A Newtonian Solver for Hypersonic Flows

K. J. Austin & P. A. Jacobs
Department of Mechanical Engineering
The University of Queensland

February 1996 (revised July 96)

Abstract

A Newtonian flow solver has been developed for aerodynamic analysis of three dimensional geometries. The program *NEWT* is designed to read in a previously configured mesh and compute the aerodynamic forces and moments for hypersonic freestream conditions. Programming was completed using FORTRAN, compatible with both UNIX and DOS operating environments.

This work was initiated for the purpose of developing a flow solver capable of providing aerodynamic data for hypersonic vehicles. In this report, Modified Newtonian theory and the computational requirements of the code are described. A number of geometric configurations are provided as examples of applications of the Newtonian solver, including a blunt cone, heat shield and a scramjet configuration.

Contents

1 Introduction

The inviscid flow solver *NEWT*, uses an impact method to determine the pressure distribution over a three-dimensional body in hypersonic flow. *NEWT* is based on Newtonian flow theory established three centuries ago by Isaac Newton for low speed fluid dynamic analysis. Ironically, Newton's model is more associable to hypersonic flow fields, finding applications as an approximate method for determining pressure distribution over hypersonic bodies. (Anderson [1] presents a discussion on the relationship of Newtonian theory to hypersonic flow.) Being computationally inexpensive, the Newtonian flow procedure is suitable for inclusion in the inner loop of a simulation program or a design procedure for control systems.

The body geometry is defined by a surface mesh, with each individual element analysed independently as either an impact or shadow flow region. The surface pressure over an impact region is calculated using the local surface inclination relative to the freestream. Shadow flow regions are processed as having pressure coefficient equal to zero, equating the surface pressure to the freestream pressure. There are no routines within *NEWT* to analyse the complete flow processing for the intergrated airframe and engine arrangement found in scramjet vehicles. For such an application the solver is limited to finding estimates for impact pressure on forward facing surfaces at the front of the body and other external surfaces over the length. Blunt body shapes such as reentry vehicles do not have this limitation and provide the most useful applications for *NEWT*.

Following is the documentation of the Newtonian flow solver. A review of literature on aerodynamic modeling of hypersonic vehicles is provided as justification for the code. The theoretical basis of the program is given along with a detailed description of the surface modelling data required as input. The structure of the code is then described, detailing the application of Modified Newtonian flow to arbitrary configurations.

1.1 Background - Aerodynamic Modelling

A significant amount of flight/launch vehicle research is focusing on the ypersonic regime, for which ground-based testing is limited (in size, performance, and cost). With this in mind and through the benefit of modern computing power, computational methods in the aerodynamic design and analysis of aircraft are becoming common place. A recent example of this is the Pegasus project [2] in America. The aerodynamic design and analysis of Pegasus, a three-stage, air-launched, winged space booster, was conducted using only computational aerodynamic and fluid dynamic approaches.

There are a large range of computational techniques used for aerodynamic design and analysis, with the desired technique generally set by design, analysis and cost requirements. Stability and control studies for systems whose behaviour are not well understood, will generally require numerous aerodynamic peformance estimates around the nominal design. As a consequence, compromises in computational techniques are required, leading to simplified models of the system. Computational fluid dynamics (CFD), whilst providing the most detailed picture, is often left for detailing special problems in practical design.

Aerodynamic prediction techniques for hypersonic configurations have often been based on the *Hypersonic Arbitrary Body Program* (HABP), developed by Gentry [3] in the late 1960's. The HABP analysis procedure is based on "non-interfering constant pressure finite-element analysis". The compression-expansion methods shown in Table 1, are suitable for determining the pressure profile for a hypersonic analysis of a surface configuration. HABP is considered an industry-standard, used for preliminary design and analysis of hypersonic vehicles. The following authors (by no means exhaustive) completed studies noting the applicability of HABP,

- Divan 1980 [4]
- Moore and Williams 1989 [5]
- Cruz and Wilhite 1989 [6]
- Maughmer, Ozoroski, Straussfogel and Long 1993 [7].

Table 1: Hypersonic analysis methods, HABP [4]

Impact flow	**Shadow flow**
Modified Newtonian	Newtonian (Cp = 0)
Modified Newtonian + Prandtl-Meyer	Modified Newtonian + Prandtl-Meyer
Tangent wedge	Prandtl-Meyer expansion
Tangent-wedge empirical	OSU blunt-body empirical
Tangent-cone empirical	Van Dyke unified
OSU blunt-body empirical	High Mach base pressure
Van Dyke unified	Shock expansion
Blunt-body shear force	Input shear coefficient
Shock expansion	Free molecular flow
Input pressure coefficient	
Hankey flat-surface empircal	
Delta wing empircal	
Dahlem-Buck empirical	
Blast wave	
Modified tangent cone	

Divan [4] constructed an interactive program which enabled mesh configuration and viewing to be linked to the analysis routines. Divan's *Aerodynamic Preliminary Analysis System* used the force calculation modules of HABP, providing a comprehensive program covering speeds from subsonic to hypersonic.

