1 Introduction and Overview

The numerical computations of flows around complex configurations require accurate calculations of force and moment coefficients for aerodynamic surfaces. When overset grid methods (ref. 1) are used, the surfaces on which force and moment coefficients are sought typically consist of a collection of overlapping surface grids (see figure 1 for example). Direct integration of flow quantities such as pressure and viscous stress on the overlapping grids would result in the overlapping regions being counted more than once. The FOMOCO Utilities is a software package for computing flow coefficients (force, moment, mass flow rate) on a collection of overset surfaces with accurate accounting of the overlapping zones.

In this user's manual, instructions on how to run FOMOCO Utilities are given from a user's point of view. Theoretical development of the methods used are described in detail in reference 2. The equations used for computing the flow coefficients are given in Appendix A.

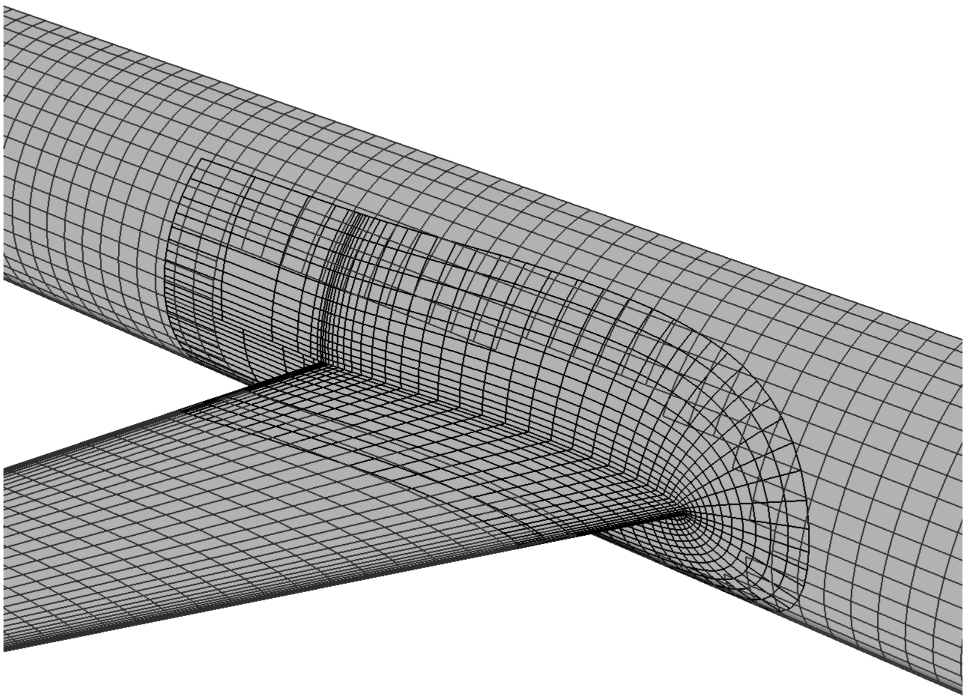


Figure 1. Overset surface grids with domain connectivity iblanks for wing/body/collar configuration (only every 3 points shown in streamwise direction for wing and collar).

The FOMOCO Utilities consist of two modules: the *MIXSUR* module and the *OVERINT* module. Both modules are written in Fortran 77. An overview of these modules is given in figure 2. The computation of flow coefficients on a set of overlapping surfaces is performed as follows.

(1) A hybrid composite surface grid consisting of non-overlapping quadrilaterals and triangles is generated using the *MIXSUR* module. This is executed by typing

mixsur < *input parameters filename* > *output information filename*

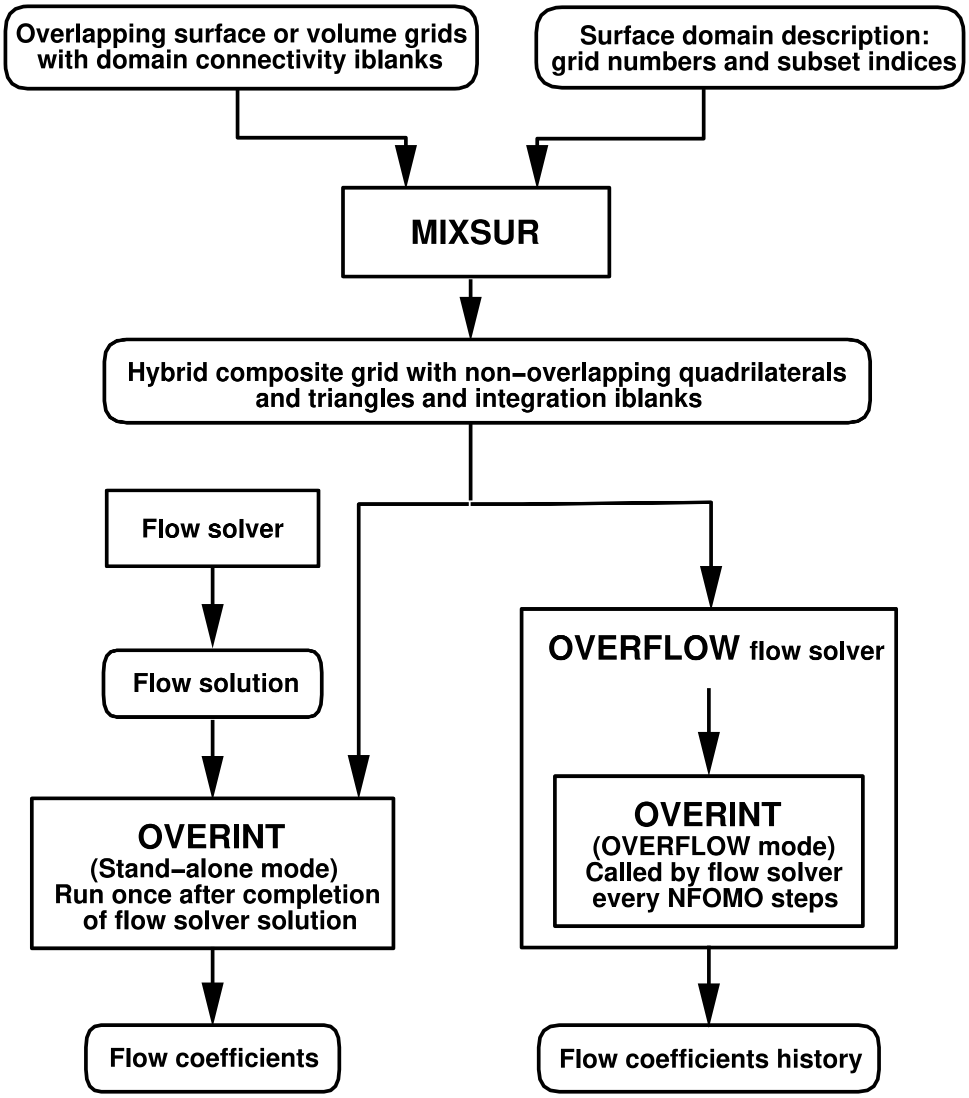


Figure 2. Overview of FOMOCO Utilities and its modules.

(2) Flow coefficients are computed by integrating flow quantities on the hybrid surface using the *OVERINT* module. *OVERINT* can be used in stand-alone mode or in conjunction with the Chimera overset grid compressible Navier-Stokes flow solver *OVERFLOW* (ref. 3). In stand-alone mode, *OVERINT* is executed by typing

overint < *input parameters filename* > *output information filename*

In *OVERFLOW* mode, *OVERINT* is performed inside *OVERFLOW*. By setting the global parameter NFOMO to a positive integer in the *OVERFLOW* input parameters file, the flow coefficients are computed and reported every NFOMO steps.

The files required to run FOMOCO Utilities are described in the subsequent sections. These include an input parameters file and the grid and solution files. Users who are not familiar with the *PLOT3D* (ref. 4) or *OVERFLOW* grid/solution file formats can refer to Appendix B for details. The same input parameters file is used to run both *MIXSUR* and *OVERINT* in stand-alone mode; and is described first in section 2. The *MIXSUR* and *OVERINT* modules are described in section 3 and section 4, respectively. Concluding remarks are given in section 5.

2 The Input Parameters File

The same input parameters file is used for both *MIXSUR* and *OVERINT* in stand-alone mode. These parameters are used to

(1) prescribe the flow conditions and the reference conditions used for flow coefficient computations, and

(2) provide a description of the surface domain on which flow coefficient computations are to be performed.

A sample input parameters file is given in table 1(a) and brief descriptions of the parameters are given in table 1(b). More detailed discussions on the input parameters are provided in sections 2.1 and 2.2.

2.1 Specification of Flow and Reference Conditions

The flow conditions and reference conditions are specified in the first part of the input parameters file. The user should be careful in selecting the ALPHA (angle of attack) and BETA (side slip angle) parameters which are only used to transform the Cartesian force coefficients into lift, drag, and side force coefficients (see eq. (4.3) in section 4.3).

For typical computational fluid dynamics (CFD) calculations, body axes are chosen such that the X-axis points nose to tail, the Y-axis points root to tip on the right wing, and the Z-axis points up. Let V_∞ and $(u_\infty, v_\infty, w_\infty)$ be the free stream velocity magnitude and Cartesian components of free stream velocity, respectively. Then the angle of attack α and the side slip angle β are defined as

$$\alpha = \tan^{-1}\left(\frac{w_\infty}{u_\infty}\right), \qquad \beta = \sin^{-1}\left(\frac{-v_\infty}{V_\infty}\right) \tag{2.1}$$

In cases where the body axes are aligned with the Cartesian axes as described above, the definition of α and β in eq. (2.1) results in the conventional lift, drag and side force coefficients. If the body axes are not aligned with the Cartesian axes, then the user must interpret the lift, drag and side force coefficients appropriately based on the ALPHA and BETA used in the input parameters file. In any case, the user can transform the Cartesian coefficients into coefficients under any other coordinate system.

