AMBER 2d

USER'S GUIDE

by

A.R. Wilkinson & D.W. Zingg

UTIAS

Version 1.0, November 1993

Contents

List of Figures

Chapter 1

INTRODUCTION

1.1 OBJECTIVE

This guide is intended to provide a quick reference for the use of the UTIAS 2d Automated MultiBlock Elliptic gRid generator (AMBER2d) developed at the University of Toronto Institute for Aerospace Studies (UTIAS). It describes the components of the grid generator, the input and output, and gives some example test cases.

1.2 OVERVIEW

AMBER2d is a package that allows the automated generation of structured multiblock grids around multiple element airfoil configurations.

Based on the potential flow solution obtained using a Kennedy-Marsden panel method, block boundaries are introduced at the airfoil walls, a rectangular or C-shaped outer boundary, stagnation streamlines, and stagnation equipotential lines. Additional equipotential lines may be utilized at any leading edge(s). This is particularly handy where the leading edge radius is small and the singular point is not in the immediate vicinity. Without the use of a leading edge phi line, the resulting mesh may not be of sufficient quality. Equipotential lines may also be added at points of discontinuous slope on the airfoil surface, called *sharp points*, where the resulting grid may not be smooth. By introducing a block boundary in this way, mesh quality can be controlled more easily.

Algebraic grids are then generated in each of the resulting blocks. This can be done manually, by creating the *DSBFILE* describing the grid parameters by hand, or by utilizing the *AUTOMESH* option. The latter enables the user to describe a few basic mesh parameters instead of a minimum of ten for each block (i.e., eight spacings, two grid point numbers, and four parameters per cluster point). In the 4 element DHC WTBV 35/35 test case provided in Appendix A, 33 blocks decompose the domain. The algebraic grid generator would then require 330 grid parameters, in the absence of cluster points, to fully describe this mesh. However, by using *AUTOMESH* only 25 user specified grid parameters were needed. This number can be further reduced if the user wishes to take advantage of default grid parameters. If these prove to be satisfactory, the user need not define any grid parameters, as *AUTOMESH* will still generate the required parameter file based on intelligently chosen default settings. Had the blunt trailing edges of the four elements in this geometry been retained, 51 blocks would have been required. This would lead to a total of 510 algebraic grid parameters to be specified. Even greater flexibility is available to the *AMBER2d* user as the *AUTOMESH* generated *DSBFILE* can be edited or tailored to meet specific needs while maximizing convenience and ease of use. If this approach is taken, or if the required files are indeed ambitiously hand generated, the *AUTOCHECK* routine in the algebraic gridder will warn the user of any possible inconsistencies. More specifically, *AUTOCHECK* ensures that the spacings, grid point numbers, and any clustering points are specified consistently across block boundaries.

Finally, the algebraic grid is smoothed elliptically to produce a high quality mesh. The user also has control of the smoothing process and as a consequence, the resulting grid. By choosing appropriate values for the angle and spacing control function decay parameters, the user can dictate how the grid is smoothed. However, to fully specify the angle and spacing control functions, a total of eight inputs are required for each block. Thus for the 4 element DHC WTBV 35/35 test case shown in Appendix A, 264 individual parameters are to be specified (or 408 with blunt trailing edges). The *AUTOMESH* option provides the user with a convenient and quick way out of the time consuming task of defining these parameters manually. *AUTOMESH* allows the user to specify only 4 individual parameters based on the type of boundary. These also have default settings that have been preprogrammed based on the test cases performed to date. As such, the default values will suffice for most of the cases that a prospective user is likely to

encounter. Similarly to the grid parameter file that can also be created by *AUTOMESH*, the control function decay parameter file (*CTLFILE*) can be edited or tailored to meet the precise needs of the user.

As a consequence, AMBER2d consists of three parts :

1. the domain decomposition routine, *kmS*,
2. the algebraic grid generator, *agrid*, and
3. the multiblock elliptic smoother, *megrid*.

This is shown schematically in figure 1.1. Each of these routines will be described in turn.

When *AMBER2d* is used in conjunction with the *TORNADO* flow solver, also developed at UTIAS, the user is afforded a degree of automation that allows complex geometries to be modeled easily. This is achieved via the *CONFILE*, *GRIDFILE*, STGFILE and the standardized topology conventions used by both codes. This is shown in figure 1.1. Although, either code can be used with other packages, the degree of automation is reduced.

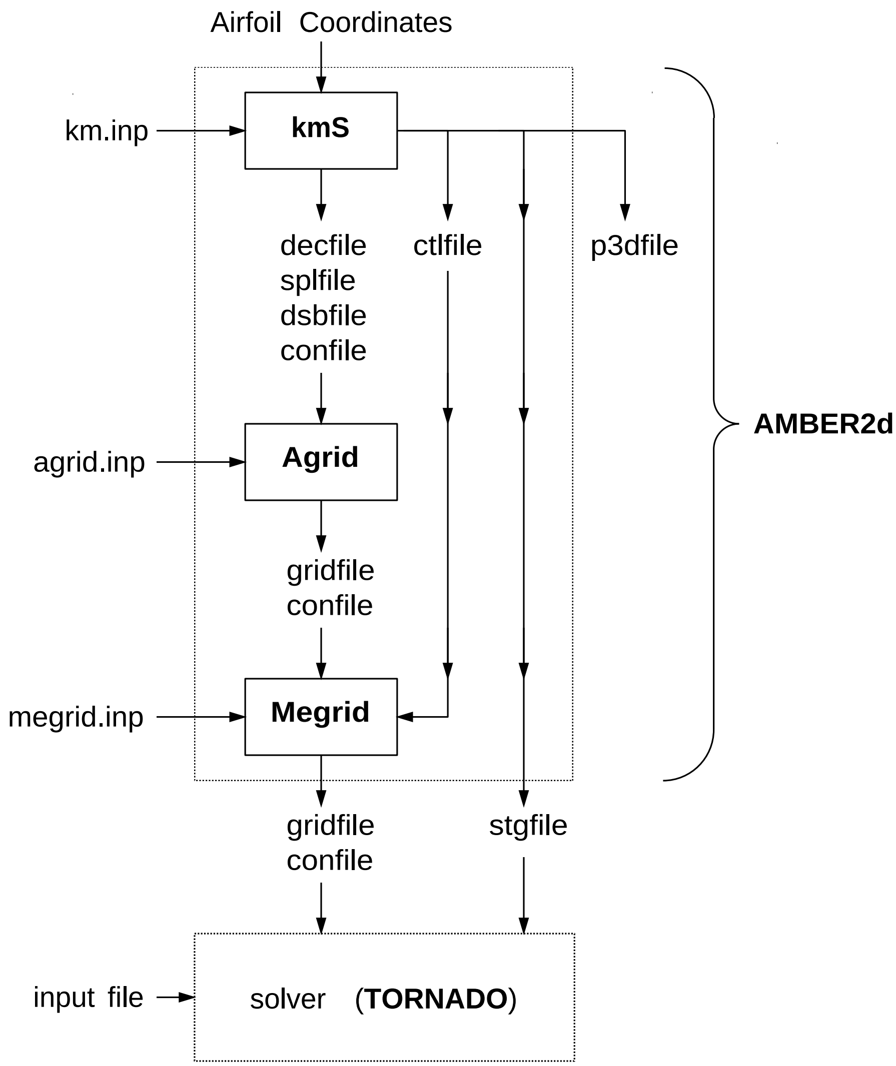

Figure 1.1: Schematic for AMBER2d and connectivity with TORNADO.

Chapter 2

DOMAIN DECOMPOSITION: 'kmS'

2.1 OVERVIEW

The domain decomposition routine, *kmS*, decomposes the domain into a series of rectangular blocks. Based on a potential flow solution obtained using a Kennedy-Marsden panel method, this is done using the airfoil element(s), a rectangular outer boundary, stagnation streamlines, and equipotential lines as block boundaries. The blocks are such that any given edge has only one boundary condition type (i.e., an airfoil, a far field boundary, or an interior boundary between blocks). This allows the flow solver to be simpler than one that allows mixed boundary conditions along the sides. The domains and, as a consequence, the grids are of the H-Mesh variety. However, the user has the option of generating C-Meshes for single element geometries.

All of the parameters in the control or input file have default settings. Thus, a decomposition can be performed by simply specifying the airfoil coordinates. The user is able to just specify the parameters that are to differ from the default values.

kmS also provides the user with the *AUTOMESH* option which greatly simplifies the process of specifying the algebraic and elliptic grids. Instead of specifying ten grid parameters and eight smoothing parameters for each block, the user need only set as few as seven grid parameters and a total of four smoothing parameters! If the user is still dissatisfied with the resulting

'first guess' grid, two options are available. The first, rather onerous, option is to edit the appropriate file(s). This allows the greatest control over the process but with the most work and requiring more know-how from the user. The second option available is to rerun the decomposition. This would, under normal circumstances, be tedious and time consuming. However, if the *DECOMP* option is set to *false*, only the grid parameter and smoothing parameter files are changed while the decomposition is unchanged. This allows the user to take advantage of the user-friendly environment of the domain decomposition routine, *kmS*, without performing the decomposition every time. The user is also provided with the number of points contained in the grid as *kmS* displays this upon execution if the *AUTOMESH* option is *active*. Some sample domain decompositions are given in the Appendices.

2.2 INPUT CONTROL FILE: kmS.inp

Some sample domain decomposition control files are shown in the Appendices and a sample showing the default settings is provided below. The contents of these files are broken up into sections by comment lines denoted by an exclamation mark '!' preceding the comment. In this file, comments may appear anywhere as long as there is a '!' on the same line as the comment and preceding the comment. Any input parameters appearing after a comment statement and on the same line will be ignored. The input type is a *NAMELIST* format (i.e., similar to *ARC2D*) whose name is *INPUT*. All entries except the airfoil coordinates have default settings. Listed below is a *kmS* input control file containing the default settings. The default for the maximum number of elements is 5, but this can be altered by changing the values of the appropriate parameters in the source code. Note that where there is no default for a particular variable, <no default> is indicated and the line is commented out:

```
!
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! DEFAULT PARAMETERS
!
! A Wilkinson
! UTIAS, November 1993
!
 &INPUT
```

```
! AIRFOIL COORDINATES :
       FOIL='airfoil.dat'         ! Airfoil coordinate file name
! CONTROL PARAMETERS :
       ALPHA  = 0.0,              ! Angle of attack
       ROTATE  = 0.0,             ! Rotation angle
       X0 = 0.5,                  ! X coordinate of rotation centrex
       Y0 = 0.0,                  ! Y coordinate of rotation center
       LEPHI  = FALSE,FALSE,      ! PHI-line from L.E. (t/f)
                FALSE,FALSE,
                FALSE,
       CMESH  = FALSE,            ! C-MESH option (t/f)
!      SHRPFL = <no default>      ! SHarP point on FoiL(s)
!      SHRPPT = <no default>      ! SHarP PoinT location(s)
       AUTOMESH = TRUE,           ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,             ! DECOMPose domain (t/f)
       MESHSEQ = 3,               ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'g.spl',         ! SPLine file (spline coefficients)
       DECFILE = 'g.dec',         ! DEComposition file (block definitions)
       CONFILE = 'g.con',         ! CONnectivity file (block connections)
       DSBFILE = 'g.dsb',         ! Define Sides of Block file
       CTLFILE = 'g.ctl',         ! ConTroL file (control fn' decay param.)
       P3DFILE = 'g.p3d',         ! Plot3D file for block boundaries
       STGFILE = 'g.stg',         ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -10.00000,           ! Left Hand Boundary location
       XRB =  10.00000,           ! Right Hand Boundary location
       YLB = -10.00000,           ! Lower Boundary location
       YUB =  10.00000,           ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 129,129,129,      ! No. PoinTS on AirFoil(s) in order from t.e.
                129,129,          !   to any cluster points to t.e. going c.w.
       NTE = 25,25,25,25,25,      ! No. points on the blunt Trailing Edge
       NFFBX = 49,                ! No. points to the Far-Field Boundary
       NFFBY = 49,                ! No. points to the Far-Field Boundary
       NINT = 49,                 ! No. vertical points IN-beTween the sections
       NGAP = 29,                 ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,2.0E-05,     ! normal off Wall spacing
             2.0E-05,2.0E-05,
             2.0E-05,
       DNFFB = 2.0,               ! normal spacing at the Far-Field Boundary
       DSSTG = 1.0E-04,1.0E-04,   ! surface spacing at the STaGnation point
               1.0E-04,1.0E-04,
```

```
              1.0E-04,
        DSTE = 5.0E-05,5.0E-05,   ! surface spacing at the Trailing Edge
               5.0E-05,5.0E-05,
               5.0E-05,
        DSLE = 1.0E-04,           ! surface spacing at any L.E. ctl line
        DSSHRP = 1.0E-04,         ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
!       CLUSFL = <no default>     ! CLUSter on FoiL # ..
!       CLUSX =  <no default>     ! norm. arc length from LE (=0.0) to TE (=1.0)!       CLUSUL =
!       CLUSDS = <no default>     ! CLUSter surface spacing (ds)
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
        SPCFFB = 0.2,             ! SPaCing ConTroL parameter for ffb
        SPCPSI = 0.01,            ! SPaCing ConTroL parameter for PSI lines
        SPCPHI = 0.02,            ! SPaCing ConTroL parameter for PHI lines
        ANGDECAY = 0.2            ! ANGle ConTroL parameter
                                &END
```

Each of the sections of this file with their variables are discussed in the following pages.