Moore & Williams [5] provided a selection rationale for the methods described in HABP. A range of hypersonic configurations were considerd with the focus on vehicle control analysis. Solution techniques where allocated to three basic body parts, nose, body and aerodynamic surfaces. For high hypersonic speeds ($M > 8$), Modified Newtonian was used for all impact surfaces, while all leeward surfaces incorporated Prandtl-Meyer Expansion. It was also noted that there was no significant improvement in prediction accuracy by applying a real gas approximation through the use of an effective γ (specific heat ratio). There was also little change in the overall vehicle pressure distribution when viscous methods were applied, viscous considerations being more critical for estimating vehicle drag. The prediction of lateral-directional aerodynamics may not be as accurate using the same selection rationale as that for the longitudinal aerodynamics.

Cruz & Wilhite used a modified version of HABP known as the *Aerodynamic Preliminary Analysis system (APAS)* [6], presumably similar to that developed by Divan [4], though no indication is given in the report. The aerodynamic characteristics of six simple configurations and three complex configurations were investigated, with independent inviscid and viscous solutions applied to a non-interacting finite element model. Lift and drag coefficients were compared with experimental and computational fluid dynamics (CFD) results, with good overall agreement. Viscous contributions to vehicle drag proved most important for cone based vehicles such as the winged-cone configuration. Vehicles with a lower slenderness ratio produce pressure distributions dominated by pressure drag rather than viscous effects.

Maughmer *et al* [7] reported on the accuracy and validity of the local surface inclination methods found in HABP, for predicting control forces and moments. Three hypersonic configurations were considered; the North American X-15, the Hypersonic Research Airplane (scramjet powered), and the Space shuttle orbiter. Comparisons were made with both experimental and flight test data, with each vehicle assigned analysis methods in relation to nose, body, and aerodynamic lifting surfaces. Modified Newtonian was used on all blunt surfaces such as leading and trailing edges. The other methods which dominated the inviscid portion were tangent-wedge, tangent-cone, and Prandtl-Meyer (shadow regions). It was concluded that all the longitudinal performance derivatives were of some use at the conceptual design stage. Regions which encounter flow separartion required careful consideration. The primary control derivatives for the lateral/directional case also provided reasonably results.

Other useful references of the application of Newtonian flow to the analysis of vehicles in hypersonic flow are Anderson [1] and Chavez & Schmidt [8]. Anderson dedicates a chapter to local surface inclination methods, presenting them as simple linear relationships capable of modelling the inherently nonlinear hypersonic flows. They are considered to allow rapid estimation of pressure distribution over hypersonic bodies, defined solely in terms of the local surface inclination angle. Of particular interest is the discussion of

Newton's model and how the characteristics of hypersonic flows allow for its application.

Schmidt has been extensively involved in investigating the integration between vehicle airframe and propulsion found in hypersonic vehicles. In Reference [8], the analysis procedure used in the dynamic analysis of a configuration known as the X-30 (NASP, National Aerospace Plane), is presented. Newtonian theory was used for determining the pressure distribution over the fore-body, and coupled to a one dimensional model of the scramjet engine. Two dimensional shock-expansion theory gave the pressure distribution on the afterbody/nozzle.

The review of relevent literature given here presents the idea that simple techniques can be useful for hypersonic aerodynamic analysis. Newtonian theory and other pressure methods have proven to be useful preliminary (conceptual) analysis tools, primarily due to the speed of computation. Critical aerodynamic and control issues can be identified and examined extensively. With no intermediate techniques separating these methods and full Euler or Navier -Stokes approach, a Newtonian based analysis for arbitrary bodies may be suitable for control system design. When a more accurate knowledge of aircraft performance is required for complex geometries, the more sophisticated CFD techniques must be used but at the computational cost that may be three or four orders of magnitude larger than the cost of running a Newtonian analysis.