Different sets of reference conditions can be specified for each integration surface and component defined by the user (see section 2.2 for explanation of integration surface and component). The main point to note is that the reference condition parameters REFL, REFA, XMC, YMC, and ZMC should be in the same units as the input grid.

Table 1(a). Sample input parameters file.

```
0.8, 2.0, 0.0, 1.67e5, 1.4, 576.0      FSMACH, ALPHA, BETA, REY, GAMINF, TINF
2                                      NREF
1.0, 871.8, 55.0, 0.0, 0.0             REFL, REFA, XMC, YMC, ZMC  } *
1.0, 168.4, 55.0, 9.0, 0.0                                        } *
3                                      NSURF

3, 1                                   NSUB, IREFS                         ]
1, 3, 1, -1, 2, -2, 1, 1               NG, IBDIR, JS, JE, KS, KE, LS, LE   ]
3, 3, 1, -1, 1, 14, 1, 1                                                   ]
4, 2, 4, -1, 1, 1, 1,-1                                                    ]  &
2                                      NPRI                                ]
2, 1                                   NU1, NU2 } #                        ]
3, 1                                            } #                        ]

2, 2                                   NSUB, IREFS                         ]
2, 3, 19, 101, 1, -1, 1, 1             NG, IBDIR, JS, JE, KS, KE, LS, LE   ]  &
3, 3, 19, 101, 14, -1, 1, 1                                                ]
0                                      NPRI                                ]

2, 2                                   NSUB, IREFS                         ]
2, 3, 101, -19, 1, -1, 1, 1            NG, IBDIR, JS, JE, KS, KE, LS, LE   ]  &
3, 3, 101, -19, 14, -1, 1, 1                                               ]
0                                      NPRI                                ]

2                                      NCOMP

WING                                                  ]
2, 2                                   NIS, IREFC     ]  @
2, 3                                   ISC(N) %       ]

TOTAL                                                 ]
3, 1                                   NIS, IREFC     ]  @
1, 2, 3                                ISC(N) %       ]
```

* Number of lines specified by NREF
& Number of blocks specified by NSURF
\# Number of lines specified by NPRI
@ Number of blocks specified by NCOMP
% Provide NIS surface numbers

Table 1(b). Description of input parameters. The name of the solution file is *q.save* (see section 4.1). Parameters indicated by * should be in same units as input grid.

FSMACH	> 0	Free stream Mach number
	≤ 0	Read free stream Mach number from *q.save*
ALPHA in range [-360,360]		Angle of attack in degrees
outside of [-360,360]		Read angle of attack from *q.save*
BETA in range [-360,360]		Side slip angle in degrees
outside of [-360,360]		Read side slip angle from *q.save*
REY	> 0	Reynolds number per grid unit for viscous flow
	$= 0$	Inviscid flow assumed
	< 0	Read Reynolds number from *q.save*
GAMINF	> 0	Free stream ratio of specific heats
	≤ 0	Read free stream ratio of specific heats from *q.save*
TINF	> 0	Free stream temperature in Rankine
	≤ 0	Read free stream temperature in Rankine from *q.save*
NREF		Number of sets of reference quantities
REFL	$\neq 0$	Reference length for flow coefficients *
	$= 0$	Use 1.0 as default reference length
REFA	$\neq 0$	Reference area for flow coefficients *
	$= 0$	Use total integrated surface area from a component in which surface belongs, or if surface does not belong to any component, use area of surface
XMC,YMC,ZMC		X,Y,Z coordinates of moment axes center *
NSURF		Number of surfaces to compute coefficients
NSUB		Number of subsets that belong to the surface
IREFS		Reference quantity set number for surface
NG		Grid number
IBDIR		Direction of surface normal ($1 = +$J, $-1 = -$J, $2 = +$K, $-2 = -$K, $3 = +$L, $-3 = -$L)
JS,JE,KS,KE,LS,LE		Start and end indices in J, K, and L, respectively. If an input index is negative, (IMAX+1) is added to form a positive index where IMAX is the grid dimension in I (I = J, K, or L), e.g., JS $= -1$ is the same as JS = JMAX.
NPRI		Number of subset pairs to specify priority
NU1,NU2		Subset numbers within surface where subset NU1 will be kept if it overlaps subset NU2
NCOMP		Number of components (can be omitted if no component)
CNAME		Component name
NIS		Number of integration surfaces for this component
IREFS		Reference quantity set number for component
ISC(N)		List of surface numbers of integration surfaces for this component

2.2 Specification of Surface Domain

The surface domain on which flow coefficients are computed is specified in the input parameters file. A complex surface domain is typically divided into one or more 'integration surfaces' where each integration surface consists of a collection of subsets from different grids in the input grid file *grid.in* (see section 3). Each subset can be any part of a J, K, or L=constant slice in the computational domain of any grid. While the flow solution is assumed to be node based, the integration scheme is cell (face) centered. For this reason, the indices used to specify the subsets (JS,JE,KS,KE,LS,LE) should include the end points of the region to be integrated. Grids in the input grid file that do not contain any of the specified subsets are disregarded.

It is frequently convenient to group a collection of integration surfaces together to form a 'component'. Flow coefficients for each component and each integration surface are computed by the integration module *OVERINT*. Each component and each integration surface is allowed to have a different set of reference quantities (length, area and moment center). The hierachy of surface domain, components, integration surfaces and subsets is shown in figure 3. Note that an integration surface may belong to more than one component although this is not typically the case.

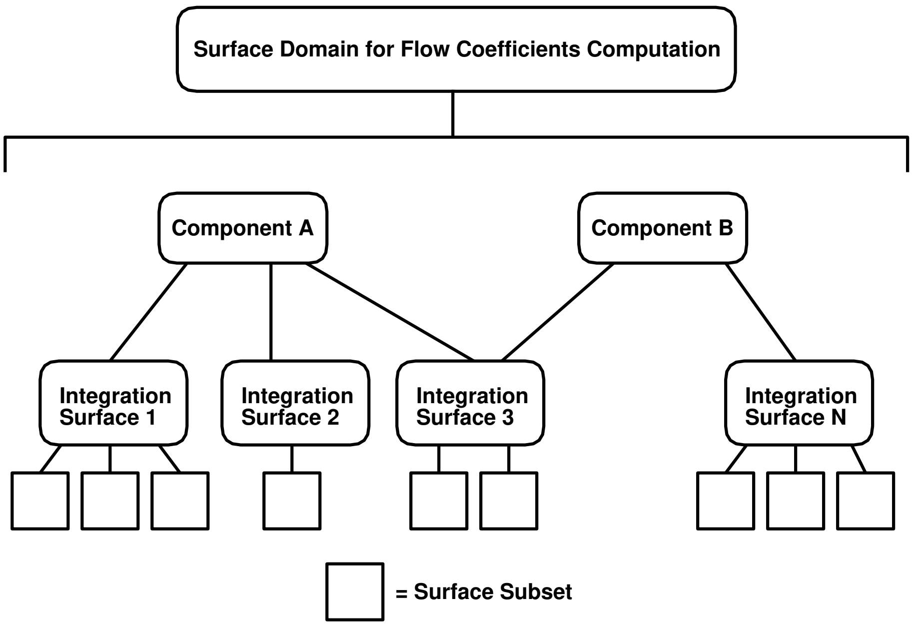


Figure 3. Hierachy of surface domain, components, integration surfaces, and subsets.

For example, a wing/body surface domain can be divided into an upper wing integration surface, a lower wing integration surface and a body integration surface where each integration surface consists of several subsets from different grids. The upper wing and lower wing may be grouped into a 'wing component'.

In the overlapped zone between two subsets from the same integration surface, the code automatically determines which subset has fewer points in the overlapped zone. Points from this subset are then blanked in the overlapped zone. The user has the option to override this scheme by specifying priorities for pairs of subsets. This is achieved by making NPRI > 0, and then providing two subset numbers for each pair (see table 1(a)). Points from the second subset will be blanked if they are in the overlapped zone with the first subset (see section 3.3 for an example).

Triangles are automatically constructed in the gap produced between neighboring subsets after the blanking step. These triangles are typically one to two layers deep (zipper grids). The blanking of points and construction of triangles are not considered between subsets belonging to different integration surfaces even if the subsets overlapped or abutted. Hence, if the user does not wish to have triangles created between two abutting subsets, the two abutting subsets should be defined under two different integration surfaces.

The surface subsets specified in the input parameters file may belong to solid surfaces (no flow-through) or field surfaces (flow-through). For example, the surface domain for a nacelle calculation may include the nacelle walls (solid) and the inlet and outlet surfaces (field). For a solid surface, the momentum flux and mass flow rate through the surface are zero.

3. The MIXSUR Module

The *MIXSUR* module is used to create a hybrid composite grid consisting of non-overlapping quadrilaterals and triangles on which flow quantities are integrated. The hybrid grid is automatically generated from the information contained in the input files described in section 3.1 below. This procedure can be performed prior to computing the flow solution since only the volume grids with domain connectivity iblanks are required as input.

