the upper level of validity for the \( 1/7 \)th power law, whereas the logarithmic law is valid for the whole range of Reynolds numbers up to \( 10^{6} \). The Prandtl-Schlichting law (8-4) for the variation of the local skin friction coefficient with
distance on the flat plate was derived from the logarithmic law of velocity distribution in the boundary layer. Although the compressibility correction is somewhat uncertain, there is quite a good match between the compressibility-corrected Prandtl-Schlichting law and the CFD-derived variation of local skin friction coefficient (figure 8.2-14).

The bump is effective at sweeping the low energy boundary layer flow outboard (Figure 8.2-13). The boundary layer displacement thickness is at a maximum approximately at the junction of the curved surface of the bump with the flat plate. This is also the location of the conical shock, in inviscid flow at the design Mach number. Because the boundary layer is thickest here, there is substantial 'smearing' of the shock evident in the CFD solutions. However, this is believed to be a true effect of viscosity rather than a numerical effect.

In general, the effect of varying flat plate length on the bump performance is as expected. The shorter the length of flat plate upstream of the bump, the thinner the approaching boundary layer. This results in a thinner boundary layer on the bump so that the bump pressure distribution is closer to the conical flow ideal. The range of static pressures on the bump is greatest for the case with the thinnest boundary layer. This means that the span-wise pressure gradients to divert the approaching boundary layer are also higher when the boundary layer is thin. Thus, the bump would be expected to perform best as a boundary layer diverter with a thin approaching boundary layer.

It is interesting to note that the static pressure coefficient, \(c_p\), on the flat plate upstream of the bump is not quite zero. In fact, for the flat plate length 50 case, the value is approximately constant at approximately 0.003. For this case, the compressible displacement thickness at \(x=0\) is 0.1151h, where h is the bump height. If the growth of the displacement thickness is assumed to be more or less linear with x, the displacement volume is a wedge whose surface slope is

\[
\frac{dz}{dx} = \frac{0.1151h}{50} = 0.00254
\]

for a bump height, h, of 1.102 units. By Ackeret's law for two-dimensional supersonic flow, the pressure coefficient on an inclined surface of slope \(dz/dx\) is

\[
c_p = \frac{2}{\sqrt{M^2 - 1}} \left( \frac{dz}{dx} \right)
\]

Substituting the surface slope of the displacement volume and the free stream Mach number (1.8) in this expression gives

\[
c_p = \frac{2}{\sqrt{1.8^2 - 1}} \left( 0.00254 \right) = 0.0034
\]

Thus, the non-zero value of \(c_p\) on the flat plate upstream of the bump observed in the CFD solutions is consistent with the notion of a wedge-shaped boundary layer displacement volume that alters the free stream flow direction, marginally. Since the uncertainty associated with experimental measurements of \(c_p\) is typically ±0.003, this effect is unlikely to be confirmed by experiment.

The span-wise pressure distributions at \(x=4\) (figure 8.2-18) show some apparent scatter in the CFD results near the centreline, i.e. at 25<y<27, approximately. This is thought to be due to the lack of alignment of the cutting plane at \(x=4\) with the grid (Figure 8.2-19). The values extracted along the cutting plane are the values at the nearest nodes to the cutting plane. Because of the lack of alignment of the grid with the cutting plane, the precise x values of the nodes at which data are extracted can vary. When this 'tolerance' in the actual x values (for a given nominal x value) is combined with significant x-wise gradients in the flow properties being extracted, the
result can be a rather ‘noisy’ distribution. For the static pressure coefficients, the x-wise
gradients are relatively low (Figure 8.2-16) and the grid non-alignment effect described above is
not too severe. However, x-wise gradients of local skin friction coefficient are very high at x=4 (Figure 8.2-15) resulting in very ‘noisy’ span-wise distributions (not presented here) at constant x values.

8.2.5 References

8- 1 Hodges, J, DERA/AS-HWA(BED)/8/2/5/4, 29th March, 1999
8- 2 Johnson, D.A, 'Nonequilibrium algebraic turbulence modeling considerations for
transonic airfoils and wings', AIAA 92-0026, 1992
8- 3 'Calculation of boundary layer displacement thickness'. Email from J.E.J.Maseland,
NLR, 20th December, 1999
8- 5 'Conical flow'. Email from J.Hodges, DERA (Bedford). 10th December, 1999
Figure 8.2-1 Sketch of the computational domain

Figure 8.2-2: 3/4 front view of bump between x=0 and x=5
Figure 8.2-3: Sectional views of the bump geometry
Figure 8.2-4: ¾ front view of half bump used in CFD study

Figure 8.2-5: Drag history on the bump forebody for a flat plate length of 98
Figure 8.2-6: Normalised residuals for a flat plate length of 98
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on wall y+ on the bump centreline

Figure 8.2-7: Effect of flat plate length on $y^+$ on the bump centreline
Figure 8.2-8: Contours of wall $y^*$ for flat plate length 17

Figure 8.2-9: Contours of wall $y^*$ for flat plate length 50
Figure 8.2-10: Contours of wall $y^+$ for flat plate length 98
Garteur TP 129

Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: Mach number variation along surface normal at x=0, y=25

Figure 8.2-11: Effect of flat plate length on Mach no. variation in the b.l at x=0
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas:
Mach number variation along surface normal at x=4, y=25θ=0 degs)