2 Modified Newtonian Flow Theory

The modified Newtonian method is one of a number of impact/shadow flow analysis methods which are commonly used in determining the surface pressure distribution over hypersonic geometries. As a *local surface inclination method*, it neglects high temperature, viscous, and boundary layer effects. If the body surface is defined as a collection (or mesh) of locally flat surface elements, Newtonian theory can be applied independently to these elements to provide a complete vehicle pressure distribution. From this distribution, aerodynamic forces and moments can be intergrated. The only geometric parameter required to obtain the surface pressure from Newtonain theory, is the angle the freestream impacts with the mesh element. To compute aerodynamic forces and moments, the element areas and centroids are also used.

Modified Newtonian theory follows the standard Newtonian sine-squared law, with an adjustment to give the correct pressure coefficient at the stagnation point. The modification to Newtonian flow is represented here, (as it is in Reference [1]), in terms of the pressure coefficient,

$$C_p = C_{p_{max}} \sin^2 \theta \quad , \tag{1}$$

where C_p is the local surface pressure coefficient,

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} \quad . \tag{2}$$

Referring to Figure 1, p is the local surface pressure, p_∞ is the freestream static pressure and $q_\infty = \frac{1}{2}\rho_\infty V_\infty^2$ is the freestream dynamic pressure. The angle θ is the body deflection angle, the angle the surface makes with the free-stream direction. Theta takes into account the geometry as defined relative to the body-fixed axes and the angle of attack of the body in the flow. The local surface normal vector is the geometric parameter used. $C_{p_{max}}$ is the maximum value of the pressure coefficient, evaluated at the stagnation point behind a normal shock wave,

$$C_{p_{max}} = \frac{p_{o_2} - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} \quad . \tag{3}$$

The Rayleigh Pitot tube formula, derived from normal shock wave relations, enables the pressure at the stagnation point p_{o_2}, to be determined. This forces the modified Newtonian analysis to be "exact" at the stagnation point, following the ($\sin^2$) law for all other areas of the body. The stagnation pressure behind a normal shock wave, p_{o_2}, is defined by the Rayliegh Pitot tube formula as,

$$\frac{p_{o_2}}{p_\infty} = \left[\frac{(\gamma+1)^2 M_\infty^2}{4\gamma M_\infty^2 - 2(\gamma-1)}\right]^{\frac{\gamma}{\gamma-1}} \frac{1-\gamma+2\gamma M_\infty^2}{(\gamma+1)} \quad . \tag{4}$$

With $C_{p_{max}}$ evaluated for given free-stream conditions, the pressure coefficient over the geometry can be determined. Modified Newtonian theory is therefore only dependent on

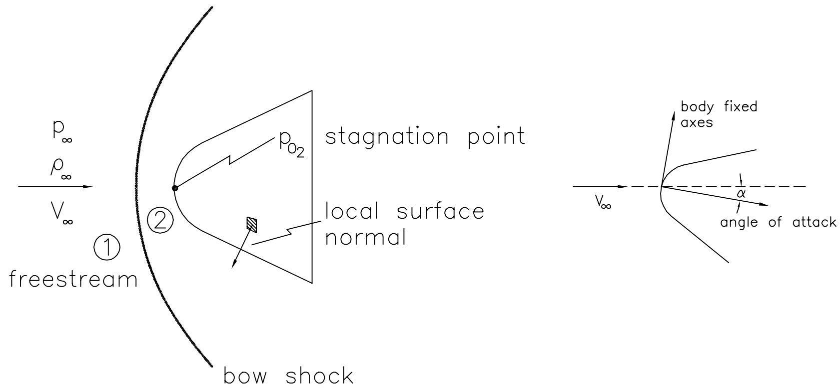

Figure 1: Nomenclature for Newtonian flow Analysis

the free-stream conditions and the geometry of the body. Combining Equations (1 - 3), the equation for the impact pressure on a mesh element can be given by,

$$\frac{p}{p_\infty} = 1 + \left(\frac{p_{O_2}}{p_\infty} - 1\right)\cos^2\eta \quad . \tag{5}$$

The reference angle η is the angle between the velocity vector and the inward normal to the surface element. Thus $\cos^2\eta$ replaces the $\sin^2(\theta)$ term in Equation 1. Once the mesh is defined, the application of Equation 5 to each element in the surface mesh, is all that is needed to determine the complete pressure profile.

2.1 Processing of Newtonian Pressure Distribution

Application of Newtonian theory to the surface mesh via Equation 5 results in an array of size n_e, storing the surface pressure for each surface element. Processing of this array with the geometric details of the mesh allow, forces, moments, centre of pressure and pressure contour data to be obtained.