The user is urged to use the automatically generated *PLOT3D* (ref. 4) command files to view and check the integrity of the composite hybrid grid before proceeding to the next step (see section 3.2). Note that a comparable grid resolution in the overlapped region between grids can greatly enhance the success of the triangulation of the region. Moreover, this property is highly desirable for accurate information transfer between grids.

3.1 Input and Output Files

The *MIXSUR* module requires two input files. The first is an input parameters file (see section 2) which describes the surface domain on which flow coefficients are to be computed. The second is a *PLOT3D* single or multiple volume grid file with domain connectivity iblanks. This file should be named *grid.in* and is identical to the input file required by the flow solver *OVERFLOW*. The user should be aware of the following two points on the *grid.in* file.

(1) For viscous computations, an alternative to supplying the entire volume grid for each grid is to provide a volume grid with two surface slices: the grid surface

on which flow coefficients are to be computed and the neighboring grid surface in the normal direction. If only inviscid flow is considered, an alternative to supplying the volume grids is to provide just the grid surfaces on which flow coefficients are to be computed.

(2) The domain connectivity iblanks in *grid.in* are only used to determine the blanked cells on the surface domain. The *MIXSUR* code has its own methods to compute grid connectivity and does not rely on the connectivity information contained in the iblanks in *grid.in*. If *grid.in* does not contain iblanks, *MIXSUR* will assume a value of 1 for all the domain connectivity iblanks.

Several output files are written by *MIXSUR*. Some of these files are used as input to the *OVERINT* module while others are used for visualization of the hybrid grid produced by *MIXSUR*. Summaries of the input and output files are given in tables 2(a) and 2(b).

Table 2(a). Input files required by *MIXSUR*.

Filename	Description
STDIN	Input parameters in ASCII format. Grids not specified in this file but in *grid.in* are disregarded by *MIXSUR* and *OVERINT*.
grid.in	Input grids in unformatted *PLOT3D* volume grid file with domain connectivity iblanks (single or multiple grid format).

Table 2(b). Output files produced by *MIXSUR*.

Filename	Description
mixsur.fmp	ASCII file containing parameters of *MIXSUR* that will be read by the flow solver *OVERFLOW* for running *OVERINT*.
grid.ibi	Surface subsets in unformatted *PLOT3D* grid file with integration iblanks (single or multiple grid format). Each grid is a surface grid defined by the J, K or L surface subsets in the input parameters file. The integration iblanks indicate which cells are to be integrated.
grid.nsf	Surface subsets in unformatted *PLOT3D* grid file (single or multiple grid format). Each grid is a surface grid next to the corresponding grids in *grid.ibi* in the user-defined normal direction. This file is generated if *grid.in* contains more than a single slice in the first grid.
grid.bnd	Unformatted *PLOT3D* multiple grid file. Each grid is a 1-D string of points marking the boundary of the region to be triangulated.
grid.tri	Unformatted *PLOT3D* multiple grid file. Each grid is a 1-D string of points marking the corners of each triangle in the triangulated region.*
grid.nor	Unformatted *PLOT3D* multiple grid fake solution file. The 2nd, 3rd and 4th component of each solution contains the x,y,z components of the normals of the triangular cells in *grid.tri*.*
grid.ptv	File containing pointers to each vertex of each triangle.*

* Generated only if triangles are needed.

3.2 Grid Visualization

The hybrid composite grid can be viewed using *PLOT3D* on the files *grid.ibi* and *grid.tri*. Table 3 shows the *PLOT3D* command files that are automatically written out. Some example plots using these command files are given in section 3.3.

Table 3. *PLOT3D* command files generated by *MIXSUR*.

Filename	Description
ib.com	Surface subsets with integration iblanks (quadrilaterals)
bnds.com	Gap boundary strings
zips.com	Zipper grids (triangles)
zn.com	Normals for zipper grids
nowalls.com	Remove all walls from any of above
minmax.com	Bounding box for all grids

To view the hybrid composite grid followed by normals of the zipper grids:

(1) Type @ib at the *PLOT3D* prompt. Type return after the plot is up.

(2) Type @zips at the *PLOT3D* prompt. Type return after the plot is up.

(3) Type @zn at the *PLOT3D* prompt.

To view the boundary strings of the triangulated regions:

(1) Type @ib at the *PLOT3D* prompt. Type return after the plot is up.

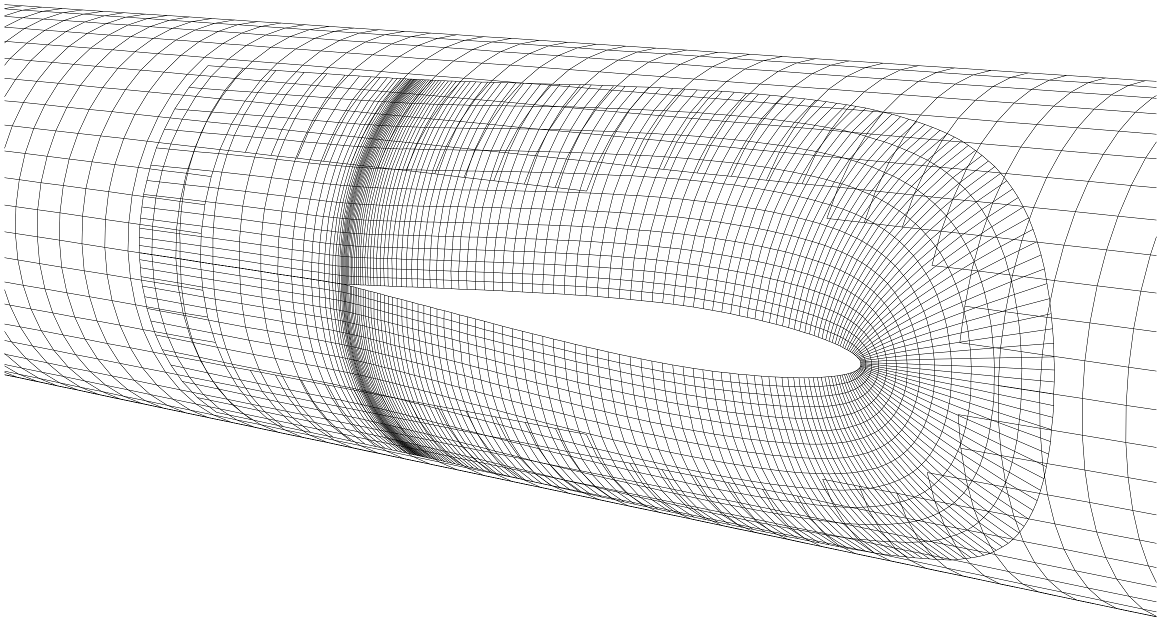
(2) Type @bnds at the *PLOT3D* prompt.

3.3 Examples

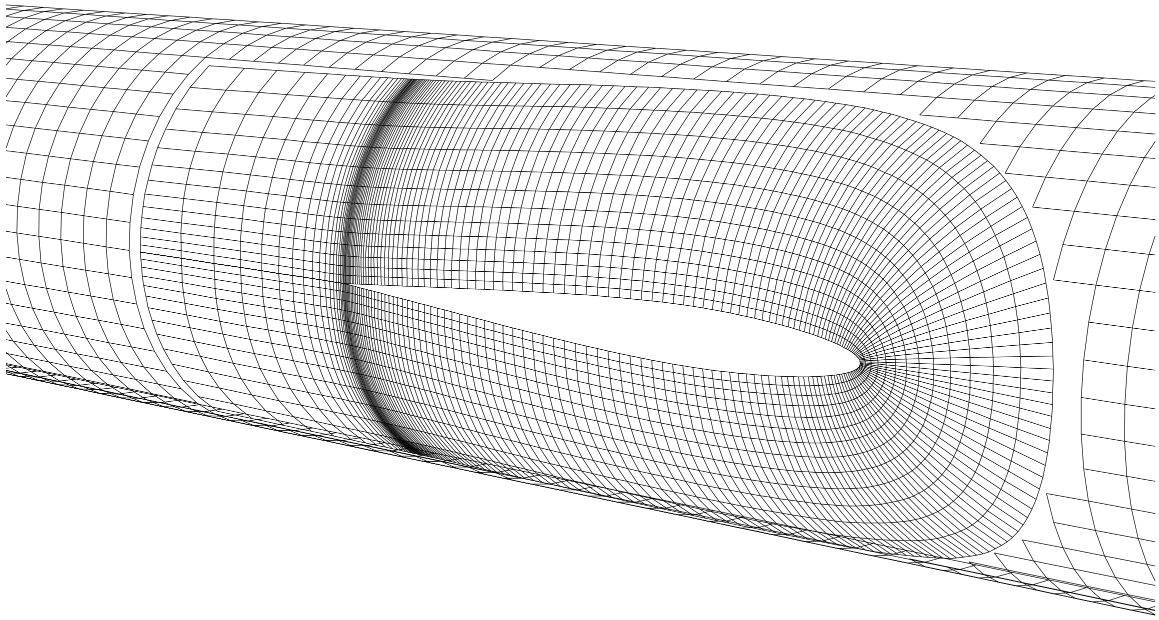
A simple configuration consisting of three grids is considered: the body, the wing and the collar (ref. 5). The overlapping surface grids on which flow coefficients are computed are shown in figure 1. Figure 4(a) shows the overlapping surface grids on the body/collar junction that are used as input to *MIXSUR*. Note that part of the body grid has been blanked out due to the presence of the wing. This blanking information is typically generated by a domain connectivity program such as PEGSUS (ref. 6) or DCF3D (ref. 7) and is stored in the *grid.in* file.