Figure 8.2-12: Effect of flat plate length on Mach no. variation in the b.l at x=4
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas.Z; predicted boundary layer displacement thicknesses at x=4: edge of b.l located using NLR method (Ref. AIAA 92-0026)

![Graph showing the spanwise distribution of displacement thickness δ/h at x=4, flat plate length 50](image)

*Figure 8.2-13: Spanwise distribution of δ*/h at x=4, flat plate length 50*
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on local skin friction on the bump centreline

Figure 8.2-14: Effect of flat plate length on local skin friction on the centreline
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on local skin friction on the bump centreline

Figure 8.2-15: Effect of flat plate length on local skin friction close to the bump

- 77 -
Garteur AG 34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on pressure coefficient on the bump centreline (y=25)

Figure 8.2-16: Effect of flat plate length on the $c_p$ distribution on the centreline
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on spanwise variation of pressure coefficient at x=2

Figure 8.2-17: Effect of flat plate length on spanwise $c_p$ distribution at x=2
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_re7.87_x17_newflu.cas, m1.8_re7.87_x98_newflu.cas: effect of flat plate length on spanwise variation of pressure coefficient at x=4

Figure 8.2-18: Effect of flat plate length on spanwise $c_p$ distribution at $x=4$
Figure 8.2-19: Plan view of surface mesh showing x=4 cutting plane
9. Presentation and Analysis of Effect of Turbulence Modelling

The investigation of the effect of flat plate length at M1.8 (see Section 8.2) made use of the k-\(\varepsilon\) turbulence model with standard wall functions. In order to investigate the effect of a different turbulence model, an additional case was run with the Spalart-Allmaras turbulence model. In its original form, this one equation turbulence model is a low Reynolds number model, requiring a volume mesh that is fine enough to resolve the laminar sub-layer of the boundary layer. However, within Fluent, if the first cell height is too large to resolve the laminar sub-layer (requiring \(y^+\) values of the order of unity) then standard wall functions are automatically invoked. The effect of the turbulence model was investigated at M1.8, only, for the datum geometry (flat plate length 50 units) and for the datum Reynolds number (7.87x10^5, based on bump height).

9.1 Mesh

The Spalart-Allmaras case (m1.8_sa_newflu.cas) made use of the same mesh as the datum case at M1.8 (m1.8_re7.87_x50_newflu.cas) which used the standard k-\(\varepsilon\) turbulence model. For this mesh, the first cell height was 0.01 units resulting in \(y^+\) values of order 10. Thus, standard wall functions were used for both the solutions discussed here. Details of the mesh are given in Section 8.2.1.

9.2 Boundary conditions

Apart from the choice of turbulence model, the boundary conditions for the Spalart-Allmaras case were identical to the conditions used for the k-\(\varepsilon\) case (see Section 8.2.2).

9.3 Results

Figure 9-1 and Figure 9-2 present the history of drag on the bump forebody and the normalised residuals for the Spalart-Allmaras case (m1.8_sa_newflu.cas). The normalised residuals show a limit cycle behaviour which is also evident in the drag history. However, the drag was converged to the third place of decimals – the criterion used for the other cases in this study (see Section 8.2) – and the solution was deemed adequate for the purpose of investigating the effect of turbulence model.

Figure 9-3 presents the variation of local skin friction coefficient, \(c_f'\), with x on the extended centreline of the bump. On the flat plate, upstream of the bump, the two turbulence models give similar results with the Spalart-Allmaras results showing marginally lower skin friction values than the k-\(\varepsilon\) results. However, on the surface of the bump (Figure 9-4), the differences between the two cases are more pronounced. The Spalart-Allmaras case is still showing consistently lower \(c_f'\) values than the k-\(\varepsilon\) case, but the differences are larger. This is most marked between x= 1 and x=4, approximately, where the pressure gradient on the centreline of the bump is adverse.

Figure 9-5 compares the spanwise variation of \(c_f'\) at x=4 for the two cases. The difference is large on the centreline and reduces with increasing distance from the centreline. The apparent scatter/noise evident in the \(c_f'\) results near the centreline at this station is believed to be due to lack of alignment of the mesh with the cutting plane, combined with a high local streamwise gradient of \(c_f'\) (Figure 9-4). This effect is discussed in Section 8.2.4.

Figure 9-6 compares the \(y^+\) values on the extended bump centreline for the two cases. These differences follow from the differences in \(c_f'\), discussed above, since the mesh and boundary conditions are the same for the two cases.

Figure 9-7 compares the distribution of static pressure coefficient, \(c_p\), on the bump centreline for the two cases. The two pressure distributions are effectively identical. Figure 9-8 compares the spanwise \(c_p\) distributions for the two cases at x=4. There is a small difference between the distributions but it is not significant.
The profiles of Mach number in the boundary layer for the two cases, at x=0 and x=4 on the centreline, are compared in Figure 9-9 and Figure 9-10, respectively. At x=0 (i.e. on the flat plate, just upstream of the bump), the two profiles are effectively identical. At x=4, the two profiles are of slightly different shape in the boundary layer but identical further out. Table 9-4 lists the boundary layer parameters for the two cases at x=0 and x=4 on the centreline. These parameters were determined from the CFD solutions using the method described in Section 8.2.3 and are non-dimensionalised by bump height (h=1.10201 units).