2.2.1 Airfoil Coordinates

FOIL is the name of the file, including the whole path and any extensions, containing the airfoil coordinate data. The format is described in section 2.3. The default airfoil coordinate filename is 'airfoil.dat'. Some examples of airfoil geometry data files are shown in the Appendices.

2.2.2 Control Parameters

This group of parameters controls the topology of the domain decomposition and, as a consequence, the resulting grid.

1. *ALPHA*. This is the angle of attack of the airfoil section as used by the panel method. Note that this does not alter the orientation of the sections with respect to the outer boundaries. Thus *ALPHA* allows the user to control the position of the singular point (i.e., the intersection of three block boundaries or streamlines and equipotential lines with the airfoil near the leading edge) and the angle of the stagnation streamlines in the far field. Note that *ALPHA* is defined positive nose up and is

expressed in degrees. The default value for the angle of attack is 0.0 degrees.

2. *ROTATE, X0, Y0.* This option allows the user to rotate the airfoil with respect to the far field boundaries by *ROTATE* degrees about *(X0,Y0)*. By changing the orientation, the effective angle of attack is also altered. *ROTATE* is defined in the same way as *ALPHA* and is also in degrees, while *(X0,Y0)* are defined in the airfoil coordinate system. The default for *ROTATE* is 0.0 degrees, while *(X0,Y0)* is set to (0.5,0.0).

3. *LEPHI.* If this logical flag is set to be *true*, then a phi (equipotential) line is drawn from the leading edge to the upper boundary. The direction this line travels (i.e., upward or downward) is such that it does not cross the stagnation streamline. As the stagnation streamline is not generally at the leading edge, the user is able to place a block boundary at the leading edge via a phi line. This can be helpful where the leading edge radius of a flap is small and the singular point is on the lower surface and further aft of the leading edge. By using a leading edge phi line in this case, a highly curved block at the leading edge can be avoided. The default setting is *false*.

4. *CMESH.* If this logical flag is set to *true*, then a C-Mesh domain decomposition is performed (providing that the airfoil is a single element only). In the case of this flag being inadvertently set to *true* for a multi-element configuration, it is ignored. The default setting is *false*. Note that if a C-Mesh is requested, the *LEPHI* and *SHARPPT/SHRPFL* options do not work.

5. *SHRPFL, SHRPPT.* This option is used with an airfoil with a discontinuous slope along its surface. When implemented, a phi (equipotential) line is drawn from the discontinuous point to another boundary (i.e., a far field boundary, a streamline, or another airfoil). *SHRPFL* (SHaRP FoiL) is set to the element number that the first sharp point is on, while *SHRPPT* (SHaRP PoinT) is the number index of the (x,y) pair at the sharp point in the airfoil coordinate file. Note that if element i has N sharp points then, *SHRPFL* will consist of *i,i,i,i,.....* with N occurences. The default is no sharp points.

6. *AUTOMESH*. This logical option automatically produces the files required to run the algebraic grid generator (*DSBFILE*) and the elliptic grid generator (*CTLFILE*) using data specified below. If this option is not active then it is assumed that the user is either going to produce the required files by hand or that they already exist. The default is *true*.

7. *DECOMP*. This logical flag is used in conjunction with *AUTOMESH* when only the grid details are being changed while the decomposition is unaltered. Once the domain decomposition has been performed, it will not change as long as the section, angle of attack, rotate angle, outer boundary placement, leading edge phi line(s), and sharp point(s) are the same. Thus if only the number of grid points, spacings, clustering or control decay parameters are altered, the domain decomposition does not have to be performed and *DECOMP* can be set to *false* to avoid this cpu intensive activity. The default is *true*.

8. *MESHSEQ*. If grid sequencing is to be used by a solver, the grid must be checked to ensure that the required number of sub-grids can be generated from the main grid. That is, the number of grid points plus one (i.e., $N + 1$) must be divisible by two in BOTH directions. If two levels of grid sequencing are required, then $N + 1$ must be divisible by four, and for three levels, by eight, and so on. This integer is the number of sequencing grids that can be generated from the main grid by ensuring that the number of grid points plus one ($N + 1$) in both directions, is divisible by two, *MESHSEQ* times. That is, *MESHSEQ* number of sub-grids will be able to be generated from the main grid. Note that a non-zero value for *MESHSEQ* will result in a slight change in the number of nodes as specified by the user, while a value of zero will result in no change but no guarantee of being able to produce sub-grids. The default is 3.

2.2.3 Output Files

This section specifies the names for the output files. The output files generated by *kmS* are described in detail in section 2.4.

- *SPLFILE* (SPLine FILE). SPLine points and coefficients from the domain decomposition in *TECPLOT* format (see section 6.1.1). The default is 'g.spl'.
- *DECFILE* (DEComposition FILE). Block definitions from the domain DEComposition. The default is 'g.dec'.
- *CONFILE* (CONNectivity FILE). How the blocks CONNect to one another. The default is 'g.con'.
- *DSBFILE* (Define Sides of Block FILE). Definitions of Block Sides, cluster points, grid point numbers, and grid point spacings (*AUTOMESH* option). The default is 'g.dsb'.
- *CTLFILE* (ConTroL FILE). Decay parameters that define the ConTroL functions in the elliptic smoother (*AUTOMESH* option). The default is 'g.ctl'.
- *P3DFILE* (Plot3D FILE). Graphical representation of blocks in 2d multiblock *PLOT3D* format (see section 6.1.2). The default is 'g.p3d'.
- *STGFILE* (SinGular poinT list FILE). Descriptions and locations of all singular points in domain. The default is 'g.stg'.

2.2.4 Boundary Placement

This parameter group defines the position of the farfield boundary. The outer boundary is either a rectangle or a C (depending on the status of *CMESH*) whose dimensions are defined by four coordinate values :

1. *XLB* is the X coordinate of the Left hand Boundary (default: -10.0).
2. *XRB* is the X coordinate of the Right hand Boundary (default: 10.0).
3. *YLB* is the Y coordinate of the Lower Boundary (default: -10.0).
4. *YUB* is the Y coordinate of the Upper Boundary (default: 10.0).

In the case of the C-Mesh, *XLB* is ignored as the C is a semicircle centred around the trailing edge with a radius defined by *YUB* and *YLB*.

2.2.5 Mesh Parameters - Nodes

This group defines the number of nodes or grid points in the mesh :

1. *NPTSAF* (Number of PoinTS on the AirFoil(s)). This is a single number defining the total number of points around the airfoil in the case of a single element and in the absence of cluster points. For a multielement geometry without cluster points, *NPTSAF* is a series of integers (separated by commas) defining the number of nodes around each of the elements. Thus there are as many numbers as elements.

   In the presence of cluster points, the number of entries increases. For a single element geometry with *N* cluster points, *NPTSAF* consists of *N+1* numbers defining the number of nodes. These are defined as starting from the (lower) trailing edge traversing clockwise around the airfoil to the first cluster point (being the one closest to the trailing edge on the lower surface), then between each successive cluster point, and finally from the last cluster point (being the one closest to the trailing edge on the upper surface) to the (upper) trailing edge.

   In the case of multielement geometries with cluster points, *NPTSAF* describes the node numbers for each element in the order they appear in the input geometry file. *NPTSAF* is constructed in a similar way as the single element case above in that the number of grid points for each element are listed as described. These individual elemental lists are then appended into one list in the order that the elements appear in the input geometry file. The default for *NPTSAF* is 129 for all regions between cluster point(s) and trailing edge(s).

   Note that the actual number of nodes placed on the airfoil is always slightly greater than specified. This is done intentionally by the *AUTOMESH* option to generate realistic grid point numbers in small blocks.

2. *NTE* (Number of points on the blunt Trailing Edge). This defines the number of points along the blunt trailing edge for each element. Thus there are as many integers as elements. As for *NPTSAF*, *NTE* is defined by listing the number of points for each element in the order they appear in the input geometry file separated by a comma. *NTE* defaults to 25 for all elements.

3. *NFFBX, NFFBY* (Number of points to the Far Field Boundary in the X and Y coordinate directions). This single input defines the number of grid lines between the surface and the farfield boundary in the two cartesian coordinate directions. *NFFBX* and *NFFBY* both default to 49.

4. *NINT* (Number of points IN beTween the elements). This single input defines the number of grid points vertically spacing any gaps in between the elements for multielement geometries. For single element geometries, this input is ignored. *NINT* defaults to 49.

5. *NGAP* (Number of points in the horizontal GAP). This single input defines the number of grid points spanning the (horizontal) negative overlap between the elements for multielement geometries (if indeed there is one). For single element geometries, the input is ignored. *NGAP* defaults to 29.

2.2.6 Mesh Parameters - Spacings

The mesh spacing parameters are all defined in airfoil units and are listed below. Note that the *D* prefix used for the spacings is to differentiate the spacings form the node numbers at a quick glance.

1. *DNW* (Normal off Wall spacing). This input consists of a series of real numbers (one number for each element) that are the distances from the wall to the first grid line off the surface. For an *N* element geometry, there are *N* numbers (all separated by commas) listed in order that the elements appear in the airfoil geometry data file. *DNW* defaults to 2.0×10^{-5} for each element.

2. *DNFFB* (Normal spacing at the Far Field Boundary). This single input consists of a real number that is the normal off-boundary spacing at the far field boundary. *DNFFB* defaults to 2.0.

3. *DSSTG* (Streamwise spacing at the SinGular poinT). This input consists of *N* real numbers (for an *N* element geometry) defining the streamwise spacing at the singular point(s) of the element(s). The default is 2.0×10^{-5}.

4. *DSTE* (Streamwise spacing at the Trailing Edge). This input consists of *N* real numbers (for an *N* element geometry) defining the streamwise spacing at the trailing edge(s) of the element(s). Note that in the case of a blunt trailing edge, this number defines the *off-wall* spacing along the blunt trailing edge. The default is 5.0×10^{-5}.

5. *DSLE* (Streamwise spacing at the Leading Edge). This single real is the streamwise spacing at all of the leading edge phi lines (if any). If the logical option *LEPHI* is set to *false*, then this is ignored. The default is 1.0×10^{-4}.

6. *DSSHRP* (Streamwise spacing at any SHaRP points). This single real input is the streamwise spacing at any user defined SHaRP points. Note that *DSSHRP* is only used if SHaRP points have been specified (see above). The default is 1.0×10^{-4}.