Figure 4(b) shows the surface grids after the integration iblank array has been constructed by *MIXSUR*. Gaps are created between neighboring grids while the remaining unblanked points belong to quadrilaterals that do not overlap. This view can be generated by using the *ib.com* command file in *PLOT3D*.

Points along the boundaries of the gaps are identified and connected to form gap boundary strings. These strings can be viewed by using the *bnds.com* command file in *PLOT3D* as shown in figure 4(c).

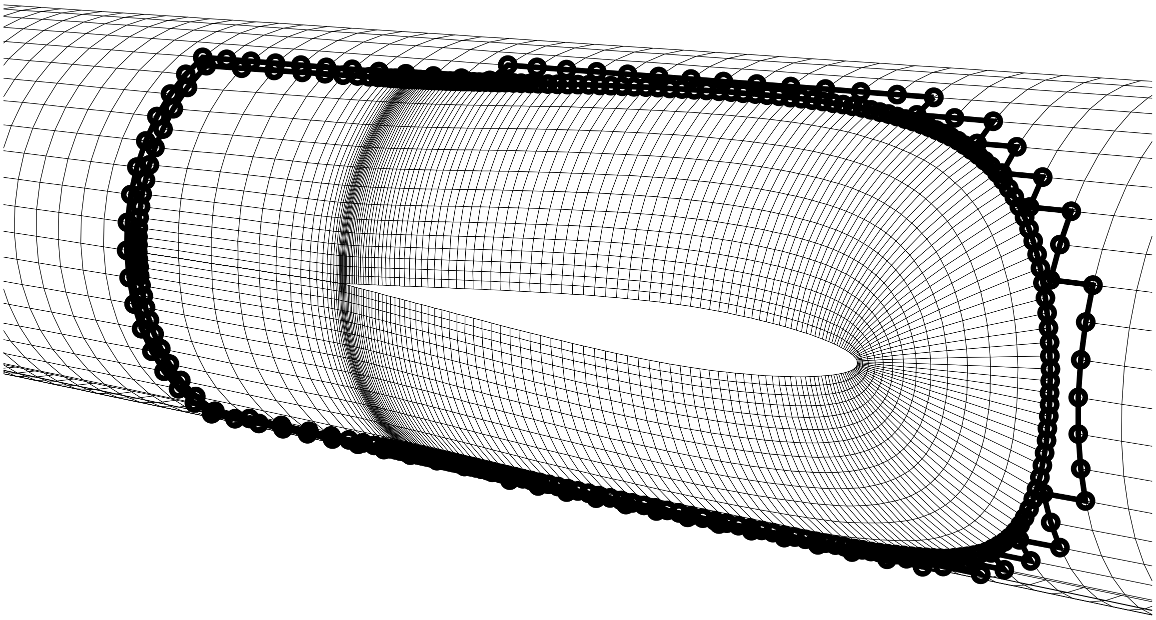


(a)

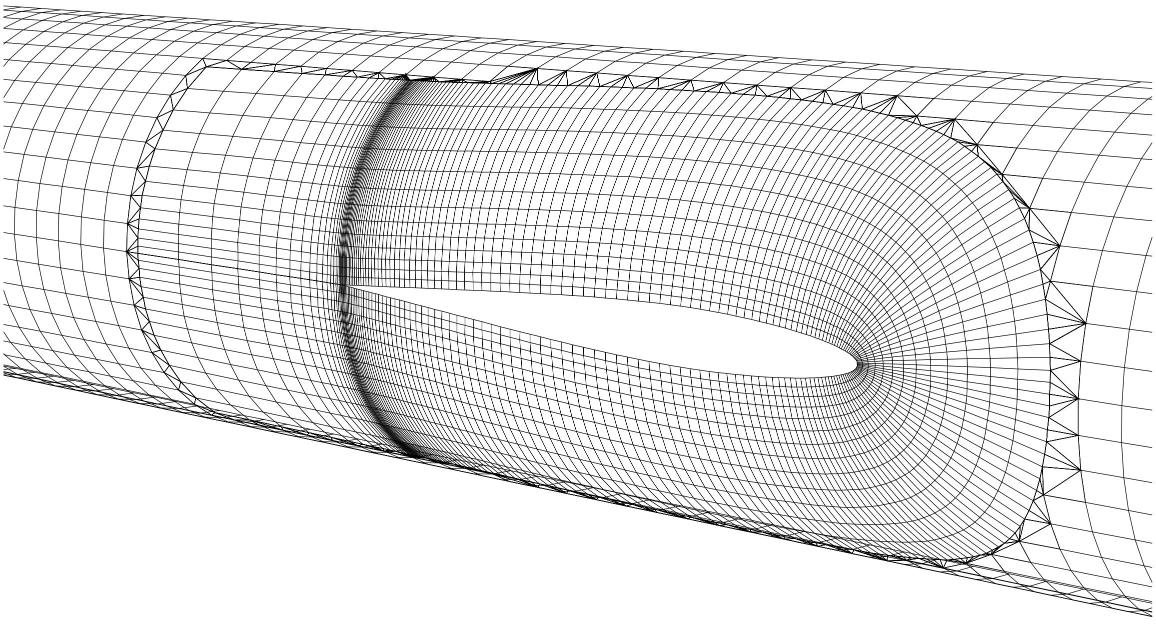


(b)

Figure 4. Body/collar junction. (a) Overlapping surface grids with domain connectivity iblanks, (b) surface grids with integration iblanks.

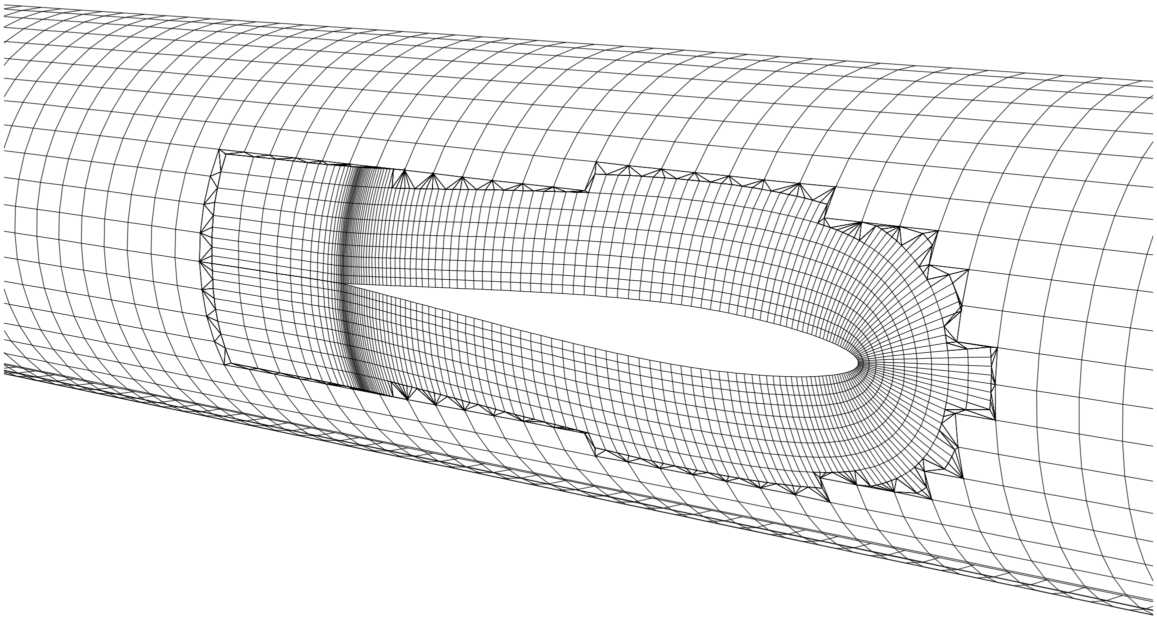


(c)



(d)

Figure 4. Body/collar junction. (c) Gap boundary strings, (d) hybrid composite surface grid with non-overlapping quadrilaterals and triangles. Collar has higher priority than body (automatically determined by default).



(e)

Figure 4. Body/collar junction. (e) Hybrid composite surface grid where the body subset is manually given higher priority over the collar subset.

Finally, triangles are constructed to fill the gaps. The resulting hybrid grid can be viewed by using *ib.com* followed by *zips.com* in *PLOT3D* as shown in figure 4(d). With this view up in *PLOT3D*, the normals of the triangular cells created can be checked by using the *zn.com* command file. Since the *MIXSUR* code is known to produce some bad triangles in difficult cases, the user is strongly urged to use these *PLOT3D* command files to check the integrity of the hybrid grid created prior to running the *OVERINT* module.

The hybrid grid in figure 4(d) is generated based on the default blanking priority, i.e. points from the coarser of two overlapping subsets are blanked. In this case, points from the body are blanked in the overlapped zone by the default scheme. The default priority may be manually overidden by the user for reasons discussed in paragraph (c) in section 3.4. In this example, the priority of the body subset can be made to be higher than that of the collar subset. The resulting hybrid grid is shown in figure 4(e).

An example of a complex configuration is demonstrated by the External Tank (ET) of the Space Shuttle Launch Vehicle as shown in figure 5. The view is looking downstream from the interior of the ET base region. The ET surface grid is overlapped by the collar surface grids from the various feedlines that intersect the ET. Further examples of complex configurations can be found in reference 2.