<table>
<thead>
<tr>
<th>Case</th>
<th>Mach</th>
<th>Turbulence model</th>
<th>x</th>
<th>δ/k</th>
<th>(δ/k)_comp</th>
<th>Bump Efficiency</th>
<th>(δ/k)_boom</th>
</tr>
</thead>
<tbody>
<tr>
<td>Datum</td>
<td>1.8</td>
<td>k-ε</td>
<td>0</td>
<td>0.6604</td>
<td>0.1151</td>
<td>0.0667</td>
<td>0.0667</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3.5</td>
<td>0.4925</td>
<td>0.0821</td>
<td>29%</td>
<td>0.0651</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4</td>
<td>0.4528</td>
<td>0.0770</td>
<td>33%</td>
<td>0.0517</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>1.8</td>
<td>S-A</td>
<td>0</td>
<td>0.6291</td>
<td>0.1115</td>
<td>0.0653</td>
<td>0.0652</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3.5</td>
<td>0.4688</td>
<td>0.0817</td>
<td>27%</td>
<td>0.0652</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4</td>
<td>0.4371</td>
<td>0.0749</td>
<td>33%</td>
<td>0.0525</td>
</tr>
</tbody>
</table>

Table 9-4: Boundary layer parameters determined from the CFD solutions

Consistent with the similarity of the boundary layer profiles at x=0, the compressible displacement thicknesses for the two solutions differ by less than 1%. This is despite a more significant difference in the height of the boundary layer edge (approximately 5%). However, at x=4, the difference in the boundary layer profiles (Figure 9-10) leads to a 2% difference in the values of the compressible displacement thickness.

9.4 Discussion

The differences between the k-ε and Spalart-Allmaras solutions for the datum geometry at M1.8, Reynolds number, based on bump height, of 7.87x10^5, are generally small. Of the parameters investigated, the differences are most evident in the variation of local skin friction coefficient, c_p', in regions of adverse pressure gradient. On the flat plate, where there is negligible pressure gradient upstream of the bump, the c_p' values for the two solutions compare well. The biggest differences in c_p' are observed at the conical shock and on the conical streamsurface portion of the bump (between x=1 and x=4, approximately) where the streamwise pressure gradient is adverse. Values of c_p' for the Spalart-Allmaras turbulence model are consistently lower than those for the k-ε turbulence model. This trend was confirmed by running a case for the flat plate length 98 mesh and the Spalart-Allmaras turbulence model, for comparison with the k-ε case for which results are presented in Section 8.2. The comparison of c_p' values for the two cases at this flat plate length is presented in Figure 9-11. Once again, values of c_p' for the Spalart-Allmaras case are consistently lower than for the k-ε case. These results suggest that the Spalart-Allmaras turbulence model is more likely to predict separation than the k-ε turbulence model.

The differences in c_p' are reflected in the boundary layer profiles: on the flat plate, at x=0, the profiles of Mach number in the boundary layer are effectively identical for the two cases. On the bump, however, at x=4, there is a small difference between the profiles of Mach number in the boundary layer. Since the distributions of static pressure coefficient, c_p, on the bump
are effectively identical, the small differences between the turbulence models have a
negligible effect on the displacement of the effectively inviscid part of the flow.

For a given number of iterations, the Spalart-Allmaras (one equation turbulence model) cases
were noticeably faster to run than the corresponding k-ε (two equation turbulence model)
cases, although there are no data to quantify this difference.
**Figure 9-1:** Drag history on the bump forebody for the Spalart-Allmaras case

**Figure 9-2:** Normalised residuals for the Spalart-Allmaras case
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on local skin friction on the bump centreline

Figure 9-3: Effect of turbulence model on $c_f'$ on the extended centreline
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on local skin friction on the bump centreline.

Figure 9-4: Effect of turbulence model on $c_f'$ on the bump centreline.
Garteur AG 34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on spanwise variation of local skin friction coefft. at x=4

Figure 9-5: Effect of turbulence model on spanwise variation of $c'_f$ at x=4
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on wall $y^+$ on the bump centreline

Figure 9- 6: Effect of turbulence model on $y^+$ on the extended bump centreline
Garteur AG 34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on pressure coefficient on the bump centreline (y=25)

Figure 9-7: Effect of turbulence model on $c_p$ on bump centreline
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on spanwise variation of pressure coefficient at x=4

Figure 9-8: Effect of turbulence model on spanwise variation of $c_p$ at x=4
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas, m1.8_sa_newflu.cas: Effect of turbulence model on Mach number along surface normal at x=0, y=25

Figure 9-9: Effect of turbulence model on Mach no. profile at x=0 on the centreline
Garteur AG34 bump: m1.8_re7.87_x50_newflu.cas and m1.8_sa_newflu.cas: effect of turbulence model on Mach number profile along surface normal at x=4, y=25 θ=0 degs

Figure 9-10: Effect of turbulence model on Mach no. at x=4 on the centreline
Garteur AG34 bump: m1.8_re7.87_x98_newflu.cas, m1.8_sa_x98_newflu.cas: Effect of turbulence model on local skin friction coefficient on the centreline: Flat plate length 98

Figure 9-11: Effect of turbulence model on $c'_f$ on centreline of flat plate length 98
10. Discussion

10.1 General Comments

Because there were so many separate contributions, each of the main sections covering the computations contains an individual detailed discussion section. The following is a brief summary of these, together with, where possible, some interpretation of the flows and implications for application of such a bump feature.