2.1.1 Forces and Moments

Using the area and centroid data, forces and moments are computed. The force acting on an element results from the local surface pressure p acting over an incremental area ds in the direction of the unit normal inward $-\hat{\mathbf{n}}$.

$$\mathbf{dF} = -p\hat{\mathbf{n}}ds \tag{6}$$

The unit outward normal $\hat{\mathbf{n}} = n_x\hat{\mathbf{i}} + n_y\hat{\mathbf{j}} + n_z\hat{\mathbf{k}}$ indicates the side of the element which is exposed to the flow. The net force is thus,

$$\mathbf{F} = -\int_s p\hat{\mathbf{n}}\,ds \tag{7}$$

Represented by a discrete summation,

$$\mathbf{F} = -\sum_{i=1}^{ne} p_i n_{ix} ds_i \hat{\mathbf{i}} - \sum_{i=1}^{ne} p_i n_{iy} ds_i \hat{\mathbf{j}} - \sum_{i=1}^{ne} p_i n_{iz} ds_i \hat{\mathbf{k}} \tag{8}$$

Throughout the summation procedure, a record of the elemental forces are kept for evaluation of the net moment as,

$$\mathbf{M} = \sum_{i=1}^{ne} \mathbf{r_i} \times \mathbf{dF_i}, \tag{9}$$

where $\mathbf{r} = x\hat{\mathbf{i}} + y\hat{\mathbf{j}} + z\hat{\mathbf{k}}$ is the location of the element centroid.

2.1.2 Centre of Pressure

With the centre of pressure defined by, $\mathbf{r_c} = x_c\hat{\mathbf{i}} + y_c\hat{\mathbf{j}} + z_c\hat{\mathbf{k}}$, the force-moment relationship becomes,

$$\mathbf{M} = \mathbf{r}_c \times \mathbf{F}. \tag{10}$$

Expanded in matrix form gives,

$$\begin{bmatrix} 0 & F_z & -F_y \\ -F_z & 0 & F_x \\ F_y & -F_x & 0 \end{bmatrix} \begin{bmatrix} x_c \\ y_c \\ z_c \end{bmatrix} = \begin{bmatrix} M_x \\ M_y \\ M_z \end{bmatrix} \tag{11}$$

This in an indeterminant problem, giving no unique solution to the location of the centre of pressure but rather a line of action of the force. The location of the centre of pressure is therefore given by the intersection of the line of action of the force and the surface. To do this, an algorithm commonly used in ray tracing [9] is used to determine the windward surface element(s) through which the line of action passes. The line of action is defined by a direction vector and a location point on the line. Locating the line requires setting the dominant coordinate of the direction vector to zero and solving the force-moment relationship (11). The algorithm is split into two sections, the first is a ray/plane interesection procedure which determines if the ray intersects the plane containing the element. This is followed by a ray intersection algorithm which tests whether the plane intersection point lies within the element boundaries.

All surface intersections are printed to the screen as possible candidates for the centre of pressure. These points are then limited to those on windward surfaces. Results appended to the general input file are given as a combination of element number and intersection point. If an axial location along the body centroid is preferred then it can be derived from the location of the centre of pressure on the surface and the direction vector of the net aerodynamic force.

2.1.3 Pressure Contouring

It is often desired to have a graphical visualization of how pressure varies over the body. A pressure contouring option has been included in the code, providing a pressure value for each vertex in the mesh. The pressure value is averaged from those elements which have the vertex as a corner point. This requires scanning of the complete mesh for each vertex, significantly increasing the computation time. A mesh with 4562 vertices and 4460 elements required 37.3 seconds on a Sun workstation, to perform pressure contouring. This amounted to 90% of the total computational effort. Any vertex which is not used by at least one element has its pressure set to zero and a warning is displayed.

Two ouput options for the pressure data have also been provided, a value file and a format compatible with Tecplot. The value file matches the format of the vertex file with the number of vertices as the first data point, followed by a list of the pressure values for vertices $1, 2, \ldots, n_v$. For those using Tecplot for data visualization, a Tecplot data file can be produced.

3 General Description of Input to *NEWT*

Most of the effort required to use *NEWT* goes into the generation of the surface description in terms of a mesh of quadrilateral or triangular surface elements. This section provides a general description of this surface mesh.

3.1 Coordinate System

From a computational point of view, it is most convenient to have a coordinate system defined relative to the body geometry. The fluid properties can then be defined relative to the body-fixed system. Locating the axes at the leading edge of the body or the geometric centre are two logical possibilities.