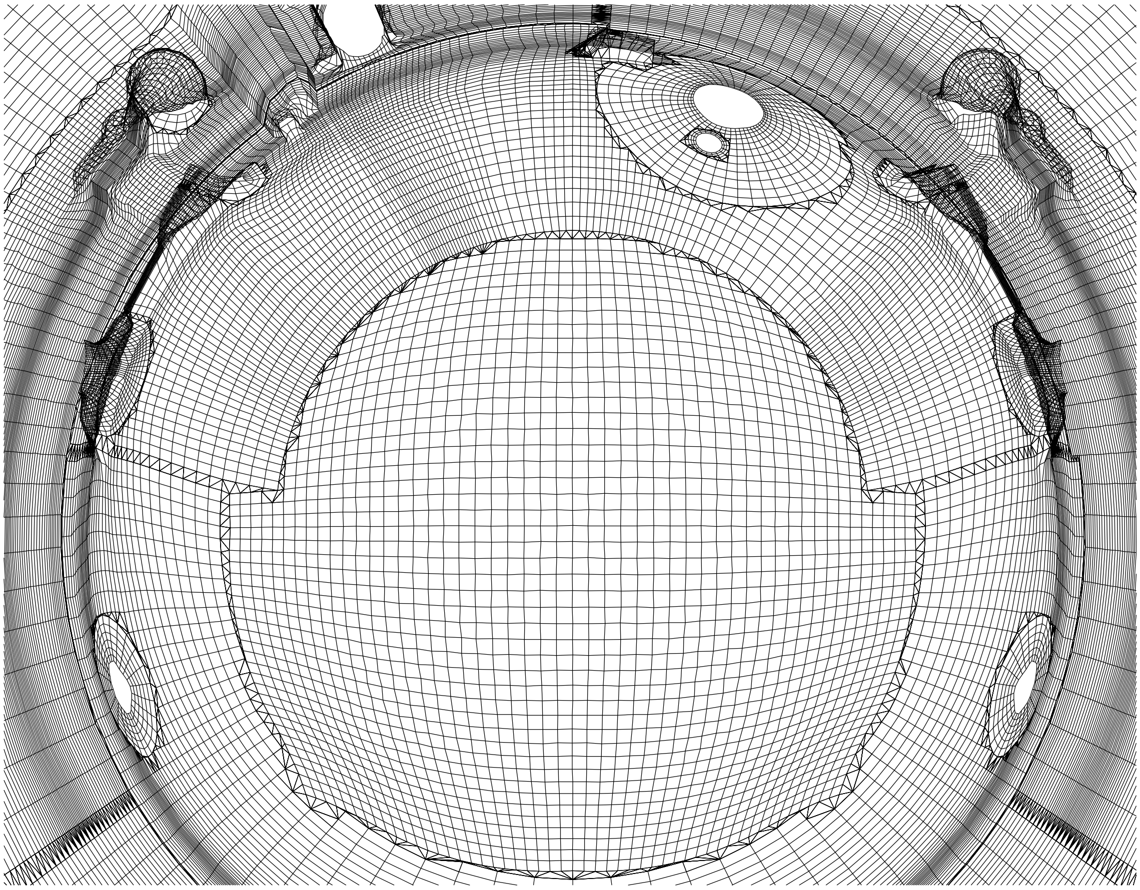


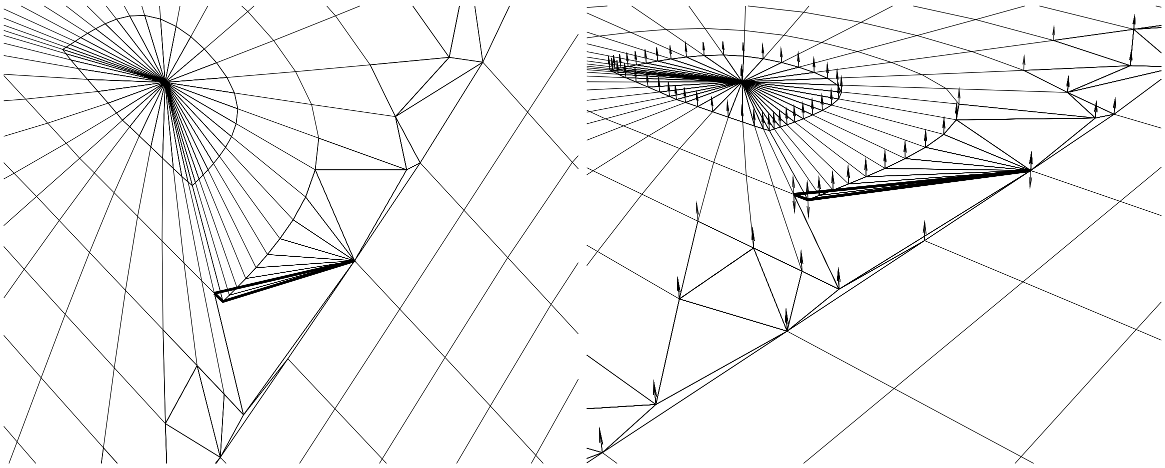
Figure 5. Hybrid composite grid for the base region of the External Tank.

3.4 Special Notes on Using MIXSUR

Given the input grids and the specification of the overlapping surface subsets, the hybrid composite grid is automatically generated by *MIXSUR*. High quality triangles are usually created when the grid resolutions of neighboring grids are comparable in the overlapped zones, and when the surface geometry is sufficiently resolved by the grid. However, when there is a large discrepancy in grid resolution between neighboring grids in the overlapped zones, the triangles created are not guaranteed to be totally correct. A triangle is considered 'bad' if it overlaps an unblanked quadrilateral. Its surface normal is pointed in the opposite direction to neighboring good triangles.

Figures 6(a) and 6(b) show the hybrid grid between two overlapping subsets and the surface normals of each triangle. For each triangle, the normals are plotted at the vertices (i.e., three arrows pointing in the same direction are plotted for each triangle). A bad triangle is produced in the region of high grid resolution mismatch. Its surface normal is pointing in the opposite direction to its good neighbors.

Also shown in figure 6 is a singular axis point in one of the subsets. Such a point is surrounded by triangular cells and may be found at a domain boundary in a structured quadrilateral mesh. Singular axis points are automatically detected by *MIXSUR* and the neighboring triangular cells are treated as a zipper grid, i.e., *OVERINT* will use the integration scheme for triangles on these cells.



(a) (b)

Figure 6. Hybrid composite grid for two overlapping subsets. A bad triangle indicated by thick lines is shown in the region of high grid resolution mismatch where the normals are pointing in the wrong direction. (a) Top view, (b) isometric view with surface normals of triangles (zipper grid).

The following notes may help to reduce the number of bad cells created:

(a) A large number of surface subsets describing a complex configuration should be split into multiple integration surfaces where subsets belonging to different integration surfaces do not communicate with each other.

(b) High curvature regions or extremely sharp corners are sometimes trouble regions. For example, points on the lower surface of a wing may be connected to the upper surface near the trailing and/or leading edge. This problem can be fixed by defining two integration surfaces for the wing: an upper surface with its collection of surface subsets and a lower surface with its collection of surface subsets. Now, points on the upper and lower surfaces are not allowed to be connected to each other.

(c) The use of the default blanking priority between overlapping subsets (points from the coarser subset is blanked) can usually result in good quality triangles in the hybrid grid. However, this is not always true in cases where there is severe grid resolution mismatch in the overlapped zones. For these cases, a smoother triangulation gap boundary may sometimes be obtained by manually imposing blanking priorities using the NPRI, NU1, and NU2 parameters (see section 2). This may then result in better quality triangles. For example, the hybrid grid for the body/collar junction in figure 4d was generated using the default priorities which produced smooth gap boundaries for the triangulation. However, if the body is given priority over the collar, a more rugged gap boundary is generated as shown in figure 4(e). In this case, the triangles generated are still of good quality, but the results from using the default priorities are preferred since the gap boundaries are smoother.

(d) The blanking scheme assumes that surface grids in the overlapped zones are approximating the same surface geometry, i.e., points from any one grid in the overlapped zone should lie very close to the bilinear surface defined by the network of points from any other grid in the overlapped zone. If 'big' gaps exist between grids in the overlapped zone, some points may not be blanked and the subsequent triangulation will be bad.

4. The OVERINT Module

The *OVERINT* module is used to compute and report the force, moment, and mass flow rate coefficients on the hybrid composite grid generated by the *MIXSUR* module. There is no interpolation of the flow solution onto new grid points since none are created by *MIXSUR*.

OVERINT can be run in stand-alone mode or be called from the flow solver *OVERFLOW*. In stand-alone mode, flow coefficients at the time step level for the given solution file are reported. In *OVERFLOW* mode, a history of the flow coefficients versus time step and versus elapsed CPU time is reported in an *OVERFLOW* output file. Software utilities for extracting and plotting histories of particular coefficients are available under the *OVERFLOW* release package.

When using *OVERINT* in *OVERFLOW* mode for moving body problems, *MIXSUR* still only needs to be run once before the flow solver, provided the grids do not deform. This is because grid points on the surface grids of rigid moving components remain in constant relative positions to each other. Hence, the connections between grid points on the hybrid surface grid and the integration iblank information produced by *MIXSUR* remain unchanged during body motions, although the physical coordinates of the grid points are changing. The connection and blanking information from *MIXSUR* is read by *OVERINT* which then uses the most recent grid coordinates to compute cell areas and normals at each time step.