![Figure 10.1; Near-surface streamlines and boundary layer profiles at Mach 0.8, zero sideslip](image)

10.2 Interpretation of the CFD Code Comparison Cases

Mesh sensitivity studies were performed very early on in the collaboration by a number of contributors. There was broad agreement on the required mesh attributes close to the bump and the majority of contributors adopted a similar approach thereafter.

![Figure 10.2-1; Surface pressure distributions at the bump centre-line and at two cross-flow planes at Mach 0.8 (from Figs 5.2-1, 5.2-4 and 5.2-5)](image)
For the two cases computed by the majority of the contributors (Mach 0.8 and 1.8), the agreement of the predicted surface pressure distributions was good (Figs 10.2-1 and 10.2-2). This level of agreement was considered essential to allow meaningful comparison of boundary layer flow parameters.

Comparing the centre-line pressure distributions, acceleration of the flow around the bump termination, commencing at $x = 4$, had a forward effect in both cases. This was modest at $M = 1.8$, as might be expected for a supersonic speed, but very substantial at $M = 0.8$. If a bump such as this were employed ahead of an external compression intake, roughly the opposite of this situation would exist at supersonic speeds. A short distance ahead of the entry the flow would pass through a strong oblique or normal shock wave yielding a strong adverse pressure gradient in this region. Exposure of the bump termination is thus somewhat unrepresentative except perhaps for the case of supercritical intake operation. With the benefit of hindsight, it would have been better to extend the conical stream surface to perhaps $x = 5$ before the termination for computation of the supersonic case. This would have eliminated this effect at $x = 4$ such that the solution upstream of this would have been as close to the idealised conical flow as possible. At subsonic speeds, the effect of the termination dominates the pressure distribution from around $x = 1$ to $x = 4$, virtually inverting the cross-flow distribution between $x = 2$ and $x = 4$. As will be seen later, the effect of this on boundary layer displacement thickness appears very favourable. However, again, in an intake installation operating at a typical cruise condition with some amount of spillage, this region would be exposed to the intake pre-entry pressure rise, which, depending on the actual operating condition, would tend to reduce or perhaps even to cancel this effect. The flows computed here are thus of high relevance to use of such a feature as an intake boundary layer diverter, but it must be appreciated that there will be very substantial differences relative to a real installation. An integrated design approach would be essential.

Predictions of local skin friction coefficient were reasonably good for the subsonic case but poor for the supersonic case. In the latter, considerable differences were present in the undisturbed boundary layer as well as on the bump. This would seem to point to an underlying problem in the methods rather than something specific to this bump case. Since skin friction was not of primary engineering interest, and was known from other experience to be problematic to predict, the problem was noted but was not explored further by any of the contributors.
Trends in predicted boundary layer displacement thickness were very consistent at both the subsonic and supersonic speeds. The DERA SAUNA code over-predicted \( \delta^* \) in both instances. There were some problems in SAUNA with initiation of the boundary layer at the plate leading edge which, it is suspected, could have led to an increased effective plate length. Discontinuities in the distribution of \( \delta^* \) are believed to be due to the method used for its extraction. Whilst the method used by each contributor was nominally the same, there were, evidently, differences in implementation that produced this effect in some contributions. The consistency in trend of \( \delta^* \) was considered to be good enough to allow effects of Mach number, sideslip angle, Reynolds number and turbulence model to be examined separately by the contributors using these different codes.

10.3 Effect of Mach Number

At Mach 0.6 to 1.2 the position of the peak displacement thickness was located (Fig 10.3-1) at the outer limit of the bump (at 1.0 bump half-span). This position moved inboard as Mach number increased further, reaching 0.78 half-spans at Mach 1.8. The lateral extent of the bump flow interaction with the approaching boundary layer was at its greatest in the transonic case, extending out to about 4.1 bump half-spans at \( x = 4 \) at Mach 1.2 (Fig 10.3-2). This corresponds to the presence of a modestly swept detached shock wave. As Mach number
was increased above this, the extent of lateral influence reduced to about 2.0 half-spans at \( x = 4 \), Mach 1.8. This corresponds to the nominal bump design condition where in an ideal inviscid flow the lateral extent of the bump influence would be the intersection of the conical oblique shock with the surface, by definition, at one bump width. Insensitivity to Mach number of the position of peak boundary layer thickness would be an advantage in choosing the width of an intake placed behind it. The intake width would not need to be chosen to favour one particular operating condition.

The location of minimum displacement thickness is the bump centre-line at all of the conditions shown. To facilitate comparison of the bump effectiveness at thinning the boundary layer at this location at a range of conditions, effectiveness is represented as:

\[
\left(1 - \frac{\delta_b^*}{\delta_u^*}\right)
\]

where \( \delta^* \) is boundary layer displacement thickness, subscript \( b \) indicates the bump centre-line and subscript \( u \) indicates a stream-wise location in the undisturbed boundary layer corresponding to the start of the bump (\( x = 0 \)). Thus if the local boundary layer thickness was reduced to zero by the bump, this quantity would equal 1.0. An alternative, and in some ways more convenient, approach would be to use the same streamwise location both in the undisturbed boundary layer and on the bump. This may be especially suitable for experimental work where, for instance, data are to be acquired at a single plane at a range of onset flow conditions.