A sample arrangement of a coordinate system is shown in Figure 2. The velocity of a flow in which the body had a zero angle of attack of the body would then be defined as $V_x\hat{\mathbf{i}} + 0\hat{\mathbf{j}} + 0\hat{\mathbf{k}}$. In this expression the unit vectors are aligned with the body-fixed axes.

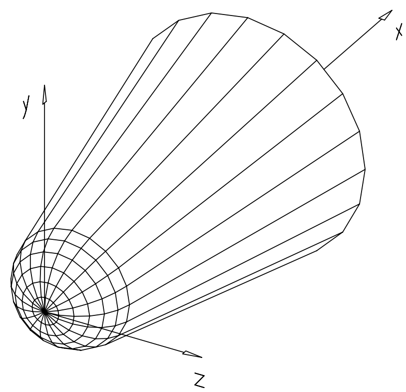

Figure 2: Body reference axes

3.2 Mesh Configuration and Development

While the coordinate system is somewhat abitrary, *NEWT* assumes certain features of the elements used to define the surface mesh. The element type has been limited to triangles and quadrilaterals. By specifying the coordinates of the mesh vertices and the arrangement of vertices into elements, a full geometric description of the mesh is obtained. The data is recorded in a *vertex file* and an *element file*, both of which must be generated prior to running the Newtonian solver.

3.2.1 Mesh Surface Definition

Vertices, numbered from 1 to n_v are read assuming the format shown in Figure 3. The first entry required in the vertex file is the total number of vertices, n_v. This is followed by the x, y, and z coordinates of each vertex.

n_v		
x(1)	y(1)	z(1)
x(2)	y(2)	z(2)
.	.	.
.	.	.
x(n_v)	y(n_v)	z(n_v)

Figure 3: Content of mesh vertex file.

Three or four vertices are used to define the perimeter of each element. Elements are numbered from 1 to n_e, with the number of elements n_e required on line 1 of the element data file, see Figure 4. Each following line defines an independent surface element made up of vertices *A, B, C,* and *D*. Each element has only one side, assumed to be exposed to the flow. Looking from the direction in which the element faces, the vertices are defined in an anticlockwise manner, as shown in Figure 5. In this case element 1 is a triangle described by vertices 1, 2, 3 and element two is a quadrilateral described by vertices 2, 6, 7, 3. The "-1" term indicates that there is no fourth vertex for element 1.

n_e			
1	2	3	-1
2	6	7	3
.	.	.	.
.	.	.	.

Figure 4: Content of mesh element file.

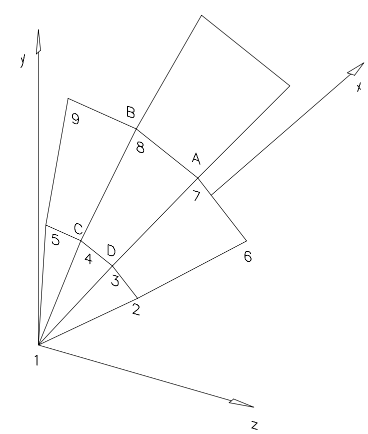

Figure 5: Sample mesh configuration

3.2.2 Element Geometry

The geometric requirements of the code are met by determining element normals, areas and centroids.

By representing each element by up to two triangles, simple vector algebra can be used. The vector product of adjacent sides of a triangular section gives both orientation and area information. The order in which the vector algebra is done requires a set method of numbering elements. *NEWT* assumes mesh elements are numbered in an anti-clockwise manner, looking at the exposed surface of the element. Therefore, the normal of an abritrary element ABCD, is approximated by averaging the normals of the triangles ABC and ACD. Adjacent sides AB & AC and AC & AD form the vectors used to determine both normals and areas. Element centroids are found by combining the area and centroid data of each triangular subelement.

NEWT does not require elements to be planar, however it is implicit in the Newtonian flow analysis as implemented in *NEWT*. Adequate representation of the surface geometry (curvature) is the responsibility of the user. If mesh generation resulted in an element being reduced to a line or point, element properties are set to zero and a warning is given.

3.3 Flow Input and Data processing

The general input file shown in Figure 6 is divided into three regions detailing velocity components, flow properties and processing options. Keywords in square brackets separate the input data and define their function. Upon execution of *NEWT* a collection of results are appended to the file.

```
Case identification
1                     no. of velocity components
5000.0 0.0 0.0       Uinfx, Uinfy, Uinfz
[properties]
300.135 1.3371       R (J/kg/K), gamma
0.009974 13378.16   rho (kg/m^3), P (Pa)
[options]
1 1             moment calculation, centre of pressure
0 1             pressure contours, output: (0)=value file or (1)=Tecplot
```

Figure 6: Input paramater file

Freestream velocity is represented in component form relative to the body fixed axes, (see Section 3.1). A table of velocity data can be placed in the input file for analysis of angle of attack or freestream velocity sensitivity. The first data value read indicates the number of velocity components listed. Under the "properties" heading, gas properties and freestream properties are defined. The gas properties required are the gas constant, R and the ratio of specific heats γ. Freestream density and pressure were chosen to define the freestream flow.