4.1 Input and Output Files

Five input files are required to run *OVERINT* in stand-alone mode (see table 4(a)). The first is the same input parameters file that is used to run *MIXSUR*. The next three files, *grid.ibi*, *grid.nsf*, and *grid.ptv*, are files produced by *MIXSUR*. The final file is the solution file on the grid system given in *grid.in*. This file is assumed to be named *q.save* and can be in *OVERFLOW* or *PLOT3D* solution file format. *OVERINT* automatically detects whether the solution file is in *OVERFLOW* or *PLOT3D* format and reads it appropriately. Note that an *OVERFLOW* solution file may contain more information than a standard *PLOT3D* solution file. The additional information may include parameters for species convection, dependent variables for turbulence tranport equations and species convection equations (see Appendix B for details of the formats).

Three input files are required to run *OVERINT* in *OVERFLOW* mode (see table 4(a)). The *grid.ibi* and *grid.ptv* files are the same as those used by the code in stand-alone mode. The third file needed is the *mixsur.fmp* file generated by the *MIXSUR* module.

Table 4(a). Input files required by *OVERINT*.

Filename	Description
STDIN	Input parameters file in ASCII format (same as *MIXSUR* 's input) *(s)*
mixsur.fmp	(generated by *MIXSUR*) *(o)*
grid.ibi	(generated by *MIXSUR*) *(s,o)*
grid.nsf	(generated by *MIXSUR*) *(s)*
grid.ptv	(generated by *MIXSUR*) *(s,o)*
q.save	*OVERFLOW* or *PLOT3D* solution file * *(s)*

* (single or multiple grid format)
(s) Required by stand-alone mode.
(o) Required by *OVERFLOW* mode.

A different output file is produced by *OVERINT* in stand-alone mode and *OVERFLOW* mode (see table 4(b)).

Table 4(b). Output file produced by *OVERINT*.

Mode	Filename	Description
Stand-alone	*STDOUT*	flow coefficients, integrated areas
OVERFLOW	*fomoco.out*	history of flow coefficients, integrated areas

4.2 Input Data

The dependent flow variables from the *q.save* solution file are assumed to be non-dimensionalized in the same manner as in the *OVERFLOW* flow solver (see eq. (A.3.2b) in Appendix A). If a spatially variable ratio of specific heats is given in *q.save*, that value will be used. Otherwise, the free stream value GAMINF in the input parameters file is used (see table 1).

The following parameters in the input parameters file, REFL (reference length), REFA (reference area), XMC, YMC, ZMC (moment reference center), should be provided in the same units as the input grid in *grid.in*. Similarly, the input Reynolds number REY is the Reynolds number based on unit length of the grid system. For example, if the grid system is represented in feet, REY should be the Reynolds number per foot.

4.3 Output Data

The *OVERINT* code computes projected areas, force, moment, and mass flow rate coefficients for each integration surface and component defined. Each integration surface and component can have its own set of reference quantities (length, area, moment center) from which the coefficients are computed. In stand-alone mode, coefficients for each integration surface and component are reported. In *OVERFLOW* mode, only the coefficients for each component are reported. Table 5(a) shows a sample output data file for *OVERINT* in stand-alone mode. Table 5(b) shows the pseudo code used to write the flow coefficient histories file when using *OVERINT* in *OVERFLOW* mode. The data reported include:

(1) Projected areas in the X, Y, and Z directions and the total area.
(2) X, Y, and Z force coefficients (pressure, viscous, momentum flux, and total).
(3) Lift, drag, and side force coefficients (pressure, viscous, momentum flux, and total).
(4) Moment coefficients about X (roll), Y (pitch), and Z (yaw) axes centered at (XMC, YMC, ZMC).
(5) Mass flow rate coefficient.

In stand-alone mode, the momentum flux and mass flow rate coefficients are reported only if they are non-zero; and X, Y, Z force coefficients are reported for components only.

Equations used to compute the non-dimensional force $\hat{\mathcal{F}}$, moment $\hat{\mathcal{M}}$ and mass flow rate $\dot{m}$ are given in Appendix A. The force coefficients C_f, moment coefficients C_m and mass flow rate coefficient C_ρ are obtained as

$$C_f = \frac{\hat{\mathcal{F}}}{\hat{Q}_\infty \hat{A}_{\text{ref}}}, \qquad C_m = \frac{\hat{\mathcal{M}}}{\hat{Q}_\infty \hat{L}_{\text{ref}} \hat{A}_{\text{ref}}}, \qquad C_\rho = \frac{\dot{m}}{\hat{\rho}_\infty \hat{V}_\infty \hat{A}_{\text{ref}}} \tag{4.1}$$

where

$$\hat{Q}_\infty = \frac{1}{2}\hat{\rho}_\infty \hat{V}_\infty^2 = \frac{1}{2}\hat{\rho}_\infty M_\infty^2 \hat{c}_\infty^2 \tag{4.2}$$

and $(\hat{\ })$ is used to denote non-dimensional variables (see section A3 in Appendix A); the subscript $_\infty$ is used to denote free stream quantities; ρ, V, M, and c are the density, flow speed, Mach number and sound speed, respectively; $\hat{L}_{\text{ref}}$ and $\hat{A}_{\text{ref}}$ are the reference length (REFL) and area (REFA), respectively.

The lift, drag and side forces (C_l, C_d, C_s) are computed from the X, Y, and Z components of forces (C_x, C_y, C_z) as follows.

$$\begin{aligned} C_l &= -C_x \sin\alpha + C_z \cos\alpha \\ C_d &= (C_x \cos\alpha + C_z \sin\alpha)\cos\beta - C_y \sin\beta \\ C_s &= (C_x \cos\alpha + C_z \sin\alpha)\sin\beta + C_y \cos\beta \end{aligned} \tag{4.3}$$

The roll, pitch, and yaw moment coefficients are counted positive for right hand rotations about the X, Y, and Z (body) axes, respectively.

Table 5(a). Sample output data file from *OVERINT* in stand-alone mode.

```
=====================================================
          Output from OVERINT version 1.0
=====================================================


Mach number = 0.800 Alpha =   2.000
Beta        = 0.000 Reynolds number = 0.16700E+06
Gamma_inf   = 1.400 Temperature_inf = 76.000

Reference quantities set number   1
Reference length for coefs =  0.10000E+01
Reference area for coefs   =  0.87180E+03
Moment axes center=0.55000E+02 0.00000E+00 0.00000E+00

Reference quantities set number   2
Reference length for coefs =  0.10000E+01
Reference area for coefs   =  0.16840E+03
Moment axes center=0.55000E+02 0.90000E+01 0.00000E+00

*****************************************************
Data for surface  1

 Integrated Areas
     X              Y              Z           Total
-----------------------------------------------------
-0.352421E+01  0.444506E+03  0.162479E-04 0.703312E+03

 Force coefs     Lift        Drag        Side
-----------------------------------------------------
 Pressure      0.005121    0.001412    0.029349
 Viscous      -0.000037    0.002498    0.000040
 Total         0.005085    0.003911    0.029389

 Moment coefs  Roll (X)    Pitch (Y)   Yaw (Z)
-----------------------------------------------------
               0.000131    0.170961    0.264587

*****************************************************
Data for surface  2

 Integrated Areas
     X              Y              Z           Total
-----------------------------------------------------
 0.222120E-06  0.184916E+01 -0.828426E+02 0.842213E+02

 Force coefs     Lift        Drag        Side
-----------------------------------------------------
 Pressure     -0.031813    0.003530   -0.002711
 Viscous      -0.000139    0.002181    0.000018
 Total        -0.031952    0.005711   -0.002693

 Moment coefs  Roll (X)    Pitch (Y)   Yaw (Z)
-----------------------------------------------------
              -0.008758   -0.825687    0.078427

*****************************************************
Data for surface  3

 Integrated Areas
     X              Y              Z           Total
-----------------------------------------------------
 0.263419E-06  0.184916E+01  0.828425E+02 0.842213E+02

 Force coefs     Lift        Drag        Side
-----------------------------------------------------
 Pressure      0.137826   -0.001455    0.006899
 Viscous       0.000103    0.002138   -0.000038
 Total         0.137929    0.000683    0.006861

 Moment coefs  Roll (X)    Pitch (Y)   Yaw (Z)
-----------------------------------------------------
               0.082796    3.609517   -0.166994

*****************************************************
Data for component WING

 Integrated Areas
     X              Y              Z           Total
-----------------------------------------------------
 0.485539E-06  0.369832E+01 -0.152588E-04 0.168443E+03

 Force coefs       X           Y           Z
-----------------------------------------------------
 Pressure     -0.001626    0.004188    0.106021
 Viscous       0.004318   -0.000020    0.000115
 Total         0.002691    0.004168    0.106136

 Force coefs     Lift        Drag        Side
-----------------------------------------------------
 Pressure      0.106013    0.002075    0.004188
 Viscous      -0.000035    0.004319   -0.000020
 Total         0.105978    0.006394    0.004168

 Moment coefs  Roll (X)    Pitch (Y)   Yaw (Z)
-----------------------------------------------------
               0.074037    2.783830   -0.088568

*****************************************************
Data for component TOTAL

 Integrated Areas
     X              Y              Z            Total
-----------------------------------------------------
-0.352421E+01  0.448204E+03  0.000000E+00 0.871755E+03

 Force coefs    X           Y            Z
-----------------------------------------------------
 Pressure      0.000919    0.030158    0.025647
 Viscous       0.003332    0.000036    0.000073
 Total         0.004251    0.030194    0.025720

 Force coefs     Lift        Drag        Side
-----------------------------------------------------
 Pressure      0.025599    0.001813    0.030158
 Viscous      -0.000044    0.003333    0.000036
 Total         0.025556    0.005146    0.030194

 Moment coefs  Roll (X)    Pitch (Y)   Yaw (Z)
-----------------------------------------------------
               0.198947    0.708695    0.242800
```