Figure 10.4-1; Effect of sideslip on surface pressure distribution and boundary layer displacement thickness at a stream-wise station (\( x=4 \)) at Mach 0.8 (from Figs 7.1-3 and 7.1-4)

Figure 10.3-2; Effect of onset Mach number on boundary layer displacement thickness at four stream-wise stations on the bump centre-line
The effectiveness of the bump defined in this simple way is revealed that there is little or no apparent sensitivity to Mach number at subsonic speeds but this changes quite dramatically (Fig 10.3-2) as Mach number becomes supersonic. Here, the effectiveness reduces to just over 0.2 at \( x = 4 \), Mach 1.8. At this level of diversion, the direct effect of the bump on the performance of a supersonic intake could be quite modest or even adverse, considering the greatly thickened boundary layer regions on either side of the bump (Fig 10.3-1). Unless the flow aft of the intake normal shock could be tailored to spill these regions, the effect would almost certainly be adverse. It is useful to note that the growth of the boundary layer is near linear with distance along the centre-line in this region even at subsonic speeds.

Comparison of the results for four contributors for \( x = 4 \), reveals that there is a disparity of up to 0.2 in bump effectiveness. Referring back to Fig 10.2-3, this is despite the apparently good agreement of the lateral \( \delta^* \) distributions for the DASA, BAe and NLR contributions. Because these curves cross, the effectiveness parameter greatly amplifies the differences.

The results acquired by Aerospatiale for Mach numbers 1.8 to 3.5 were performed very early on in the collaboration and due to changes in membership of the AG34 it did not prove possible to extract corresponding results for \( x = 4 \). Results were obtained at station \( x = 6 \), aft of the crest of the bump. This data is thus incompatible with the other bump effectiveness data and for this reason it is plotted separately (Fig 10.3-4). Data at \( x = 6 \) were extracted from two of the other contributions. The DERA SAUNA data for \( M = 1.8 \) corresponded very closely to the Aerospatiale result. The DASA data point did not agree as well, and its relationship with the SAUNA data was roughly the reverse of the situation at \( x = 4 \). Given the high stream-wise gradients of the boundary layer parameters in this region, this is not that surprising and it is difficult to draw any firm conclusion from these comparisons. The Aerospatiale results could suggest that diversion effectiveness reduces only gradually above the bump design Mach number, but further work would be required to verify this.

Figure 10.3-3: Effect of onset Mach number on boundary layer displacement thickness at a stream-wise station (\( x = 4 \)) on the bump centre-line
10.4 Effect of Sideslip

At Mach 0.8 the main effect of sideslip was to greatly increase boundary layer thickening on the windward side of the bump (Fig 10.4-1). There was a small increase in thickness across the rest of the bump relative to the zero sideslip case. The thickness distributions from the centre-line out onto the leeward side were very similar for 10 and 20 degrees of sideslip, despite the large difference in peak suction on the leeward side of the centre-line. Centre-line surface static pressure was almost invariant with sideslip.

Unexpectedly, the position of peak boundary layer thickness also varied little with sideslip. This is potentially an important result for an intake design in that it was anticipated that the bump would not divert the boundary layer effectively at high sideslip angles. It is important to note that what is referred to here as sideslip could be produced by aircraft incidence in a side-mounted position on the side of an aircraft. Cross-flow is a more generic description of what is simulated here.

At Mach 1.8, the effect of 10 degrees of sideslip was to thicken the boundary layer relative to the zero sideslip case across the whole of the bump lateral thickness distribution at x = 4 (Fig 10.4-2). In a similar manner to the subsonic case, there was little difference in the two results at non-zero sideslip. The pressure distribution was always positive, again with little variation close to the centre-line. The position of peak thickness was again almost invariant with sideslip.
10.5 Effect of Reynolds Number

In terms of the centre-line diversion effectiveness parameter at \( x = 3.5 \) and 4.0 (Fig 10.5-1), the bump became marginally less effective as onset boundary layer thickness was increased at \( M=0.8 \). This was the expected result. The Reynolds number range was chosen to represent a range of altitudes for a typical aircraft scale.

![Figure 10.5-1; Effect of Reynolds number on boundary layer diversion characteristics at Mach 0.8 (from Figs 8.1-10 and 8.1-12)](image)

The corresponding result for Mach 1.8 shows a similar trend for \( x = 3.5 \) (Fig 10.5-2). There is, broadly, a smaller difference between the two stations at this speed, which is consistent with Fig 10.3-2. The results for \( x = 4 \) show the opposite trend, that is, slightly increasing effectiveness with increasing onset boundary layer thickness. This effect was not investigated further. A difference between stations 3.5 and 4.0 is that the latter is within the region of forward influence of the bump termination. A difference in this forward effect seems the only possible physical explanation.