2.2.7 Mesh Parameters - Clustering

The domain decomposition routine, *kmS*, has the function of automating the clustering option inherent in the algebraic grid generator. This parameter group controls the automated clustering option. Note that *kmS* reorders the clustering data, so as long as the information can be cross-referenced between the following variables, the order is arbitrary (note also that the default is no cluster points).

1. *CLUSFL* (CLUSter on FoiL number ..). This is a list of the element numbers that each cluster point resides on. Thus for a geometry that contains *N* user specified cluster points, there will be *N* entries. If this geometry consists of a single element, then *CLUSFL* will be a list of *N* ones all separated commas (i.e., '1,1,1,1,1....' *N* times). This works in the same way as *SHRPFL* variable described above.

2. *CLUSX* (CLUSter point location). This is the cluster point location defined as a fraction of the normalized arc length. In other words, the leading edge is specified as 0.0, while the trailing edge is specified as 1.0. Note that there is one entry for each cluster point (again entries are separated by commas).

3. *CLUSUL* (CLUSter point on the Upper or Lower surface?). This input is a list of single characters. The upper surface is specified by a 'u' or a 'U', while the lower surface is specified by a 'l' or a 'L'. Again, there is a single entry for each cluster point that cross-references with the other inputs, all separated by commas.

4. *CLUSDS* (CLUSter point streamwise Spacing). This input consists of a list of real numbers defining the streamwise spacing in real space. As above, all entries must cross-reference with the other *CLUSter* point data entries and be separated by commas.

2.2.8 Mesh Parameters - Smoothing Decay Parameters

These parameters are written to *CTLFILE*, and define the angle and spacing control functions in the elliptic smoother. The first three parameters, *SPCFFB*, *SPCPSI*, and *SPCPHI*, determine how much the grid lines parallel to an side are pulled in toward the side. The higher the decay rates the less the effect on interior grid lines. Conversely, the lower the decay rates, the further out into the interior the angle and spacing control extends. *ANGDECAY* determines how close the grid lines are to being normal to the surface and to what extent this continues into the field. Generally, for the elliptic smoother to be stable, *ANGDECAY* is set to be higher than *SPCDECAY*. The default settings for *SPCFFB*, *SPCPSI*, and *SPCPHI* are 0.2, 0.01, and 0.02 respectively. The default for *ANGDECAY* is 0.2.

2.3 AIRFOIL COORDINATE INPUT FILE

The airfoil geometry data file is in a slightly modified DeHavilland *RMS* format. This consists of a simple list of (x,y) coordinate pairs (in two columns). The data progresses from the trailing edge, on the lower surface, clockwise around the airfoil passing the leading edge and finishing at the trailing edge on the upper surface. Thus, the trailing edge point appears twice in the list, once at the start, and once at the end of the list (in the case of a sharp trailing edge). At the start of the (x,y) list for an element, there must be a '*NAME=<airfoil name>' to identify the start of the section data. Simi-

larly, following the coordinate data list there must be a '*EOR' (i.e., End Of Record) to signify the end of the section data. This must be included even if there is another stream of data to follow defining another section (e.g., a flap for a multi element case).

If a section is blunt then it is assumed that the trailing edge curve is straight (i.e., the upper and lower trailing edge points are joined with a straight line). If this is not the case (e.g., the trailing edge is concave) then this can be specified also. This is done by inserting a '*TE=<airfoil name>' directly following the (x,y) pairs for the section and prior to the '*EOR' and listing the (x,y) coordinates defining the trailing edge. The coordinate pairs must run clockwise around the airfoil to be consistent with the rest of the data (i.e., the list must start at the upper trailing edge point and traverse to the lower trailing edge point). Note that a '*EOR' must follow the whole section (i.e., airfoil and trailing edge).

Comments may be placed in the airfoil data list by using an exclamation mark (i.e., '!') preceding the comment in the same manner as in the input control file. These may be used at any point in the data file, as they are just ignored when the coordinates are read.

2.4 OUTPUT

Once *kmS* finishes running, an estimate of the number of grid points based on *AUTOMESH* parameters is given. An array size allowing for 2 rows of halo points around all blocks is also provided.

There are eight output files that *kmS* generates. These are :

1. *SPLFILE* (SPLine FILE). This file contains the spline coefficients and points for all curves decomposing the domain. It is used by the algebraic grid generator. *SPLFILE* is always created and defaults to 'g.spl'. The format is *TECPLOT* as shown in section 6.1.1.

2. *DECFILE* (DEComposition FILE). This file contains the block definitions for the decomposition. The first and second columns define the block and side numbers. The third through fifth columns show the curve number and the starting and ending parametric values that define the block side. The numbering convention for the sides is shown in

figure 2.1, while the block and side numbering convention for a simple single element airfoil with a H-domain and a C-domain are shown in figures 2.2 and 2.3. *DECFILE* is always created and defaults to 'g.dec'.

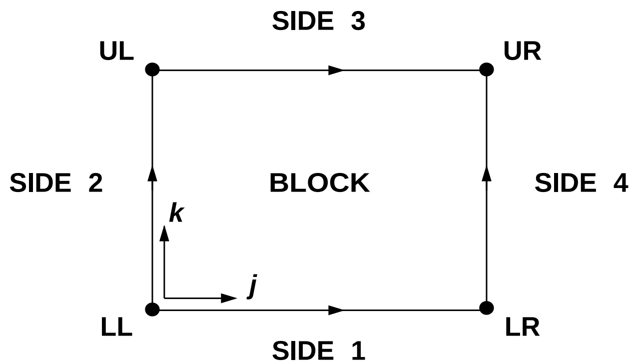

Figure 2.1: Side numbering and direction conventions for an arbitrary block

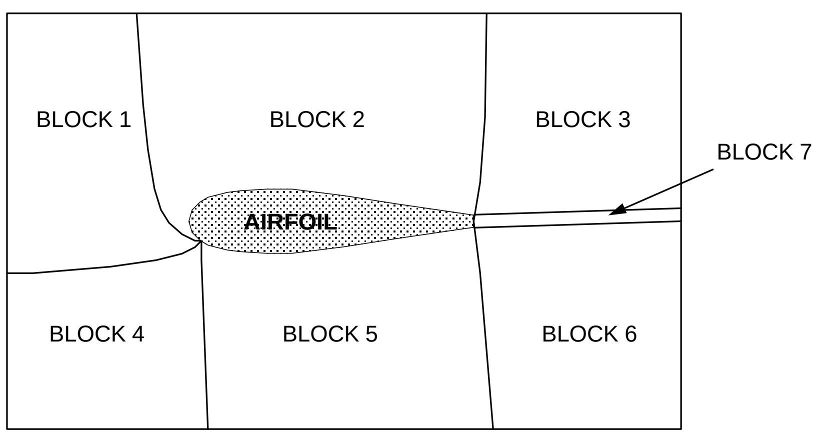

Figure 2.2: Simple H-domain decomposition showing block numbers.

3. *CONFILE* (CONnectivity FILE). This file contains the map of how the blocks fit together. This information is used by the flow solver and the *AUTOCHECK* routine in the algebraic grid generator. The first two columns define the block in question, while the third and fourth columns show which block and side that connect to it. The fifth column

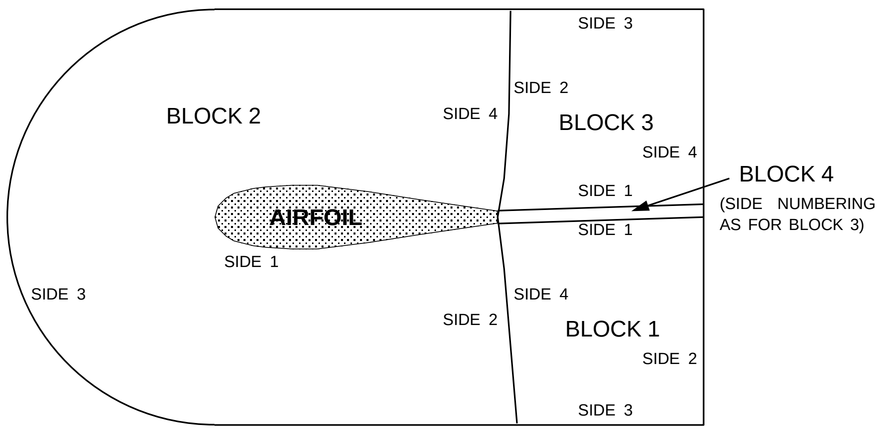

Figure 2.3: Simple C-domain decomposition showing block and side numbers.

is the relative direction of the coincident sides. This number is '-1' if the directions are opposing and '+1' if the directions are identical. The sixth column is the boundary condition type:

"0" denotes no boundary condition as used at interior interfaces.

"1" denotes an airfoil wall. The direction associated with this boundary condition type is important also. A non-zero value of the direction (contained in column 7) indicates both element number and direction (see below) whereas a zero value indicates an inviscid wall along which no integration is performed in calculating the force and moment coefficients.

"2" denotes far field boundary condition (inviscid).

Column seven lists the boundary condition direction and consists of two pieces of information, namely the magnitude and the sign. The magnitude indicates the element number that the particular side belongs to while the sign indicates the direction of the boundary. A plus sign indicates that boundary will be integrated in the same direction as the block side. Conversely, a negative sign indicates that the boundary is to be integrated in the opposing direction to the block side. A value of zero indicates an inviscid wall along which no integration is performed and as such this edge does not contribute to the force and moment coefficients.

The final two columns are the range of grid point indices where the turbulence model is not active (i.e., where the flow is laminar). These columns will consist of "-99", indicating that the turbulence model is to be used, or that the flow is turbulent, along that edge. *TORNADO* automatically determines the transition grid point indices from the user specified *TRANSUP* and *TRANSLO* variables. This provides the user an easier way to set transition than manually adjusting the last two columns. Note that this is irrelevant unless *TORNADO* is used. More details can be found in the *TORNADO* user's guide. *CONFILE* is always created and defaults to 'g.con'.

4. *DSBFILE* (Define Sides of Blocks FILE). This file contains the grid information required by the algebraic grid generator. The first two columns define the block and side numbers. In the third column, the number of grid points along the block side is defined. Columns four and five contain the starting and ending parametric values for the curve that define the block side. In the absence of any clustering points, this information (in columns four and five) is identical to that in *DECFILE*, and as such is a reference only. For a decomposition that contains clustering points, the parametric value of the cluster point is also included. This is done by defining the ending parametric value of the given side to be the cluster point parametric value. The next line starts with the same block and side number but the starting parametric value is that of the cluster point, whereas the ending value is the ending value for the whole block side. Finally the last two columns define the starting and ending grid point spacings along the defined edge in real space. *DSBFILE* is only created if the *AUTOMESH* logical option is set to *true*. The default is 'g.dsb'.

5. *CTLFILE* (ConTroL FILE). This file is used by the elliptic grid smoother and defines the control decay rate parameters. Columns one and two index the block and side number. The third column is a reference only and shows the number of grid points along the side (as defined in *DSBFILE*). In columns four and five define the boundary condition type and the control type to be used within the elliptic smoother. The last two columns contain the angle and spacing decay rates used by the elliptic smoother determine the control functions. See section 2.2.8 for

more details. *CTLFILE* is only created in conjunction with *DSBFILE* when the *AUTOMESH* logical option is set to *true*. The default is 'g.ctl'.

6. *P3DFILE* (Plot3D FILE). This file contains the block coordinates in double precision 2d multiblock *PLOT3D* format. Thus this file allows the user to view the domain decomposition block by block via *PLOT3D* or *FIELDVIEW* to ensure that the decomposition is suitable. Note that the appropriate conversion routine must be used prior to viewing. The block numbering logic can also be seen using this feature. *P3DFILE* is always created and defaults to 'g.p3d'. Double precision 2d multiblock *PLOT3D* format is shown in section 6.1.2.