Table 5(b). Pseudo code used to write flow coefficient histories when using *OVERINT* in *OVERFLOW* mode.

```
DO I=1,NITER
   IF ( (MOD(I,NFOMO)=0) OR (I=NITER) ) THEN
     DO NC=1,NCOMP
       WRITE CNAME(NC)
       WRITE I,–NC,SAC(NC),TAXC(NC),TAYC(NC),TAZC(NC)
       WRITE CXPC(NC),CYPC(NC),CZPC(NC),CXVC(NC),CYVC(NC),CZVC(NC)
       WRITE CXMC(NC),CYMC(NC),CZMC(NC),CLPC(NC),CDPC(NC),CSPC(NC)
       WRITE CLVC(NC),CDVC(NC),CSVC(NC),CLMC(NC),CDMC(NC),CSMC(NC)
       WRITE CMRC(NC),CMPC(NC),CMYC(NC),CFRC(NC),TIMECP(I)
     END DO
   END IF
END DO
```

```
CNAME                 = component name
NITER                 = number of time steps in flow solver
NFOMO                   write force/moment data every NFOMO steps

I                     = time step number
NC                    = component index number

SAC,TAXC,TAYC,TAZC    = total surface area, projected X, Y, Z areas
CXPC,CYPC,CZPC        = X, Y, Z pressure force coefficients
CXVC,CYVC,CZVC        = X, Y, Z viscous force coefficients
CXMC,CYMC,CZMC        = X, Y, Z momentum flux coefficients
CLPC,CDPC,CSPC        = lift, drag, side pressure force coefficients
CLVC,CDVC,CSVC        = lift, drag, side viscous force coefficients
CLMC,CDMC,CSMC        = lift, drag, side momentum flux coefficients
CMRC,CMPC,CMYC        = roll, pitch, yaw moment coefficients
CFRC                  = mass flow rate coefficient

TIMECP                = elapsed CPU time
```

4.4 Special Notes on Using OVERINT

(1) When computing flow coefficients for a single grid, the *MIXSUR* module still needs to be run before the *OVERINT* module.

(2) The *OVERINT* code integrates $(\hat{p} - \hat{p}_\infty)$ over the defined subsets where p is the pressure. If the defined subsets form a closed surface, then the true aerodynamic force and moment coefficients are computed. If the defined subsets form an open surface, the user must decide whether to adjust the reported coefficients by the $\hat{p}_\infty$ term.

(3) The user should be aware that the answers obtained with the current flow quantity integration scheme (see Appendix A) may vary slightly depending on the following:

(a) Quadrature rule used to integrate a function over the quadrilaterals and triangles.

(b) Integration using triangles versus quadrilaterals.

(c) Size and shape of the triangulated region.

Initial results seem to indicate that the variations caused by the above factors are typically much smaller than the accuracy of the flow coefficients required.

5 Concluding Remarks

The FOMOCO Utilities are currently in beta release stage. The software has been used by a number of groups within NASA and industry. These include oblique wing with pylons, fins and nacelles (ref. 2), subsonic transport with wing, body, pylon, nacelle, core-cowl (refs. 2 and 8), Stratospheric Observatory for Infrared Astronomy (SOFIA) (ref. 9), Space Shuttle Launch vehicle (refs. 2 and 10), and the X-CRV crew return vehicle (ref. 10). Very good results have been obtained for comparisons between numerical computations and experiments.

Bug reports and comments are most appreciated and can be sent to
wchan@nas.nasa.gov

Appendix A

The equations used to compute forces, moments, and mass flow rate for a surface are described in this appendix.

A1 Surface Integrals of Flow Quantities

In general, the forces and moments acting on a solid or field surface consist of contributions from the pressure, viscous stress and momentum flux. For a solid surface, the momentum flux and mass flow rate through the surface are zero.

In Cartesian tensor notation, the force $\mathcal{F}_i$ acting on a surface A due to a moving fluid is given by

$$\mathcal{F}_i = \int_A \sigma_{ij} n_j \, dA \tag{A.1.1}$$

where σ_{ij} is the stress tensor and n_i is the local unit normal to the surface. The force Θ_i due to momentum flux through a field surface is given by

$$\Theta_i = \int_A \rho u_i u_j n_j \, dA \tag{A.1.2}$$

where ρ is the density and u_i is the velocity. The mass flow rate $\dot{m}$ through a field surface is given by

$$\dot{m} = \int_A \rho u_j n_j \, dA \tag{A.1.3}$$

A2 General Form of the Stress Tensor

The stress tensor is given by

$$\sigma_{ij} = -p\delta_{ij} + \tau_{ij} \tag{A.2.1}$$

where p is the pressure, δ_{ij} is the Kronecker delta function and τ_{ij} is the viscous stress tensor given by

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \lambda \delta_{ij} \frac{\partial u_k}{\partial x_k} \tag{A.2.2a}$$

The laminar viscosity is denoted by μ. From the Stokes hypothesis $\lambda = -\frac{2}{3}\mu$, eq. (A.2.2a) becomes

$$\tau_{ij} = \mu \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right] \tag{A.2.2b}$$

When the stress tensor is evaluated at a solid wall, the turbulent eddy viscosity can be assumed to be zero.

A3 Non-dimensional Form of the Stress Tensor

Let $(\hat{\ })$ denote non-dimensional variables and $(\ )_\infty$ denote the free stream value of the variable. A typical term in the stress tensor is given by

$$\sigma \sim -p + \mu \frac{\partial u}{\partial x} \tag{A.3.1}$$

The non-dimensionalization used in *OVERFLOW* is summarized by

$$\hat{x} = \frac{x}{L}, \quad \frac{\partial}{\partial x} = \frac{1}{L}\frac{\partial}{\partial \hat{x}} \tag{A.3.2a}$$

$$\hat{u} = \frac{u}{c_\infty}, \quad \hat{\rho} = \frac{\rho}{\rho_\infty}, \quad \hat{\mu} = \frac{\mu}{\mu_\infty}, \quad \hat{p} = \frac{p}{\rho_\infty c_\infty^2}, \quad \hat{\sigma} = \frac{\sigma}{\rho_\infty c_\infty^2} \tag{A.3.2b}$$

where L is the length scale and c is the sound speed. Substituting in eq. (A.3.1) gives

$$\rho_\infty c_\infty^2 \hat{\sigma} \sim -\rho_\infty c_\infty^2 \hat{p} + \frac{\mu_\infty \hat{\mu}}{L} \frac{\partial (c_\infty \hat{u})}{\partial \hat{x}} \tag{A.3.3a}$$

or

$$\hat{\sigma} \sim -\hat{p} + \frac{\mu_\infty}{\rho_\infty c_\infty L} \frac{\partial \hat{u}}{\partial \hat{x}} \tag{A.3.3b}$$

The Reynolds number Re and the scaled Reynolds number $\widetilde{Re}$ used internally in *OVERFLOW* are defined as follows

$$Re \equiv \frac{\rho_\infty V_\infty L}{\mu_\infty}, \qquad \widetilde{Re} \equiv \frac{\rho_\infty c_\infty L}{\mu_\infty} = \frac{Re}{M_\infty} \tag{A.3.4}$$

where M_∞ is the free stream Mach number. Using the scaled Reynolds number in eq. (A.3.4) and dropping all hats, the non-dimensional form of the stress tensor is given by

$$\sigma_{ij} = -p\delta_{ij} + \frac{\mu}{\widetilde{Re}} \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3}\delta_{ij} \frac{\partial u_k}{\partial x_k} \right] \tag{A.3.5}$$

A4 Stress Tensor in Cartesian and Generalized Coordinates

In Cartesian coordinates, the components of the non-dimensional stress tensor (following from eq. (A.3.5)) are given by

$$\begin{aligned}
\sigma_{xx} &= -p + \frac{2\mu}{3\widetilde{Re}}(2u_x - v_y - w_z) \\
\sigma_{yy} &= -p + \frac{2\mu}{3\widetilde{Re}}(2v_y - w_z - u_x) \\
\sigma_{zz} &= -p + \frac{2\mu}{3\widetilde{Re}}(2w_z - u_x - v_y) \\
\sigma_{xy} &= \frac{\mu}{\widetilde{Re}}(u_y + v_x) \\
\sigma_{yz} &= \frac{\mu}{\widetilde{Re}}(v_z + w_y) \\
\sigma_{xz} &= \frac{\mu}{\widetilde{Re}}(u_z + w_x)
\end{aligned} \tag{A.4.1}$$

The right hand sides of eq. (A.4.1) can be expressed in generalized coordinates ξ, η, and ζ using the chain rule. For example, σ_{xx} can be written as

$$\begin{aligned}\sigma_{xx} = -p + \frac{2\mu}{3\widetilde{Re}}\big[&2(u_\xi\xi_x + u_\eta\eta_x + u_\zeta\zeta_x) - (v_\xi\xi_y + v_\eta\eta_y + v_\zeta\zeta_y) \\ &- (w_\xi\xi_z + w_\eta\eta_z + w_\zeta\zeta_z)\big]\end{aligned} \tag{A.4.2}$$

Quantities on the right hand side of eq. (A.4.2) can be conveniently computed in the flow solver.