![Figure 10.5-2; Effect of Reynolds number on boundary layer diversion characteristics at Mach 1.8](image)

10.6 Effect of Turbulence Modelling

The main difference between the results for the two turbulence models studied was that for a given number of iterations, the Spalart-Allmaras (one equation turbulence model) cases were noticeably faster to run than the corresponding k-\( \varepsilon \) (two equation turbulence model). Refer to Section 9 for a more detailed description of the study.
10.7 Thoughts on the Application of the Work

10.7.1 Use as an Intake Boundary Layer Diverter

The results of the Action Group are encouraging for the application of a bump as a primary intake boundary layer diverter. The nominal bump design Mach number was 1.8. The main purpose of AG34 was to look first of all at how the bump performed at this design condition and then at off-design and manoeuvre conditions. A degree of boundary layer diversion was provided by the bump at all of the conditions studied. Surprisingly, the least diversion appeared to be provided at the design condition and it was most effective at subsonic speeds. It was, however, clear in the analysis that at subsonic speeds, flow acceleration around the termination produced a strong positive forward interaction with the flow on the bump. This interaction was strong enough to overcome the rather greater degree of boundary layer growth ahead of the bump than was present at supersonic speeds. Removal of this effect as a result of integration with the pre-entry flow around a subsonic intake could produce a very different result. It is worth noting that if a bump was to be used on a subsonic intake then it is unlikely that the current shape, derived from a supersonic cone flow, would be ideal. Furthermore, the ideal shape would be likely to be dependent on the shape of the intake entry. Determination of the optimum bump shape would be a complex process and could itself form the basis of a considerable research exercise. At Mach 1.8, the lateral extent of the region of thinned boundary layer at the end of the bump (x = 4) was quite small. The thickened regions on either side of the bump would be likely to fall inside the width of an intake. The bump thus appears suitable only for an external compression intake intended for sub-critical operation. This is because it would be essential to arrange the intake entry region to spill the thickened regions of the bump boundary layer. This could most easily be achieved by reverse-sweeping the intake entry such that the spillage regions were at the aft-most corners of the entry at the intersection of the side-walls with the body (e.g. Figs 1.1-1 and 1.1-2). This would perhaps tend to favour aircraft application over weapon application where extensive supercritical operation is usually required.

It was encouraging that the position of minimum bump boundary layer thickness was virtually invariant with both onset Mach number and sideslip (or cross-flow) angle. The latter result was quite unexpected and would greatly ease the task of integration with an intake. Thinking of this cross-flow as the effect of aircraft incidence in a side-mounted intake installation, it may be possible to further extend the positive incidence capability (at the expense of unused negative incidence capability) by cutting-back, or ‘scarfing’ the intake entry slightly.

It is clear that great care would be required in integrating a bump as a diverter for either a subsonic or supersonic intake. Elimination of the conventional diverter feature in some ways simplifies the intake design, as well as yielding a range of non-aerodynamic benefits, but this is at the expense of greatly increased aerodynamic complexity and hence integration difficulty. Complex aerodynamic interactions will need to be captured and designs evolved in a highly interactive manner. CFD tools appear ideally suited to this considerable challenge for the majority of the design space. Limitations will undoubtedly be encountered in the prediction of off-design operating boundaries and it is here that the majority of the research effort is likely to be required both to improve and to calibrate the tools.

10.7.2 Use in Combination with a Conventional Pitot-Type Boundary Layer Diverter

Refer to the Appendix at Section 14 for a possible application of a bump as a means of improving the performance of a weapon intake mounted on a conventional pitot-type diverter.

10.8 Collaborative Working

The collaboration was felt to have been generally successful in terms of compatibility of the results though it proved difficult to obtain an absolutely self-consistent set of calculations with the range of codes employed. New predictive capabilities have been implemented by the majority of contributors as a result of the collaboration and all participants have benefited from the shared experience of applying CFD methods to challenging and highly relevant intake problems. Successful use of CFD by a team drawn from different organisations to explore a new design and, also, to expand knowledge of the design space around familiar design has
been demonstrated. Both the benefits and the limitations of this way of working have been observed in the course of this Action Group. Overall, the outcome of the collaboration has been very positive.
11. Conclusions

11.1 General Conclusions
1. The flow regimes being studied appear to be amenable to CFD, which has matured sufficiently not to need experimental measurements for this class of case.
2. Although the geometry was simple, there were some difficulties using some codes. In particular disturbances at the leading edge of the plate caused initial problems for some codes.
3. There are several methods for the extraction of boundary layer thickness, $\delta^*$. None appears wholly satisfactory, but the best (for this application) appears to be that described by Johnson in Ref. 6.1-2. The prediction of skin friction proved problematic, especially at supersonic speeds. Differences in the prediction of skin friction in the undisturbed boundary layer point to a problem in the CFD codes used rather than something specific to the current case.
4. No gaps or particularly controversial results emerged from the CFD comparison exercise. No specific experimental work is seen as being required, though calibration of the boundary layer displacement thickness distribution results would be a worthwhile exercise.
5. Working as a group has enabled sensible results covering a wide range of flow conditions to be achieved quickly. Learning from others’ results and experiences helped greatly.
6. It is considered that all of the main objectives of this Study Case have been met.

11.2 Bump Flow Physics
7. The position of minimum boundary layer displacement thickness was found to vary little with both onset Mach number and sideslip (or cross-flow) angle. This result is potentially important for application of the bump as a primary intake boundary layer diverter.
8. The boundary layer diversion effectiveness of the bump at speeds up to Mach 1.8 was found to be least at the nominal design, Mach 1.8, condition itself and greatest at subsonic speeds (similar results were obtained for Mach 0.6 and 0.8).
9. Preliminary results obtained for Mach numbers up to 3.5 suggest that diversion effectiveness reduces gradually with increasing Mach number above 1.8 but further work would be required to verify this.
10. The bump termination had a very marked favourable forward influence on the bump flow. It is possible that this was mainly responsible for the high apparent bump diversion effectiveness at subsonic speeds. Further work will be required to determine whether this can be exploited in a real intake design.