7. *STGFILE* (SinGular poinT FILE). This file contains the description(s) and location(s) of all singular points in the decomposed domain. The flow solver *TORNADO* requires this input. The first three lines are descriptors and are not used. Columns two and three contain the block number (integer) containing the singular point. The location of the singular point is contained in columns 12 and 13 and is one of the following: "UL" (Upper Left), "UR" (Upper Right), "LL" (Lower Left), or "LR" (Lower Right). This input describes the corner of the block number that the singular point lies at using the convention shown in figure 2.1. The last input (columns 22 and 23) define the type ("LE" for Leading Edge, and "TE" for Trailing Edge). Note that for a blunt trailing edge airfoil, there are two trailing edge singular points to be defined. Also, for a C-Mesh domain, there is no leading edge singular point, as there are no streamlines or equipotential originating from this region. *STGFILE* is always created and defaults to 'g.stg'.

8. *RESTART FILE* ("kmS.dat"). This file contains the data required by *kmS* to perform the *AUTOMESH* option without redoing the decomposition. The user need not concern themselves with the format of this file. No control over the name is given and "kmS.dat" is always created when the decomposition is performed.

Chapter 3

ALGEBRAIC GRID GENERATOR: agrid

3.1 OVERVIEW

The algebraic grid generator, *agrid*, is intended to quickly produce a rough starting grid that will be smoothed by the elliptic smoother. This routine takes the domain generated by *kmS*, along with the required grid parameters produced by *AUTOMESH*, and develops a grid. Alternatively, the grid parameter file, *DSBFILE*, can be generated by either editing the *AUTOMESH* version or constructing it manually. The routine *AUTOCHECK* automatically examines *DSBFILE* and warns the user of any inconsistencies in the mesh spacing and grid point specification. This can save time and effort in generating a customized mesh. The number of grid points is counted by *agrid* and displayed when executed. Any wrappings or other undesirable features in the algebraic grid will be smoothed out by *megrid* in the next stage.

3.2 INPUT

There are four input files that are used by the algebraic grid generator, the input control file, *DECFILE*, *SPLFILE*, *DSBFILE*, and *CONFILE*. Each of these files will be discussed in turn.

3.2.1 INPUT CONTROL FILE: agrid.inp

This file, named *'agrid.inp'*, specifies the names of the other input files and the output grid file in the following order :

1. *DECFILE* (DEComposition FILE)
2. *SPLFILE* (SPLine data FILE)
3. *DSBFILE* (Define Sides of Block FILE)
4. *CONFILE* (CONnectivity FILE)
5. OUTPUT GRID FILE

3.2.2 DECFILE

This file contains the block definitions for the decomposition. The first and second columns define the block and side numbers. The third through fifth columns show the curve number and the starting and ending parametric values that define the block side. The file is created by the domain decomposition routine *kmS*.

3.2.3 SPLFILE

This file contains the spline coefficients and points for all curves decomposing the domain. The format is in *TECPLOT* format (see section 6.1.1). It is also created by *kmS*.

3.2.4 DSBFILE

This file contains the grid information required by the algebraic grid generator *agrid*. The first two columns define the block and side numbers. In the third column, the number of grid points along the block side is defined. Columns four and five contain the starting and ending parametric values for the curves that define the block side. In the absence of any clustering points, this information (in columns four and five) is identical to that in *DECFILE*, and as such is a reference only. For a decomposition that contains clustering points, the parametric value of the cluster point is also included. This is done

by defining the ending parametric value of the given side to be the cluster point parametric value. The next line starts with the same block and side number but the starting parametric value is that of the cluster point, whereas the ending value is the ending value for the whole block side. Finally the last two columns define the starting and ending grid point spacings along the defined edge in real space. This file can be created using the *AUTOMESH* option when running the domain decomposition routine *kmS* or created by hand. If the user is unhappy with the resulting algebraic grid, either the *DSBFILE* can be edited to alter the spacings and/or number of grid points, or *kmS* can be rerun using altered grid parameters with *DECOMP* set to *false*.

3.2.5 CONFILE

This file describes the connectivity of all the blocks that decompose the domain. It is generated by *kmS* automatically. The routine *AUTOCHECK* uses this information to check for grid point and grid spacing specification inconsistencies. For the exact format of this file, see section 2.4.

3.3 OUTPUT

Agrid creates a single output grid file, usually denoted by a *'.agr'* extension. This file contains the algebraic grid coordinates in a double precision version of *PLOT3D* unformatted 2d multiblock format. This can now be smoothed using the elliptic smoother. Double precision 2d multiblock unformatted *PLOT3D* format is shown in section 6.2.

Chapter 4

MULTIBLOCK ELLIPTIC SMOOTHER: megrid

4.1 OVERVIEW

The elliptic smoother, *megrid*, smooths the algebraic grids generated by *agrid* in order to produce a high quality mesh. This is important in order to achieve an accurate flow solution. Sorenson's method is employed in this routine allowing the user control of the smoothing and general appearance of the grid. An optional graphical window is also provided to show the smoothing in action.

The grid smoothing parameters are defined in the *CTLFILE*. This file can either be created manually, a rather arduous task indeed, or created automatically by the *AUTOMESH* option in the domain decomposition routine *kmS*. To create this file manually, the user must specify eight separate smoothing parameters (four spacing control and four angle control decay parameters) for each block! To create the smoothing parameter file using *AUTOMESH*, the user simply specifies a total of four parameters (one angle control decay rate and three spacing control decay rates) based on block side type. This produces grids of high quality that will suffice for the majority of cases. For the small number of cases that extra control is required, the *CTLFILE* can be edited to achieve the desired results.

4.2 INPUT

There are four input files that are required by *megrid*, the input control file *megrid.inp*, the grid file (the algebraic *.agr* grid file, *CTLFILE*, *CONFILE*, and an offset file (denoted by the *.off* extension) that can be used to specify varying offwall spacing (this option is currently not supported). These are described below.

4.2.1 INPUT CONTROL FILE: megrid.inp

Sample *megrid.inp* input control files are shown in the Appendices. Each entry in this file will be described individually :

1. GRAPHICS. The first entry is a simple logic switch that indicates if the graphic display is to be active (*true* or *false*).

2. GRID INPUT FILE NAME. Row two contains the grid input file name. This grid could be generated by the algebraic grid generator, *agrid*, or a previously smoothed grid from *megrid*. Note that the format of this input file is either double precision unformatted *PLOT3D* 2d multiblock or (double precision) *TECPLOT* (see section 6.3).

3. GRID INPUT FORMAT. This is either "plot3d" or "tecplot".

4. *CTLFILE*. This file is generated by *kmS* but may be edited or replaced with a handmade version.

5. *CONFILE*. The connectivity file as generated by *kmS*.

6. OFFSET FILE NAME. This currently unsupported option allows the user to specify a distribution of offwall spacing along block sides.

7. GRID OUTPUT FILE NAME. This file is the smoothed grid output file in either double precision *PLOT3D* unformatted 2d multiblock format or (double precision) *TECPLOT* format (see section 6.3).

8. GRID OUTPUT FORMAT. This is either "plot3d" or "tecplot".

9. NUMBER OF BLOCKS. This is the number of blocks in the domain as created by *kmS*. Note that this is for reference only.

10. SOR RELAXATION PARAMETER. This is the SOR relaxation parameter used by the numerical scheme in *megrid*. Note that this number must be greater than zero and less than two (1.0 works well).

11. MAX ITERATIONS. This integer sets the maximum number of iterations used to smooth the grid.

12. RESIDUAL. This is the largest residual required to terminate smoothing (i.e., once the residual falls below this value, smoothing is terminated).

13. RELAXATION FACTORS FOR P AND Q. These are the relaxation factors for the control functions P and Q used in the elliptic gridder. Values of 0.1 and 0.1 work well.

14. LIMITING FACTORS FOR P AND Q. These numbers control how large the control functions get during the successive iterations. Values of 0.1 and 0.1 work well.

4.2.2 GRID INPUT FILE

As stated above, this is a grid file generated by *agrid* (although a grid from *megrid* could be used if more smoothing is wanted). The format is double precision *PLOT3D* unformatted 2d multiblock or it can be in *TECPLOT* format (see section 6.3) if specified.

4.2.3 CTLFILE

This file contains the control function decay parameters and boundary condition type for all block sides. The user can manually create this file or use the *AUTOMESH* option. Columns one and two index the block and side number. The third column is a reference and shows the number of grid points along the indexed block side. Column four defines the boundary condition type and is currently set to 1 indicating fixed boundaries. The fifth column defines the control type and defaults to Sorenson's method (indicated by the integer 2). This can be changed to a Laplace method which tends to equispace grid lines with the control functions being ignored (indicated by the integer 1). The sixth column is the angle decay rates for each side. This

determines how close the grid lines are to being normal to the surface, and how far this extends into the interior. The larger the number, the greater the decay rate of the angle control function. Finally, the last column defines the spacing control decay rates which determine to what extent the grid lines parallel to the body are attracted to the body. The higher the number the less the attraction. In fact, the lines will generally move away from walls (i.e., the tendency is to achieve an equispaced grid). A low, or zero, decay rate will counter this outward migration somewhat. This is also discussed in section 2.4.

4.2.4 CONFILE

As the name implies, this file contains information on how the blocks connect together. This file is created by the decomposition routine *kmS*. For a full description see section 2.4.

4.2.5 OFFSET FILE

This file contains information which allows the user to specify the distribution of off wall spacing along any block side(s). Currently, this option is not supported.

4.3 OUTPUT

There is a single output file, being the output grid file. This file, usually denoted by the *.egr* extension, contains the coordinates of the elliptically smoothed grid. The format is user specified, being either double precision *TECPLOT* format, or double precision *PLOT3D* unformatted 2d multiblock format (see section 6.3).

Chapter 5

PLOTTING GRID FILES

5.1 OVERVIEW

There are a number of ways that can be used to graphically view the resulting grids from either *agrid* or *megrid*. These are discussed below in turn. It should be noted that the *plot3d* format used by *AMBER2d* is not standard. All the *plot3d* format grid files (i.e., *.agr* and *.egr* files) are in double precision instead of the standard single precision. This was done to preserve accuracy for very fine meshes where grid points are so close together that single precision is not precise enough to resolve the points as distinct. It was deemed prudent to take this path rather than to use a completely new data format.

5.2 FIELDVIEW

FIELDVIEW is the graphical interface chosen by DeHavilland Aircraft Company for use with *AMBER2d*. This plotting package is *PLOT3D* 'compatible'. However this compatibility does not extend to 2d multiblock format. Thus a post processor called *prefv* is provided to convert the data into a form readable by *FIELDVIEW*. What *prefv* (PRE FieldView) does is to convert the double precision 2d multiblock *PLOT3D* format into single precision 3d multiblock *PLOT3D* format. The grid file can then be read in by *FIELDVIEW* and plotted. The usage is as follows :

prefv <*input*> <*output*>

where <*input*> is the input file name (double precision 2d multiblock *PLOT3D* format) and <*output*> is the output file name (single precision 3d multiblock *PLOT3D* format). Both file names are assumed to include any extensions. Usage of a preprocessor was chosen over outputting a *FIELDVIEW* format as padding the 2d grids with 1's required to express the data in 3d would increase the size of these files by fifty percent!

5.3 PLOT3D

If the GRID INPUT FORMAT in *megrid.inp* is set to *plot3d* then the resulting format is double precision unformatted 2d multiblock format. In order that this grid file be read by *PLOT3D*, it must be simply converted into single precision, all else being equal. The post processor *prep3d* (PRE-Plot3D) is provided for just this purpose. The usage is :

prep3d <*input*> <*output*>

where <*input*> is the input name in double precision 2d multiblock *PLOT3D* format and <*output*> is the equivalent in single precision. Note that, again, the filenames include any extensions.