Processing options to augment the aerodynamic forces include evaluation of aerodynamic moments, centre of pressure, and pressure contours, and options for the output format of the pressure contours. Specification of 1 or 0 controls whether these options are performed (1 = yes, 0 = no). Centre of pressure is linked with moment calculation, so both must be set to one if centre of pressure is required. A check applied to pressure contouring requires that only one velocity component be listed. Details of data processing are given in Section 2.1

4 Structure of the Computer Program

This section details functions and the flow of data between them. The main processing stages can be summarised as follows.

1. *Flow Information :* Freestream conditions, fluid properties and processing options are read from the general input file detailed in Section 3.3.

2. *Mesh Input :* The vertex file and element file are read according to the format detailed in Section 3.2.

3. *Mesh Processing :* Geometric information is obtained from the vertex and element data. Element normals, areas and centroids are stored.

4. *Newtonian Analysis :* Equation 5 is applied to each element in the mesh to give the local impact pressure. Shadow flow regions defined by areas where the freestream flow does not impact with the surface, are treated as having surface pressure equivalent to the freestream.

5. *Further Processing :* The pressure distribution over the surface is combined with the geometric data to give the aerodynamic loading on the body. The default output of axial and normal forces, is appended to the general input file. Lift and drag are also given. Aerodynamic moments and centre of pressure are calculated if requested and appended to the input file. Pressure contours may also be evaluated.

4.1 Program Function Summary

Table 2 details the routines used by *NEWT*, all of which are found within the one source code file (shown in full in Appendix 6).

4.2 Program Use

A single source file *newt.f*, contains all the routines required by the Newtonian solver. Thus, compiling in a UNIX environment is simply a matter of typing the following,

```
f77 newt.f -o newt.x
```

Note, the include file *newt.h* contains declarations for matrices XYZ and ABCD and should be checked to ensure an adequate size has been specified. The following files need to be prepared prior to execution of *NEWT*.

- *name.vtx* => Mesh vertex file described in Section 3.2.
- *name.elm* => Mesh element file described in Section 3.2.
- *name.par* => Flow parameter file described in Section 3.3.

Table 2: NEWT routines

Subroutine name	Content
main	initialise variables, control processing
open_files	3 input files and 1 output file
input_par	read values defining flow and data processing
mach_no	Mach number and dynamic pressure
meshdef	group routines which define mesh properties
meshread	read vertex and element files
meshgeom	normal, area and centroid of surface elements
x_product	vector cross product
v_unit	determine unit vector
v_dot	returns vector scalar product and magnitudes
dotprod	function returns scalar product
newt_app	apply Newtonian theory
pratio_rf	pressure ratio for Rayleigh flow
p_surf	surface pressure for each mesh element
moment	calulate aerodynamic moments
cp	centre of pressure
p_vertex	generate pressure contouring file
close_files	close all files

- *flist.t* => List of files used in the order they are opened. Optional file.
- *newt.x* => Executable form of Newtonian Analysis Tool program.

Running the program in a unix environment is achieved with the command,

```
newt.x < flist.t
```

Alternatively just typing *newt.x* will run the program, with the user being asked to enter the input and output file names.

5 Sample Applications

To date *NEWT* has been applied to relatively blunt body shapes such as cones, and re-entry heat shields, and the more slender model of a scramjet. Mesh generation was achieved via customised routines.

5.1 Blunt Cone

A blunt cone of 15^o half angle (Figure 7), was investigated subject to hypersonic freestream conditions. Output collected from a number of tasks supplemented the experimental data collected by David Mee (personal communication). Figure 8 shows results for a 5^o angle of attack cone, appended to the flow parameter file. The angle of attack is given by the angles between the freestream vector and the body coordinate axes.