Typically, forces and moments acting on a solid wall are required. Suppose the solid wall is along a ζ = constant surface. Since all velocity components are zero at a viscous solid wall, all terms containing derivatives of velocity components in ξ and η are exactly zero in the viscous part of the stress tensor when it is evaluated at the wall. Note that no thin layer approximation is invoked in dropping the ξ and η derivative terms. The components of the non-dimensional stress tensor at the wall now become

$$\begin{aligned}\sigma_{xx} &= -p + \frac{2\mu}{\widetilde{Re}}(\zeta_x u_\zeta - m) \\ \sigma_{yy} &= -p + \frac{2\mu}{\widetilde{Re}}(\zeta_y v_\zeta - m) \\ \sigma_{zz} &= -p + \frac{2\mu}{\widetilde{Re}}(\zeta_z w_\zeta - m) \\ \sigma_{xy} &= \frac{\mu}{\widetilde{Re}}(\zeta_x v_\zeta + \zeta_y u_\zeta) \\ \sigma_{yz} &= \frac{\mu}{\widetilde{Re}}(\zeta_y w_\zeta + \zeta_z v_\zeta) \\ \sigma_{xz} &= \frac{\mu}{\widetilde{Re}}(\zeta_x w_\zeta + \zeta_z u_\zeta)\end{aligned} \tag{A.4.3a}$$

where

$$m = \frac{1}{3}(\zeta_x u_\zeta + \zeta_y v_\zeta + \zeta_z w_\zeta) \tag{A.4.3b}$$

Consider the terms involving derivatives of velocity in the tangential direction to the surface in eq. (A.4.2). There are two situations when these terms may be non-zero. The first is in the case of a field (flow-through) surface where tangential velocity gradients are not necessarily zero. The second is in the case of a solid surface in unsteady motion. In an inertial reference frame fixed in space, different points on the solid surface may have different velocities (e.g., a rotating body), thus resulting in non-zero tangential velocity gradients. In the above two cases, all derivative terms must be retained in computing the stress tensor components. These tangential derivative terms are disregarded in the current release of FOMOCO Utilities. They will be implemented in the near future.

A5 Flow Quantities Integration

The total force $\mathcal{F}_i$, moment $\mathcal{M}_i$ and mass flow rate $\dot{m}$ are computed by summing the respective elemental forces $\Delta\mathcal{F}_i$, elemental moments $\Delta\mathcal{M}_i$ and elemental mass flow rates $\Delta\dot{m}$. The elemental quantities are computed in a cell-centered scheme for each cell in the surface domain consisting of non-overlapping quadrilaterals and triangles.

The elemental force $\Delta\mathcal{F}_i$ acting on a vector area element $n_i\Delta A$ where n_i is the local unit normal at the surface is given by

$$\Delta\mathcal{F}_i = \overline{\sigma}_{ij} n_j \Delta A \tag{A.5.1}$$

where ($\overline{\ }$) denotes some averaging operator (see below). Similarly the elemental moment $\Delta\mathcal{M}_i$ about a point r_i^0 is given by

$$\Delta\mathcal{M}_i = \epsilon_{ijk}(r_j - r_j^0)\Delta F_k \tag{A.5.2}$$

where r_i is the position vector of the centroid of the local area element and ϵ_{ijk} is the Levi-Civita permutation symbol. The elemental mass flow rate $\Delta\dot{m}$ is given by

$$\Delta\dot{m} = \overline{\rho}\,\overline{u}_j n_j \Delta A \tag{A.5.3}$$

The ($\overline{\ }$) quantities above are evaluated at the cell center using a zeroth order scheme where the value at the center is taken to be the arithmetic average of the values at the vertices. Higher order schemes such as assuming a bilinear quadrilateral or triangle may be used but are not employed here. Simple numerical experiments seem to indicate that such higher order schemes are not needed for the typical accuracy required of the flow coefficients.

A6 Evaluation of Area Elements and Metrics

The local vector area element $n_i\Delta A$ for a quadrilateral is computed by one half times the cross product of its diagonals. Similarly, the local vector area for a triangle is computed by one half times the cross product of the two vectors along any two of its edges. In both cases, the normal is ensured to be outward pointing from the surface.

The metric terms $(\zeta_x, \zeta_y, \zeta_z) = \nabla\zeta$ are evaluated by

$$\nabla\zeta = n_i\Delta A/\Delta V \tag{A.6.1}$$

where ΔV is the local cell volume. This local cell volume is evaluated in a finite volume sense as in references 11 and 12. However, a symmetric evaluation independent of the index direction is used. The symmetric value is obtained by averaging the four volumes computed using the four diagonals of the hexahedral cell.

Appendix B

The various file formats used by FOMOCO Utilities are given below.

PLOT3D multiple grid XYZ file with IBLANKS

```
      READ(IUNIT) NGRID
      READ(IUNIT) (JDIM(N),KDIM(N),LDIM(N),N=1,NGRID)
      DO 10 N=1,NGRID
         READ(IUNIT) (((X(J,K,L),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N)),
     &               (((Y(J,K,L),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N)),
     &               (((Z(J,K,L),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N)),
     &               (((IBLANK(J,K,L),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N))
 10   CONTINUE
```

PLOT3D multiple grid solution file

```
      READ(IUNIT) NGRID
      READ(IUNIT) (JDIM(N),KDIM(N),LDIM(N),N=1,NGRID)
      READ(IUNIT) FSMACH,ALPHA,REY,TIME
      DO 10 N=1,NGRID
         READ(IUNIT) ((((Q(J,K,L,NV),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N)),NV=1,5)
 10   CONTINUE
```

OVERFLOW multiple grid solution file (most general form since version 1.7e)

```
      READ(IUNIT) NGRID
      READ(IUNIT) (JDIM(N),KDIM(N),LDIM(N),N=1,NGRID),NQ,NQC
      READ(IUNIT) FSMACH,ALPHA,REY,TIME,GAMINF,BETA,TINF,
     &            IGAM,HTINF,HT1,HT2,(RGAS(I),I=1,MAX(2,NQC))
      DO 10 N=1,NGRID
         READ(IUNIT) ((((Q(J,K,L,NV),J=1,JDIM(N)),K=1,KDIM(N)),L=1,LDIM(N)),NV=1,NQ)
 10   CONTINUE
```

Special Notes

(1) The number of grids is given by NGRID. The grid dimensions are given by JDIM, KDIM, and LDIM for the J, K, and L directions, respectively.

(2) Single grid format for all of the above files is recovered by omitting the first READ statement and using NGRID=1 for the rest of the READ statments.

(3) XYZ files without IBLANKS are also acceptable and are read by omitting the READ on the IBLANK array.

(4) The solution file can be in *PLOT3D* or *OVERFLOW* format. NQ is the number of Q variables. The first five Q variables contain the density, X, Y, and Z momentum and total energy per unit volume, respectively. The sixth Q variable in an *OVERFLOW* solution file contains the ratio of specific heats. All other Q variables, if any, are disregarded. If a *PLOT3D* solution file is used, the ratio of specific heats is assumed to be constant everywhere and equal to GAMINF in the input parameters file.

(5) The NQC,IGAM,HTINF,HT1,HT2, and RGAS parameters are disregarded.

(6) Solution files for earlier versions of *OVERFLOW* are also acceptable. These may not contain the BETA and TINF parameters.

References

1. Benek, J. A.; Buning, P. G.; and Steger, J. L.: A 3-D Chimera Grid Embedding Technique. AIAA Paper 85-1523, July 1985.

2. Chan, W. M.; and Buning, P. G.: Zipper Grids for Force and Moment Computation on Overset Grids. AIAA Paper 95-1681, Proceedings of the AIAA 12th Computational Fluid Dynamics Conference, San Diego, Calif., June 1995.

3. Buning, P. G.; Jespersen, D. C.; Pulliam, T. H.; Chan, W. M.; Slotnick, J. P.; Krist, S. E.; and Renze, K. J.: OVERFLOW User's Manual, Version 1.7e. NASA Ames Research Center, Moffett Field, Calif., 1996.

4. Walatka, P. P.; Buning, P. G.; Pierce, L.; and Elson, P. A.: PLOT3D User's Manual. NASA TM-101067, 1990.

5. Parks, S. J.; Buning, P. G.; Steger, J. L.; and Chan, W. M.: Collar Grids for Intersecting Geometric Components Within The Chimera Overlapped Grid Scheme. AIAA Paper 91-1587, Proceedings of the AIAA 10th Computational Fluid Dynamics Conference, Honolulu, Hawaii, June 1991.

6. Suhs, N. E.; and Tramel, R. W.: PEGSUS 4.0 User's Manual. AEDC-TR-91-8, June 1991.

7. Meakin, R.: A New Method for Establishing Intergrid Communication Among Systems of Overset Grids. AIAA Paper 91-1586, Proceedings of the AIAA 10th Computational Fluid Dynamics Conference, Honolulu, Hawaii, June 1991.

8. Cheung, S.: Aerodynamic Design: Parallel CFD and Optimization Routines. AIAA Paper 95-1748, Proceedings of the AIAA 12th Computational Fluid Dynamics Conference, San Diego, Calif., June 1995.

9. Srinivasan, G.: Flowfield and Acoustic Characteristics of Telescope Cavity in SOFIA Platform. AIAA Paper 95-1862, Proceedings of the AIAA 13th Applied Aerodynamics Conference, San Diego, June 1995.

10. Greathouse, J.; and Gomez, R.: Unpublished results on Space Shuttle Launch Vehicle and X-CRV Vehicle, NASA Johnson Space Center, 1996.

11. Kordulla, W.; and Vinokur, M.: Efficient Computation of Volume in Flow Predictions. AIAA J., vol. 21, no. 6, 1983, pp. 917–918.

12. Obayashi, S.: Free-Stream Capturing in Fluid Conservation Law for Moving Coordinates in Three Dimensions. NASA CR-177572, Jan. 1991.