5.4 TECPLOT

If the GRID INPUT FORMAT option in *megrid.inp* is set to *tecplot* then the *TECPLOT* plotting package can be used to plot and view the resulting grids. Alternatively the *TECPLOT* graphics package will process *PLOT3D* format data files correctly. Thus *prep3d* could also be used here.

Chapter 6

GRID FILE DATA FORMATS

6.1 DOMAIN DECOMPOSITION: kmS

6.1.1 SPLFILE

The *SPLFILE* is in double precision *TECPLOT* format as shown below:

```
      WRITE(IUNIT,*)
     +  'TITLE = " SPLINES FROM KM DOMAIN DECOMPOSITION" '
      WRITE(IUNIT,*)
     +  'VARIABLES="X","Y","S","CX1","CX2","CX3","CY1","CY2","CY3"'
      DO 10 II=1,ICCNT
        WRITE(IUNIT,101) NCRV(II)
        DO 10 JJ=1,NCRV(II)
          WRITE(IUNIT,102) XCRV(JJ,II),YCRV(JJ,II),SCRV(JJ,II),
     +                     CX(JJ,3,II),CX(JJ,2,II),CX(JJ,1,II),
     +                     CY(JJ,3,II),CY(JJ,2,II),CY(JJ,1,II)
   10 CONTINUE
  101 FORMAT('ZONE, I=',I3,', F=POINT')
  102 FORMAT(9E16.8)
```

6.1.2 P3DFILE

The *P3DFILE* is in double precision unformatted multiblock 2d *PLOT3D* format as described by the write statements below:

```
      WRITE(IUNIT) NBLOCK
      WRITE(IUNIT) (N(IGRID),1,IGRID=1,NBLOCK)
      DO 10 IGRID=1,NBLOCK
        WRITE(IUNIT) (X(II,IGRID),II=1,N(IGRID)),
     +               (Y(II,IGRID),II=1,N(IGRID))
   10 CONTINUE
```

6.2 ALGEBRAIC GRID FILE

The output algebraic grid file is in double precision unformatted multiblock 2d *PLOT3D* format as shown:

```
      WRITE(IUNIT) NGRID
      WRITE(IUNIT) (JDIM(IGRID),KDIM(IGRID),IGRID=1,NGRID)
      DO 10 IGRID=1,NGRID
        WRITE(IUNIT)
     +    ((X(J,K),
     +      J=1,JDIM(IGRID)),
     +      K=1,KDIM(IGRID)),
     +    ((Y(J,K),
     +      J=1,JDIM(IGRID)),
     +      K=1,KDIM(IGRID))
   10 CONTINUE
```

6.3 ELLIPTIC GRID FILE

The elliptic grid output file can be in either (double precision) *TECPLOT* format or in double precision unformatted multiblock 2d *PLOT3D* format. These are shown individually below.

6.3.1 Tecplot

```
      WRITE(IUNIT,*) 'TITLE = " TEST OF TGRID BLOCK GRIDS" '
      WRITE(IUNIT,*) 'VARIABLES = "X","Y"'
      DO 10 KK=1,NBLKS
        WRITE(IUNIT,101) NGRDX(KK),NGRDY(KK)
```

```
      WRITE(IUNIT,102) ((X(II,JJ,KK),II=1,NGRDX(KK)),
     +                               JJ=1,NGRDY(KK))
      WRITE(IUNIT,102) ((Y(II,JJ,KK),II=1,NGRDX(KK)),
     +                               JJ=1,NGRDY(KK))
   10 CONTINUE
  101 FORMAT('ZONE  I=',I3,', J=',I3,', F=BLOCK')
  102 FORMAT(8E14.6)
```

6.3.2 Plot3d

```
      WRITE(IUNIT) NGRID
      WRITE(IUNIT) (JDIM(IGRID),KDIM(IGRID),IGRID=1,NGRID)
      DO 10 IGRID=1,NGRID
        WRITE(IUNIT)
     +    ((X(J,K),
     +      J=1,JDIM(IGRID)),
     +      K=1,KDIM(IGRID)),
     +    ((Y(J,K),
     +      J=1,JDIM(IGRID)),
     +      K=1,KDIM(IGRID))
   10 CONTINUE
```

Appendix A : WTBV 35/35

This Appendix contains example input and output for the four element De-Havilland WTBV 35/35 test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.1)
5. Algebraic H-grid plot (figure 6.2)
6. Elliptically smoothed H-grid plot (figure 6.3)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! BENCHMARK #2 TEST CASE
!
 &INPUT
! AIRFOIL COORDINATES :
       FOIL='/usr/people/andrew/airfoils/benchmarks/bm2newS.dat'
! CONTROL PARAMETERS :
       ALPHA  = 2.0,            ! Angle of attack
       ROTATE  = 0.0,           ! Rotation angle
       X0 = 0.5,                ! X coordinate of rotation centre
       Y0 = 0.0,                ! Y coordinate of rotation centre
       LEPHI  = FALSE,          ! PHI-line from L.E. (t/f)
       CMESH  = FALSE,          ! C-MESH option (t/f)
       AUTOMESH = TRUE,         ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,           ! DECOMPose domain (t/f)
       MESHSEQ = 0,             ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'bm2.spl',     ! SPLine file (spline coefficients)
       DECFILE = 'bm2.dec',     ! DECComposition file (block definitions)
       CONFILE = 'bm2.con',     ! CONnectivity file (block connections)
       DSBFILE = 'bm2.dsb',     ! Define Sides of Block file
       CTLFILE = 'bm2.ctl',     ! ConTroL file (control fn' decay param.)
       P3DFILE = 'bm2.p3d',     ! Plot3D file for block boundaries
       STGFILE = 'bm2.stg',     ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -10.00000,         ! Left Hand Boundary location
       XRB =  10.00000,         ! Right Hand Boundary location
       YLB = -10.00000,         ! Lower Boundary location
       YUB =  10.00000,         ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 29,39,          ! No. PoinTS on AirFoil(s) in order from t.e.
                29,125,         !   to any cluster points to t.e. going c.w.
                24,29,49,
                29,39,
       NTE = 15,15,15           ! No. points on the blunt Trailing Edge
       NFFBX = 29,              ! No. points to the Far-Field Boundary
       NFFBY = 29,              ! No. points to the Far-Field Boundary
       NINT = 35,               ! No. vertical points IN-beTween the sections
       NGAP = 32,               ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,2.0E-05,   ! normal off Wall spacing
             2.0E-05,2.0E-05,
       DNFFB = 3.0,             ! normal spacing at the Far-Field Boundary
```