Other results involved pressure contouring and investigating the contributions of axial sections to the overall aerodynamic loading. The later, while not being an option within *NEWT*, was easily achieved by adapting *NEWT*, and using the centroid and pressure data already in memory.

```
sample cone specification for David Mee (1994)
15.0        cone half angle (theta_c : degrees)
220.0 0.31  sharp cone length (mm), bluntness ratio (nose/base)
360 10 1    nr, nas, nac (radial and axial divisions)

base area (m^2)       0.10917D-01
length: tip to base (mm)  0.16767D+03
hnose (mm)   13.544
hcone (mm)   154.124
```

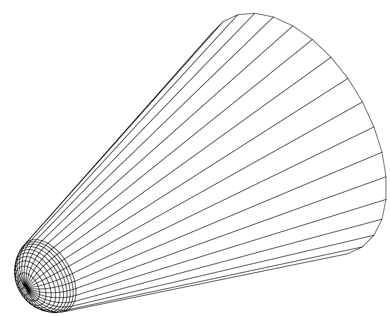

Figure 7: Blunt cone specification.

```
Shot 3745, simulating test 4213: (for David Mee)
1                          single velocity setting
4343.409 379.999 0.0   freestream: Ux,Uy,Uz  m/s
[properties]
297.2 1.32               R (J/kg/K), gamma
0.0263 11200            freestream density(kg/m^3) and pressure(Pa)
[options]
1 0                    moment calculation, centre of pressure
0 0                    pressure contours, output: (0)=value file or (1)=Tecplot

Newtonian solver ... Rev. 3.2, June 1996
---------------------------------

Uinf (m/s)  Minf  qinf(Pa):   0.43600E+04  0.58153E+01  0.24998E+06
Angle of attack (deg):    5.00   85.00   90.00

 Fx (N)   0.682736E+03
 Fy (N)   0.388736E+03
 Fz (N)  -0.631106E-12
 Drag (N)   0.714018E+03
 Lift (N)   0.327753E+03
 Mx (Nm)   0.557887E-14
 My (Nm)  -0.179613E-12
 Mz (Nm)   0.635531E+02
```

Figure 8: Simulation of Newtonian aerodynamic loading for a blunt cone.

5.2 Re-entry Heat Shield

The Apollo heat shield has been the subject of force balance experiments within the X1 (expansion tube) facility located at The Univeristy of Queensland [10]. Being a blunt body, the heat shield shown in Figure 9, is well suited to a Newtonian flow solution. The test model used has a maximum diameter of 20 mm and is split into three sections along its length. Drag measurements in a test flow of partially dissociated carbon dioxide at Mach 7.4 gave a drag coefficient of 1.56 $\pm 10\%$. The Modified Newtonian code estimated a drag coefficient of 1.50.

To further aid the experimentalist, *NEWT* is used to determine the sensitivity of forces produced on the model to angular misalignment. Variation of freestream conditions (test flow) can also provide interesting data to complement with experimental results. Note that the estimate of the freestream conditions provided the largest component of uncertainty in the experimental measurement.

5.3 Scramjet Model

One of the motivations for the development of a Newtonian based solver was its possible use in defining the aerodynamic characteristics of a three-dimensional scramjet. The scramjet design being considered is shown in Figure 10, with the cowl removed. It is an axisymmetric vehicle designed around a conical forebody with six scramjet modules

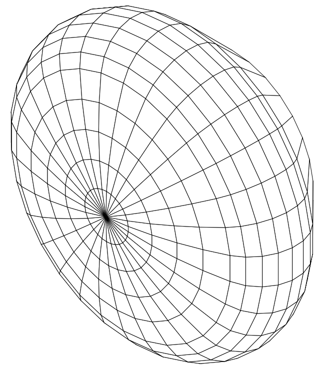

Figure 9: Surface mesh of an Apollo heat shield.

wrapped around its perimeter. At present the Newtonian solver is a stand alone program and cannot account for the three-dimensional flow processing, specifically that which occurs in the internal passages. As mentioned in Section 1.1, Newtonian flow approximations are adequate when used over windward surfaces for vehicles in the high Mach number (hypersonic) regime. The present analysis can therefore indicate the inlet drag and give an idea as to how the forces and moments depend on angle of attack. Simlilarly based flow theories can be incorporated into the code and linked to specific geometric regions.