12. Recommendations
1. Integration of a bump diverter with both subsonic and supersonic intakes should be studied. A method for the optimisation of bump shape for subsonic intake application would be required. The influence of intake entry shaping, particularly forward-sweep or scarfing, should be determined.
2. The influence of bump closure shaping on bump diversion effectiveness in an intake design should be studied carefully as a research exercise to determine whether this could be exploited in a real intake design.
### 13. Notation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>Constant in streamline calculation</td>
</tr>
<tr>
<td>Cp</td>
<td>Coefficient of pressure</td>
</tr>
<tr>
<td>Cf</td>
<td>Skin friction coefficient</td>
</tr>
<tr>
<td>h</td>
<td>Distance of plane from axis of cone</td>
</tr>
<tr>
<td>k</td>
<td>Bump height (above plane)</td>
</tr>
<tr>
<td>ℓ</td>
<td>Plate length ahead of x=0</td>
</tr>
<tr>
<td>M</td>
<td>Mach Number</td>
</tr>
<tr>
<td>P&lt;sub&gt;T&lt;/sub&gt;</td>
<td>Total pressure</td>
</tr>
<tr>
<td>r</td>
<td>Distance from axis of cone</td>
</tr>
<tr>
<td>Re&lt;sub&gt;k&lt;/sub&gt;</td>
<td>Reynolds number based on bump height</td>
</tr>
<tr>
<td>T&lt;sub&gt;T&lt;/sub&gt;</td>
<td>Total Temperature</td>
</tr>
<tr>
<td>U</td>
<td>Freestream velocity</td>
</tr>
<tr>
<td>V</td>
<td>Local velocity</td>
</tr>
<tr>
<td>x</td>
<td>Streamwise distance from apex of cone</td>
</tr>
<tr>
<td>y</td>
<td>Spanwise distance from axis of cone (but see note below)</td>
</tr>
<tr>
<td>z</td>
<td>Height of bump above x-y plane (but see note below)</td>
</tr>
<tr>
<td>α</td>
<td>Semi-angle of cone</td>
</tr>
<tr>
<td>β</td>
<td>Semi-angle of shock cone</td>
</tr>
<tr>
<td>γ</td>
<td>Ratio of specific heats, 1.4 in air at ISA conditions</td>
</tr>
<tr>
<td>δ&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Boundary layer displacement thickness</td>
</tr>
<tr>
<td>θ</td>
<td>Angle from z-axis of point in conical flow</td>
</tr>
<tr>
<td>μ&lt;sub&gt;0&lt;/sub&gt;</td>
<td>Characteristic viscosity</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
</tr>
</tbody>
</table>

N.B. The definitions of y and z given above are those used in Section 4.1 (Stream surface calculation). Having produced the geometry, 25.0 was added to all values of y and 1.0 subtracted from all values of z before distributing it to the participants. This was so that y had positive values in the area of interest and z was relative to the plane on which the bump was generated. The centreline of the bump is thus at y = 25.0 and the flat plate is at z = 0 for the remainder of this document.

This part presents from an industrial point of view an application of the bump as a useful external boundary layer bleed for supersonic air intakes. The work was performed by Aerospatiale Matra Missiles and is included in this report by their kind permission, as it was not part of the original work plan of the Action Group.

14.1 Design of the Bump

The generic design of intake described here is a typical 2D wedge intake where the external bleed diverter has been replaced by a conical shock bump. It is designed to operate at $M=2.5$. This differs from the concept used for low supersonic aircraft, where the bump is intended to replace both diverter and the ramp.

![Figure 14-1: An ‘on-design’ bump](image)

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_b$</td>
<td>Length of the bump</td>
</tr>
<tr>
<td>$h_c$</td>
<td>Height of the cowl</td>
</tr>
<tr>
<td>$\theta_1$</td>
<td>Angle of the ramp</td>
</tr>
<tr>
<td>$h_p$</td>
<td>Height of the external bleed</td>
</tr>
<tr>
<td>$l_{PA}$</td>
<td>Width of the air intake</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Angle of the shock</td>
</tr>
</tbody>
</table>

The air intake works on-design if the conical shock issued from the bump intersects the middle of the cowl, that is:

$$\tan \beta = \frac{h_c + h_p}{L_b + L_c}$$

The bump shape is designed to follow the streamlines created by the desired conical shock. Corresponding solutions are not unique since several bumps can generate the same conical shock. In order to obtain a solution, a variable has to be chosen (length or width).

14.1.1 Geometries Studied

Three different bumps were computed. The first one uses the $M=1.8$ so called “reference” bump used for the AG34 study. Its implantation (in place of the external bleed) is shown in figure 3.
The two other bumps were determined so that the intake works on-design for Mach 2.5, as described in the previous paragraph. The second bump corresponds to a wide solution (figure 4), whereas the third one is much longer and less wide (figure 5). One important point which has to be noticed is that these two bumps create a conical shock which impacts the centre of the intake cowl at Mach 2.5.