```
        DSSTG = 1.0E-03,1.0E-03,  ! surface spacing at the STaGnation point
                1.0E-03,1.0E-03,
        DSTE = 2.0E-04,2.0E-04,   ! surface spacing at the Trailing Edge
               2.0E-04,2.0E-04,
        DSLE = 5.0E-04,           ! surface spacing at any L.E. ctl line
        DSSHRP = 6.0E-04,         ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
        CLUSFL = 1,2,3,3,4,       ! CLUSter on FoiL # ..
        CLUSX = 0.15,0.65,0.65,   ! norm. arc length from LE (=0.0) to TE (=1.0)
        CLUSUL = 'L','L','L',     ! U for Upper surface, L for Lower surface
                 'L','L',
        CLUSDS = 4.0E-03,4.0E-03, ! CLUSter surface spacing (ds)
                 4.0E-03,
                 3.0E-03,2.5E-03,
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
        SPCFFB = 0.2,             ! SPaCing ConTroL parameter for ffb
        SPCPSI = 0.01,            ! SPaCing ConTroL parameter for PSI lines
        SPCPHI = 0.02,            ! SPaCing ConTroL parameter for PHI lines
        ANGDECAY = 0.2            ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
bm2.dec                         ! Input decomposition file
bm2.spl                         ! Input spline file
bm2.dsb                         ! Input block side file
bm2.con                         ! Input connectivity file
bm2.agr                         ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                                ! graphics on/off
bm2.agr                              ! grid input file
plot3d                               ! grid format (tecplot or plot3d)
bm2.ctl                              ! boundary condition file
bm2.con                              ! block connectivity file
bm2.off                              ! offset file
bm2.g                                ! output file in tecplot format
plot3d                               ! output file format (tecplot or plot3d)
33                                   ! number of blocks
1.0                                  ! SOR relaxation parameter (0<w<2)
2000                                 ! Max iterations
1.0E-5                               ! Min residual for stopping
0.100000  0.1000000                  ! Relaxation Factors for P & Q
0.100000  0.1000000                  ! Limiting Factors for P & Q
```

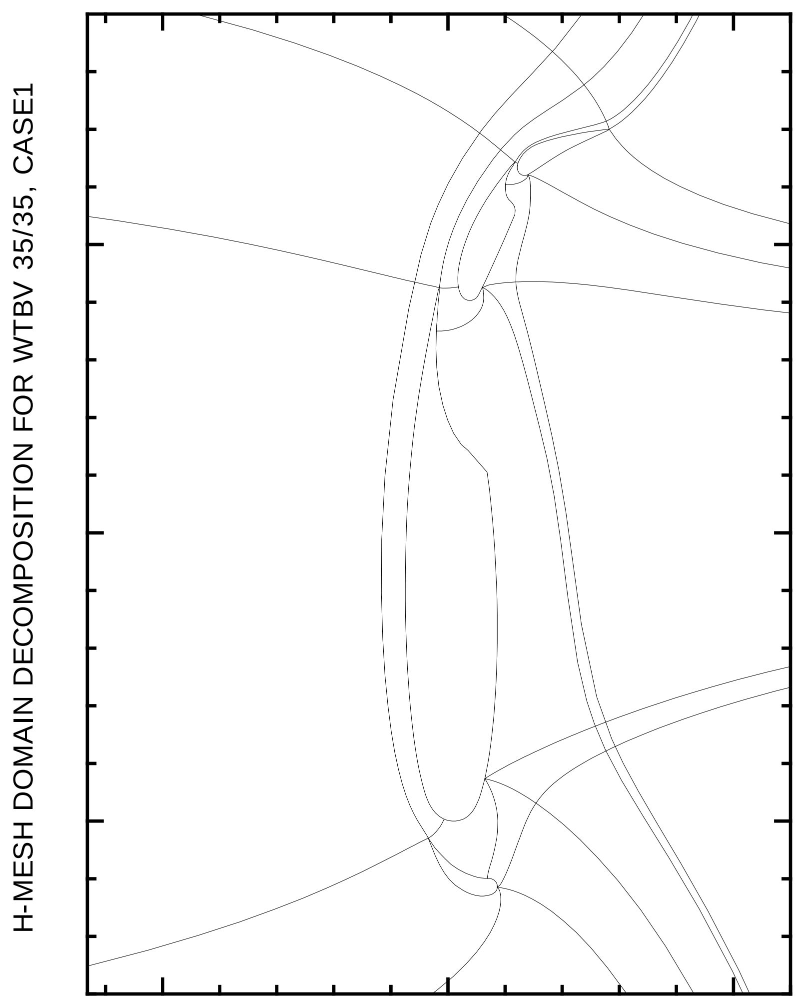

Figure 6.1: Domain decomposition for Test Case 1

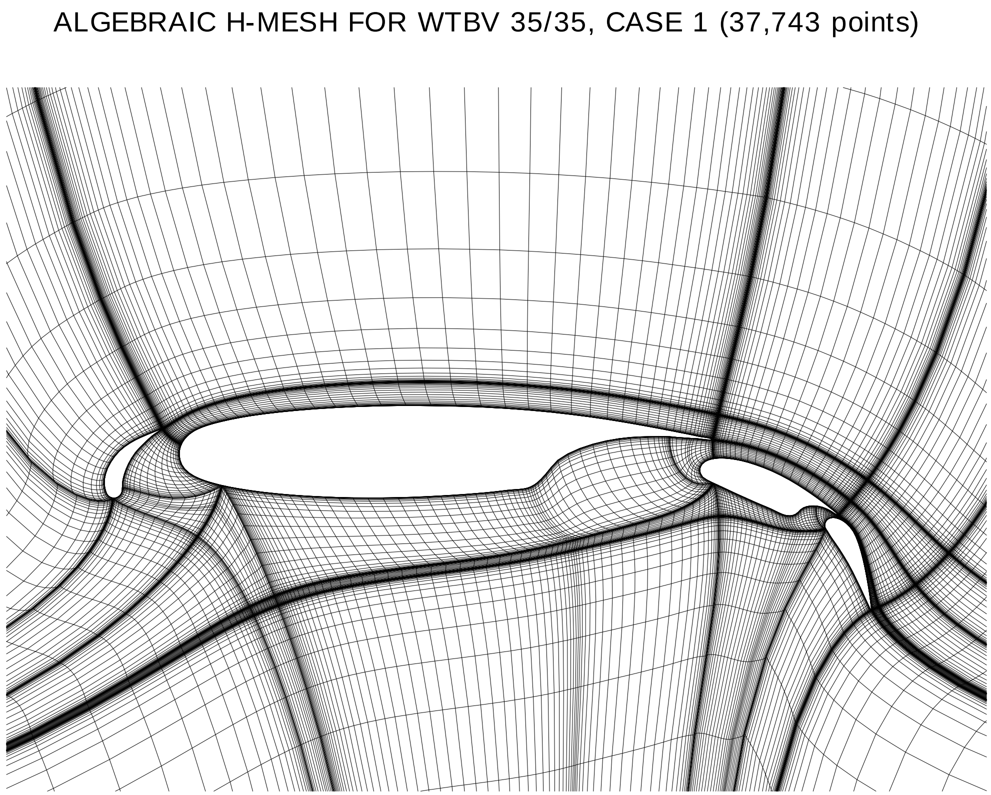

Figure 6.2: Algebraic grid for Test Case 1

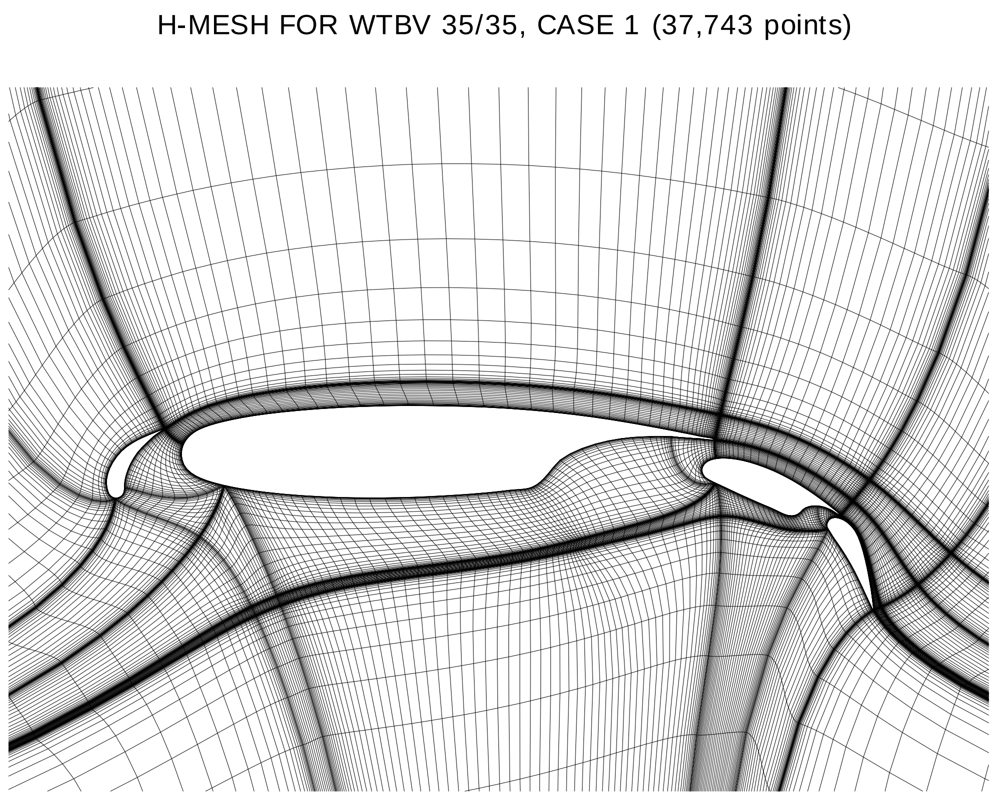

Figure 6.3: Smoothed grid for Test Case 1

Appendix B : WTBV 25/00

This Appendix contains example input and output for the three element DeHavilland WTBV 25/00 test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.4)
5. Algebraic H-grid plot (figure 6.5)
6. Elliptically smoothed H-grid plot (figure 6.6)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! BENCHMARK #3 TEST CASE
!
 &INPUT
! AIRFOIL COORDINATES (NO DEFAULT) :
       FOIL='/usr/people/andrew/airfoils/benchmarks/bm3s.dat'
! CONTROL PARAMETERS :
       ALPHA  = 10.0,            ! Angle of attack
       ROTATE  = 0.0,            ! Rotation angle
       X0 = 0.5,                 ! X coordinate of rotation centre
       Y0 = 0.0,                 ! Y coordinate of rotation centre
       LEPHI  = FALSE,           ! PHI-line from L.E. (t/f)
       CMESH  = FALSE,           ! C-MESH option (t/f)
       AUTOMESH = TRUE,          ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,            ! DECOMPose domain (t/f)
       MESHSEQ = 0,              ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'bm3.spl',      ! SPLine file (spline coefficients)
       DECFILE = 'bm3.dec',      ! DEComposition file (block definitions)
       CONFILE = 'bm3.con',      ! CONnectivity file (block connections)
       DSBFILE = 'bm3.dsb',      ! Define Sides of Block file
       CTLFILE = 'bm3.ctl',      ! ConTroL file (control fn' decay param.)
       P3DFILE = 'bm3.p3d',      ! Plot3D file for block boundaries
       STGFILE = 'bm3.stg',      ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -10.00000,          ! Left Hand Boundary location
       XRB =  10.00000,          ! Right Hand Boundary location
       YLB = -10.00000,          ! Lower Boundary location
       YUB =  10.00000,          ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 29,39,           ! No. PoinTS on AirFoil(s) in order from t.e.
                25,105,          !   to any cluster points to t.e. going c.w.
                99,
       NTE = 5,15,19             ! No. points on the blunt Trailing Edge
       NFFBX = 29,               ! No. points to the Far-Field Boundary
       NFFBY = 29,               ! No. points to the Far-Field Boundary
       NINT = 35,                ! No. vertical points IN-beTween the sections
       NGAP = 32,                ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,2.0E-05,    ! normal off Wall spacing
             2.0E-05,
       DNFFB = 3.0,              ! normal spacing at the Far-Field Boundary
       DSSTG = 1.0E-03,1.0E-03,  ! surface spacing at the STaGnation point
```

```
           1.0E-03,
       DSTE = 2.0E-04,2.0E-04,   ! surface spacing at the Trailing Edge
              2.0E-04,
       DSLE = 5.0E-04,           ! surface spacing at any L.E. ctl line
       DSSHRP = 6.0E-04,         ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
       CLUSFL = 1,2,             ! CLUSter on FoiL # ..
       CLUSX = 0.15,0.65,        ! norm. arc length from LE (=0.0) to TE (=1.0)
       CLUSUL = 'L','L',         ! U for Upper surface, L for Lower surface
       CLUSDS = 1.0E-03,4.0E-03, ! CLUSter surface spacing (ds)
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
       SPCFFB = 0.2,             ! SPaCing ConTroL parameter for ffb
       SPCPSI = 0.01,            ! SPaCing ConTroL parameter for PSI lines
       SPCPHI = 0.02,            ! SPaCing ConTroL parameter for PHI lines
       ANGDECAY = 0.2            ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
bm3.dec                         ! Input decomposition file
bm3.spl                         ! Input spline file
bm3.dsb                         ! Input block side file
bm3.con                         ! Input connectivity file
bm3.agr                         ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                               ! graphics on/off
bm3.agr                             ! grid input file
plot3d                              ! grid format (tecplot or plot3d)
bm3.ctl                             ! boundary condition file
bm3.con                             ! block connectivity file
bm3.off                             ! offset file
bm3.egr                             ! output file in tecplot format
plot3d                              ! output file format (tecplot or plot3d)
28                                  ! number of blocks
1.0                                 ! SOR relaxation parameter (0<w<2)
2000                                ! Max iterations
1.0E-5                              ! Min residual for stopping
0.100000  0.1000000                 ! Relaxation Factors for P & Q
0.100000  0.1000000                 ! Limiting Factors for P & Q
```