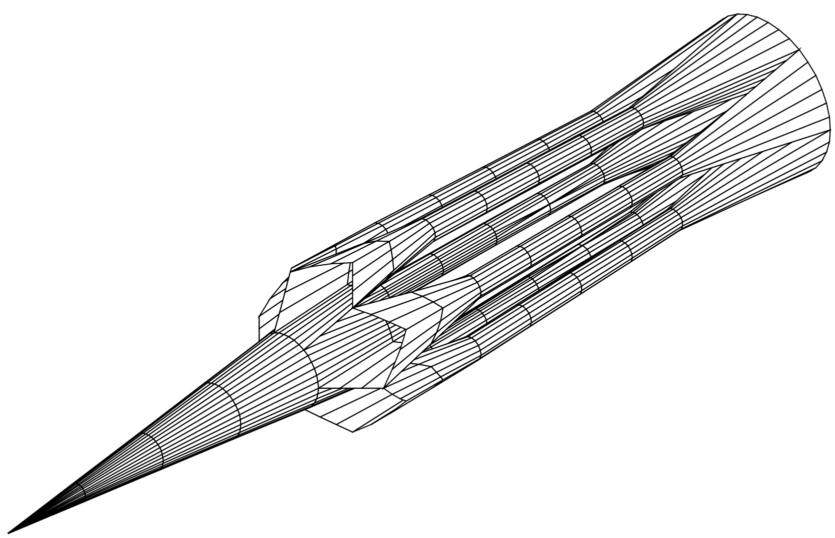

Figure 10: Surface mesh for proposed scramjet-powered stage. Cowl section, removed showing scramjet modules.

6 Concluding Remarks

An arbitrary body analysis program, *NEWT* has been developed using modified Newtonian flow theory. The flow solver has provided data confirming experimental results obtained for hypersonic flow around blunt cones and heat shields. In the process, a number of useful mesh generating routines have been developed.

The usefulness of a Newtonian flow solver was motivated by a review of aerodynamic solvers currently being used for hypersonic aircraft research. Being able to compare with experimental work within the Department, also highlighted the benefits of having a fast aerodynamic solver which is easily adapted to satisfy a range of freestream conditions and geometrical arrangements. To improve the performance of the code, future versions may offer a little more sophistication by,

- replacing the shadow flow analysis with Prandtl-Meyer expansion,
- providing tangent-cone and tangent-wedge methods for impact regions,
- providing a range of processing options to investigate the effect of each independent freestream parameter, and
- the inclusion of a viscous model.

It is envisaged that these adaptions will make the code more applicable to a wider range of configurations. The coupling of an engine model could also enable the investigation of controller design for a three dimensional scramjet. For this task it is necessary to have a fast routine representing the flow processing so simulation of the dynamics can be performed.

References

[1] J. D. Anderson. *Hypersonic and High Temperature Gas Dynamics.* McGraw-Hill, 1989.

[2] M. R. Mendenhall, S. C. Lesieutre, S. C. Caruso, F. E. Dillenius, and G. D. Kuhn. Aerodynamic design of Pegasus: Concept flight with computational fluid dynamics. *A.I.A.A. Journal of Spacecraft and Rockets*, 31(6):1007–1015, 1994.

[3] A. E. Gentry. Hypersonic arbitrary-body aerodynamic computer program (mark iii version). vol. 1-user's manual. Rep. DAC 61552, McDonnell-Douglas Corp., April 1968.

[4] P. E. Divan. Aerodynamic analysis system for conceptual and preliminary analysis from subsonic to hypersonic speeds. AIAA Paper 80-1897, 1980.

[5] M. Moore and J. Williams. Aerodynamic prediction rationale for analyses of hypersonic configurations. AIAA Paper 89-0525, 1989.

[6] C. I. Cruz and A. W. Wilhite. Prediction of high-speed aerodynamic characteristics using the aerodynamic preliminary analysis system (APAS). AIAA Paper 89-2173, 1989.

[7] M. Maughmer, L. Ozoroski, D. Straussfogel, and L. Long. Validation of engineering methods for predicting hypersonic vehicle control forces and moments. *A.I.A.A Journal of Guidance, Control, and Dynamics*, 16(4):762–769, 1993.

[8] F. R. Chavez and D. K. Schmidt. An integrated analytical Aeropropulsive/Aeroelastic model for the dynamic analysis of hypersonic vehicles. AIAA Paper 92-4567, 1992.

[9] S. Glassner, Andrew. *An Introduction to Ray Tracing / edited by Andrew S. Glassner.* Academic Press, 1989.

[10] A.L. Smith and D.J. Mee. Drag measurement in a hypervelocity expansion tube. *International Journal of Shock Waves*, 1996, Accepted.

A *NEWT* Source Code

The source code is provided here for a full description of the structure of routines. A disk containing the code, executables and sample files has also been included. There is no restriction on modifying the code.

A.1 Include File *newt.h*

```
integer PDIM,UDIM
PARAMETER(PDIM=12700)
PARAMETER(UDIM=30)
integer          ABCD(0:PDIM,4)
double precision XYZ(0:PDIM,3)
double precision Uxyz(UDIM,3)
```

A.2 Main Program *newt.f*

Begins on following page.