14.2 Performance of a Supersonic Inlet

The performance of an air intake is mostly given by the graph of the total pressure recovery versus the inlet mass flow rate for all possible terminal shock positions. At a given flight point, three different operating regimes exist:

- the **supercritical regime** occurs when the terminal shock is located downstream of the throat (that is in the subsonic diffuser). As the shock moves upstream because of an increasing pressure in the combustor, the mass flow rate remains unchanged and the total pressure recovery increases.
- the **critical regime** occurs when the position of the terminal shock corresponds to the aerodynamic throat
- the **subcritical regime** is obtained when the shock moves upstream from the throat and also from the cowl, so that the mass flow rate decreases significantly.

If, at some subcritical position of the terminal shock, it becomes aerodynamically impossible for the inlet to increase its pressure with respect to the pressure in the combustor, the shock will become unstable (this is the so called “buzz” phenomenon).

14.3 Numerical Procedure

The RANS computations were conducted with the GASP code using a two-equations k-ε turbulence model with wall laws. Inviscid fluxes are obtained with a Van-Leer scheme, the spatial reconstruction is made at 3rd order with the use of the Min-Mod limiter. The mesh sizes were about 300,000 nodes.

The following methodology has been applied in order to obtain the performance of the intake at all the regimes described in the last paragraph:

- a first computation is conducted with a supersonic outflow boundary condition (1st order extrapolation) at the end of the subsonic diffuser. This gives the **supercritical mass flow rate**.

- then a subsonic outflow condition (back-pressure) is applied. Several computations are made with different values of this back-pressure in order to describe the complete performance curve with all its operating modes (critical, sub-critical).

It should also be noticed that the convergence of the computation is checked of course with a decrease (or plateau) of the residuals but mostly with a real convergence of the physical values (mass flow rate and total pressure recovery).

14.3.1 Results of the Calculations

14.3.1.1 The Reference Bump

The results of the computations with the “reference” bump are presented in figure 6. Several points should be noticed:

- the bump is not designed for the Mach number (M=2.5) of our calculations. Therefore the boundary layer deviation is not perfect at this flight condition.

- The bump is not designed specifically for the intake. In this case, this means that the shock it generates is too weak and that the intake will work above design
Of course, this will lead to losses in total pressure recovery. Figure 6 shows the Mach number contours on the symmetry plane for three values of the back-pressure: \( P = 180, 195 \) and \( 215 \) kPa. Therefore three different positions of the terminal normal shock are obtained. At \( P = 215 \) kPa, the shock is located at the throat, and this is also the last stable point. In this case, no subcritical regime exists and the performance curve is almost a vertical line.

### 14.3.1.2 The “Wide” Bump

The second computed geometry is the so-called “wide bump”. It has been designed to create a shock impinging the cowl on its middle. The Mach number contours on the symmetry plane are shown in figure 7 for the supercritical regime. It should be first noticed that the bump design was achieved: the oblique shock from the bump impacts the centre of the cowl. Three different positions of the normal shock in the subsonic diffuser are obtained for \( P = 180, 185 \) and \( 190 \) kPa. The mass flow rate remains constant and the total pressure recovery increases from 0.607 to 0.62.

Figure 8 shows the Mach contours for the critical and subcritical regimes. Back-pressures from 195 to 205 kPa make the shock move upstream around the throat leading to a loss of mass flow rate (0.927 to 0.90) with an increase of the total pressure recovery (0.62 to 0.65).

One important fact is the gap in the terminal shock position between 190 kPa (full supercritical regime) and 195 kPa (critical point). This is probably due to the very weak angle of the subsonic diffuser which leads to an almost constant section around the throat. Instability can arise because of interaction between the normal shock and the boundary layer.

### 14.3.1.3 The “Long” Bump

The last geometry is a “long” bump, which was also designed to make the inlet work on-design at \( M = 2.5 \). This objective was achieved as shown in figure 9, which presents the Mach number contours in the symmetry plane. Compared with the “wide” bump, this configuration has almost no subcritical regime, leading to a vertical line type performance curve.
14.4 Comparison of Performance

The three performance curves of the computed configurations are plotted and compared in figure 2. The best bump is the wide one, with the best total pressure recovery and mass flow rate. This is logical in the sense that this bump makes more streamlines get into the intake.

It should be noticed that several designs of bump make it possible to obtain any desired flow rate (with the limit of the flow rate that is normally lost in the external bleed).

![Performance curves](image)

*Figure 14-2: Performance curves*

14.5 Conclusion

An air intake equipped with a bump has a mass flow rate superior than unity with very good total pressure recovery (because of the pre-compression created by the conical shock), so that the bump can be a great way to replace the external boundary layer bleed.

All the computed bumps were designed so that the inlet works on-design at the middle of the cowl. Because the shock created by the bump is conical, another way would be to generate bumps that make the inlet be on-design at the cowl edges.

The design of the bump for an inlet is a very challenging task, but it gives the engineer more degrees of freedom in its conception and the effects of the bump are far more beneficial than an external bleed both for the mass flow rate and the total pressure recovery.
Figures:

Figure 14-3: The ‘Reference’ Bump

Figure 14-4: The ‘Wide’ Bump

Figure 14-5: The ‘Long’ Bump
Figure 14-6: Computed flow in the intake (reference bump)

Figure 14-7: Computed flow in the intake, supercritical operating mode (wide bump)

Figure 14-8: Computed flow in the intake, critical operating mode (wide bump)
Figure 14-9: Computed flow in the intake (long bump)