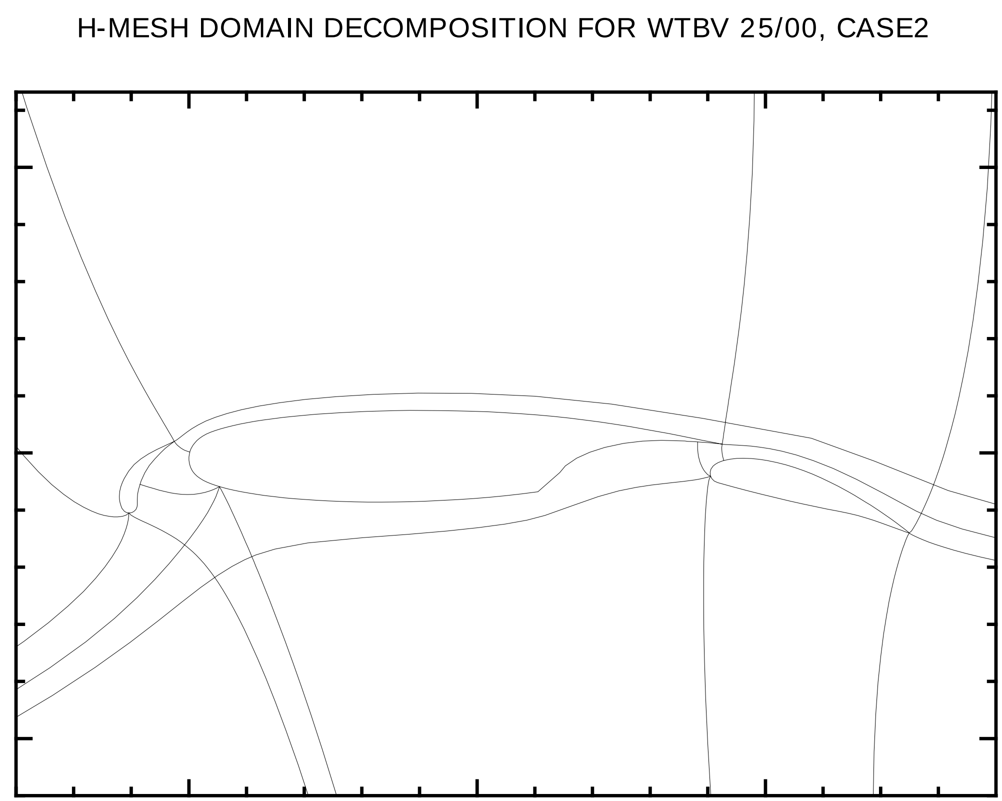

Figure 6.4: Domain decomposition for Test Case 2

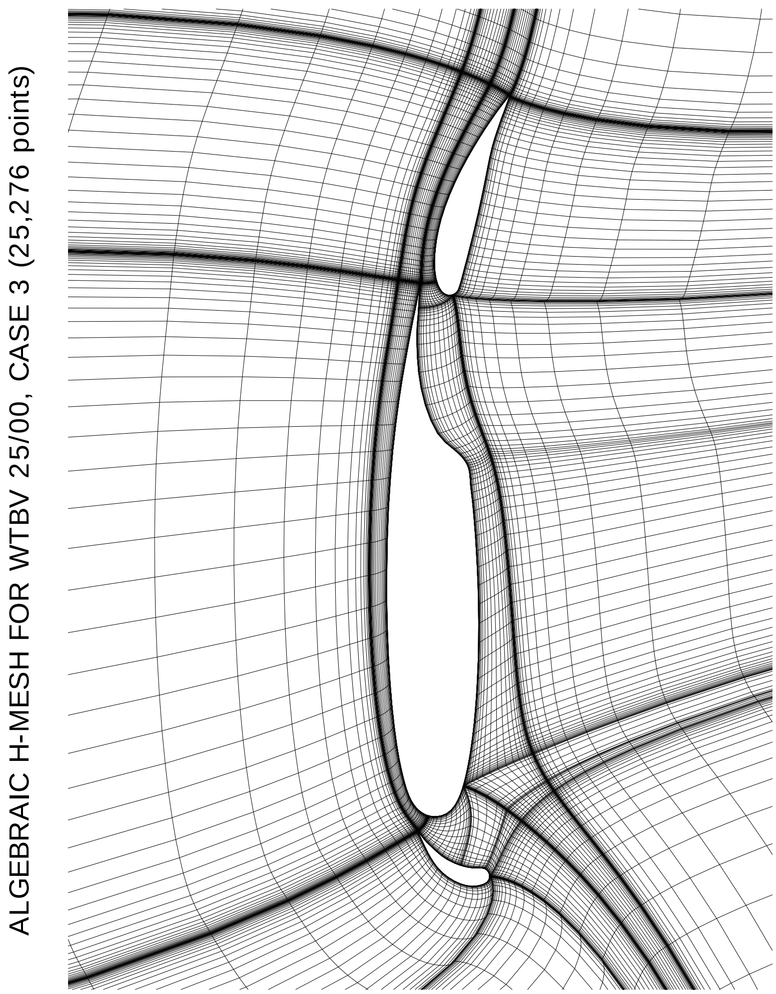

Figure 6.5: Algebraic grid for Test Case 2

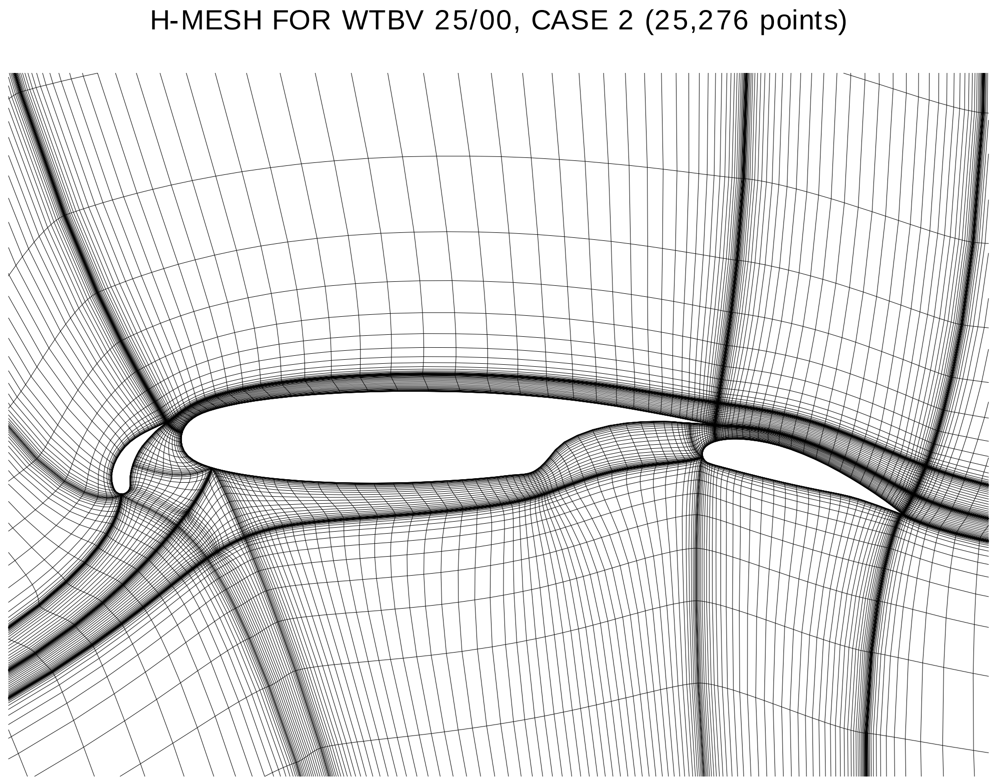

Figure 6.6: Smoothed grid for Test Case 2

Appendix C : WTEA4 t.e.=0.5%c

This Appendix contains example input and output for this single element DeHavilland WTEA t.e.=0.5%c test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.7)
5. Algebraic H-grid plot (figure 6.8)
6. Elliptically smoothed H-grid plot (figure 6.9)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! BENCHMARK #4 TEST CASE
!
 &INPUT
! AIRFOIL COORDINATES (NO DEFAULT) :
       FOIL='/usr/people/andrew/airfoils/benchmarks/bm4.dat'
! CONTROL PARAMETERS :
       ALPHA  = 0.0,            ! Angle of attack
       ROTATE  = 0.0,           ! Rotation angle
       X0 = 0.5,                ! X coordinate of rotation centre
       Y0 = 0.0,                ! Y coordinate of rotation centre
       LEPHI  = FALSE,          ! PHI-line from L.E. (t/f)
       CMESH  = TRUE,           ! C-MESH option (t/f)
       AUTOMESH = TRUE,         ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,           ! DECOMPose domain (t/f)
       MESHSEQ = 0,             ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'bm4.spl',     ! SPLine file (spline coefficients)
       DECFILE = 'bm4.dec',     ! DEComposition file (block definitions)
       CONFILE = 'bm4.con',     ! CONnectivity file (block connections)
       DSBFILE = 'bm4.dsb',     ! Define Sides of Block file
       CTLFILE = 'bm4.ctl',     ! ConTroL file (control fn' decay param.)
       P3DFILE = 'bm4.p3d',     ! Plot3D file for block boundaries
       STGFILE = 'bm4.stg',     ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -8.00000,          ! Left Hand Boundary location
       XRB =  9.00000,          ! Right Hand Boundary location
       YLB = -8.00000,          ! Lower Boundary location
       YUB =  8.00000,          ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 100,100,        ! No. PoinTS on AirFoil(s) in order from t.e.
                                !   to any cluster points to t.e. going c.w.
       NTE = 29,                ! No. points on the blunt Trailing Edge
       NFFBX = 49,              ! No. points to the Far-Field Boundary
       NFFBY = 49,              ! No. points to the Far-Field Boundary
       NINT = 89,               ! No. vertical points IN-beTween the sections
       NGAP = 32,               ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,           ! normal off Wall spacing
       DNFFB = 1.0,             ! normal spacing at the Far-Field Boundary
       DSSTG = 1.0E-03,         ! surface spacing at the STaGnation point
       DSTE = 2.0E-04,          ! surface spacing at the Trailing Edge
       DSLE = 5.0E-04,          ! surface spacing at any L.E. ctl line
```

```
       DSSHRP = 6.0E-04,          ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
       CLUSFL = 1,                ! CLUSter on FoiL # ..
       CLUSX = 0.0,               ! norm. arc length from LE (=0.0) to TE (=1.0)
       CLUSUL = 'u',              ! U for Upper surface, L for Lower surface
       CLUSDS = 2.0E-04,          ! CLUSter surface spacing (ds)
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
       SPCFFB = 0.2,              ! SPaCing ConTroL parameter for ffb
       SPCPSI = 0.01,             ! SPaCing ConTroL parameter for PSI lines
       SPCPHI = 0.02,             ! SPaCing ConTroL parameter for PHI lines
       ANGDECAY = 0.2             ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
bm4.dec                         ! Input decomposition file
bm4.spl                         ! Input spline file
bm4.dsb                         ! Input block side file
bm4.con                         ! Input connectivity file
bm4.agr                         ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                               ! graphics on/off
bm4.agr                             ! grid input file
plot3d                              ! grid format (tecplot or plot3d)
bm4.ctl                             ! boundary condition file
bm4.con                             ! block connectivity file
bm4.off                             ! offset file
bm4.egr                             ! output file in tecplot format
plot3d                              ! output file format (tecplot or plot3d)
4                                   ! number of blocks
1.0                                 ! SOR relaxation parameter (0<w<2)
2000                                ! Max iterations
1.0E-5                              ! Min residual for stopping
0.100000  0.1000000                 ! Relaxation Factors for P & Q
0.100000  0.1000000                 ! Limiting Factors for P & Q
```

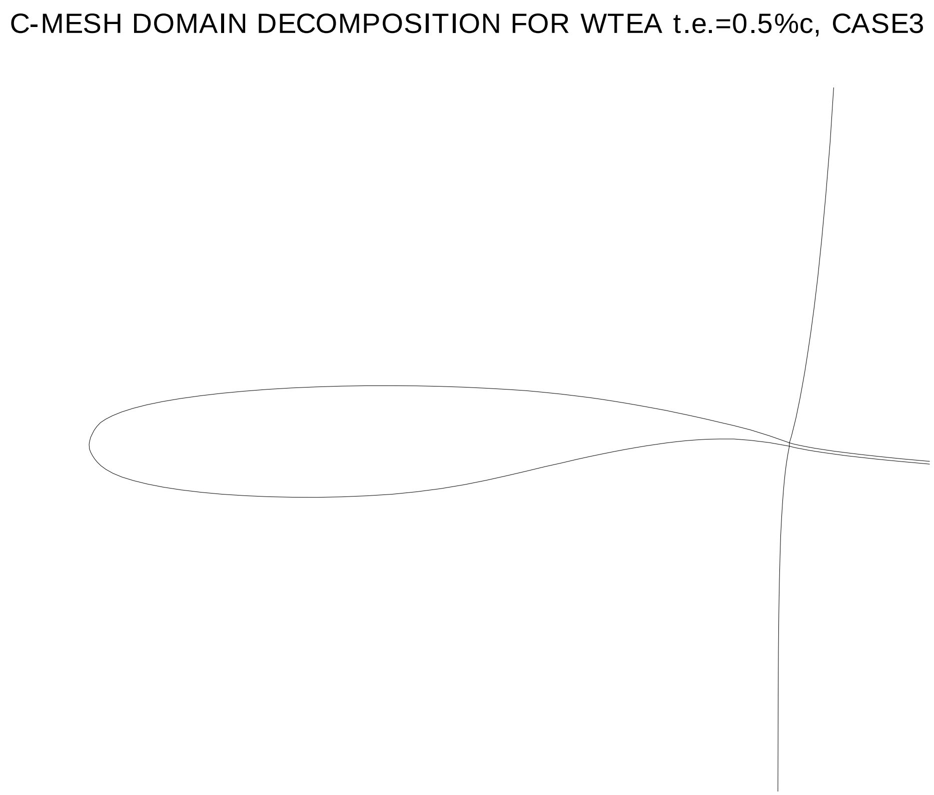

Figure 6.7: Domain decomposition for Test Case 3

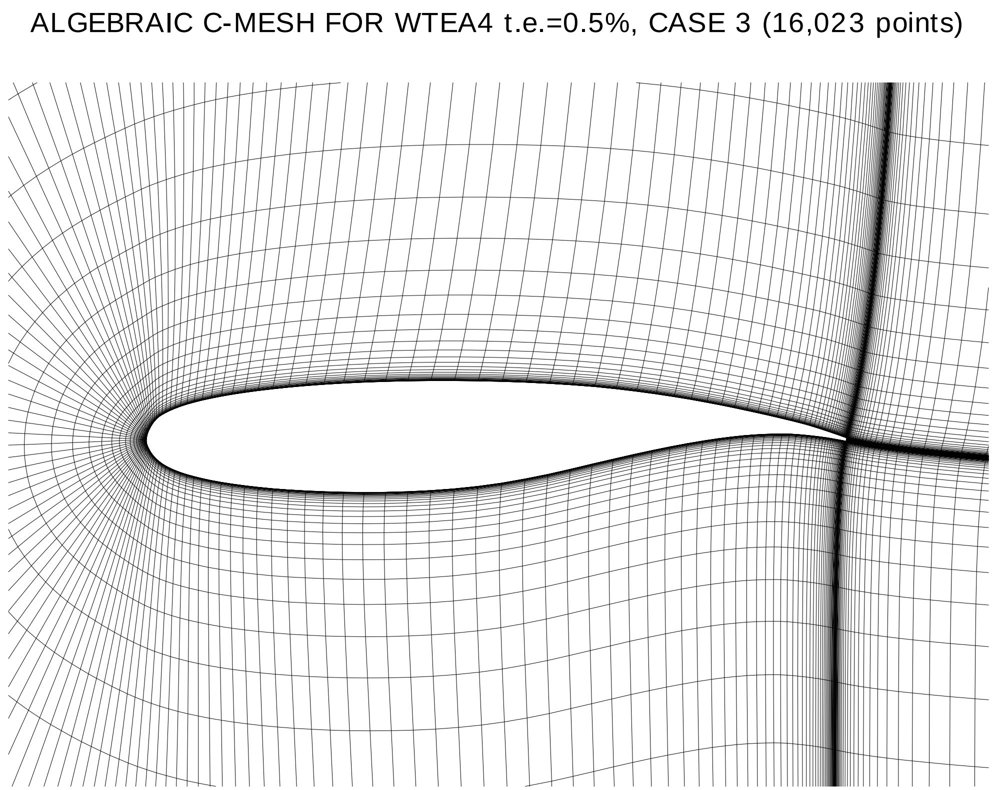

Figure 6.8: Algebraic grid for Test Case 3

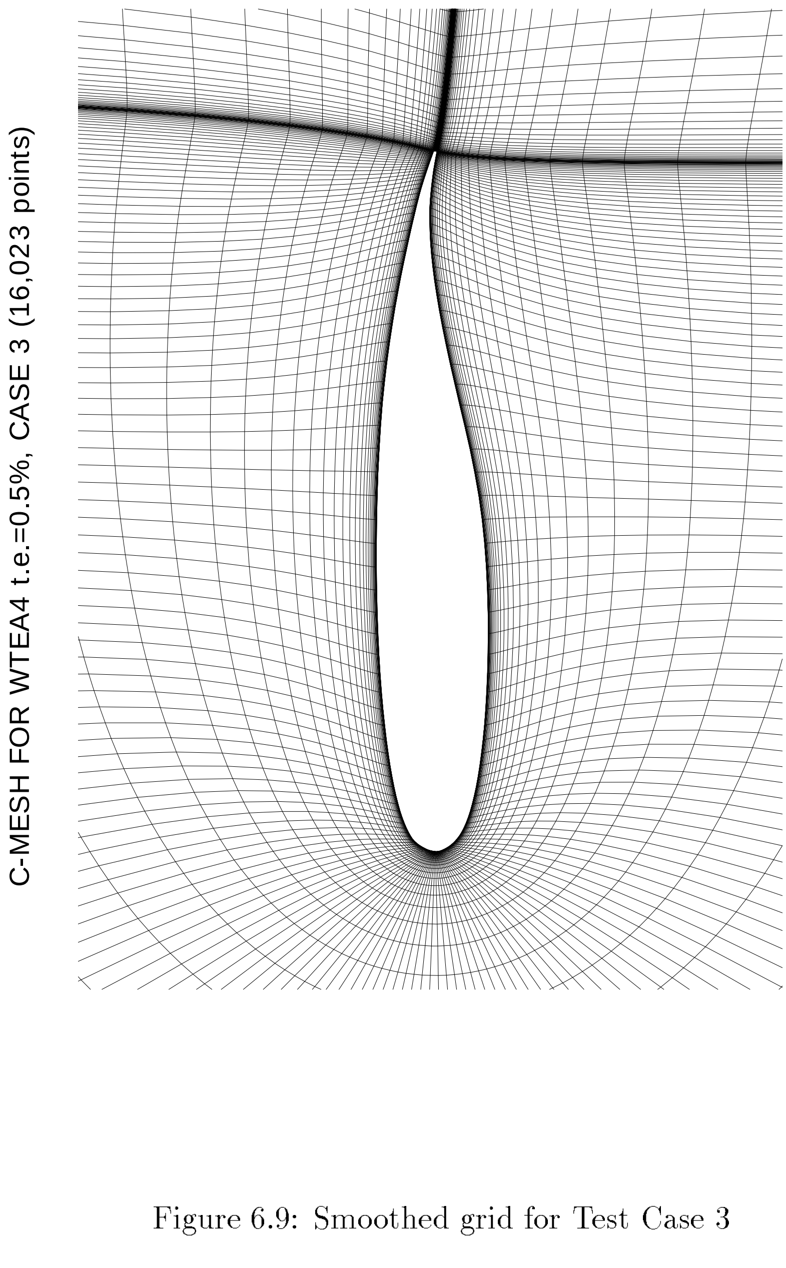

Figure 6.9: Smoothed grid for Test Case 3

Appendix D : WTEA4 t.e.=1.0%c

This Appendix contains example input and output for this single element DeHavilland WTEA t.e.=1.0%c test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.10)
5. Algebraic H-grid plot (figure 6.11)
6. Elliptically smoothed H-grid plot (figure 6.12)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! BENCHMARK #5 TEST CASE
 &INPUT
! AIRFOIL COORDINATES (NO DEFAULT) :
       FOIL='/usr/people/andrew/airfoils/benchmarks/bm5.dat'
! CONTROL PARAMETERS :
       ALPHA  = 0.0,             ! Angle of attack
       ROTATE  = 0.0,            ! Rotation angle
       X0 = 0.5,                 ! X coordinate of rotation centre
       Y0 = 0.0,                 ! Y coordinate of rotation centre
       LEPHI  = FALSE,           ! PHI-line from L.E. (t/f)
       CMESH  = TRUE,            ! C-MESH option (t/f)
       AUTOMESH = TRUE,          ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,            ! DECOMPose domain (t/f)
       MESHSEQ = 0,              ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'bm5.spl',      ! SPLine file (spline coefficients)
       DECFILE = 'bm5.dec',      ! DEComposition file (block definitions)
       CONFILE = 'bm5.con',      ! CONnectivity file (block connections)
       DSBFILE = 'bm5.dsb',      ! Define Sides of Block file
       CTLFILE = 'bm5.ctl',      ! ConTroL file (control fn' decay param.)
       P3DFILE = 'bm5.p3d',      ! Plot3D file for block boundaries
       STGFILE = 'bm5.stg',      ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -8.00000,           ! Left Hand Boundary location
       XRB =  9.00000,           ! Right Hand Boundary location
       YLB = -8.00000,           ! Lower Boundary location
       YUB =  8.00000,           ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 100,25,69,49,    ! No. PoinTS on AirFoil(s) in order from t.e.
                                 !   to any cluster points to t.e. going c.w.
       NTE = 29,                 ! No. points on the blunt Trailing Edge
       NFFBX = 49,               ! No. points to the Far-Field Boundary
       NFFBY = 49,               ! No. points to the Far-Field Boundary
       NINT = 89,                ! No. vertical points IN-beTween the sections
       NGAP = 32,                ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,            ! normal off Wall spacing
       DNFFB = 1.0,              ! normal spacing at the Far-Field Boundary
       DSSTG = 1.0E-03,          ! surface spacing at the STaGnation point
       DSTE = 2.0E-04,           ! surface spacing at the Trailing Edge
       DSLE = 5.0E-04,           ! surface spacing at any L.E. ctl line
       DSSHRP = 6.0E-04,         ! surface spacing at any SHaRp points
```

```
! MESH PARAMETERS - CLUSTERING :
       CLUSFL = 1,1,1,            ! CLUSter on FoiL # ..
       CLUSX = 0.0,0.035,0.64,   ! norm. arc length from LE (=0.0) to TE (=1.0)
       CLUSUL = 'u','u','u',     ! U for Upper surface, L for Lower surface
       CLUSDS = 2.0E-04,4.0E-04, ! CLUSter surface spacing (ds)
                2.0E-03,
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
       SPCFFB = 0.2,              ! SPaCing ConTroL parameter for ffb
       SPCPSI = 0.01,             ! SPaCing ConTroL parameter for PSI lines
       SPCPHI = 0.02,             ! SPaCing ConTroL parameter for PHI lines
       ANGDECAY = 0.2             ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
bm5.dec                         ! Input decomposition file
bm5.spl                         ! Input spline file
bm5.dsb                         ! Input block side file
bm5.con                         ! Input connectivity file
bm5.agr                         ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                               ! graphics on/off
bm5.agr                             ! grid input file
plot3d                              ! grid format (tecplot or plot3d)
bm5.ctl                             ! boundary condition file
bm5.con                             ! block connectivity file
bm5.off                             ! offset file
bm5.egr                             ! output file in tecplot format
plot3d                              ! output file format (tecplot or plot3d)
4                                   ! number of blocks
1.0                                 ! SOR relaxation parameter (0<w<2)
2000                                ! Max iterations
1.0E-5                              ! Min residual for stopping
0.100000  0.1000000                 ! Relaxation Factors for P & Q
0.100000  0.1000000                 ! Limiting Factors for P & Q
```

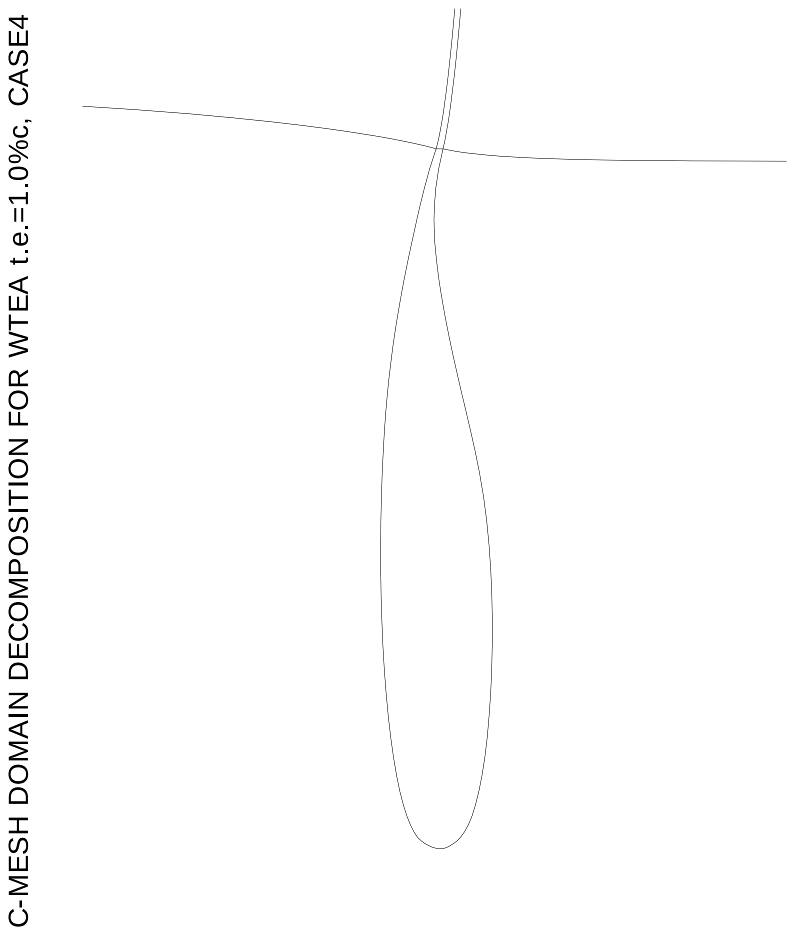

Figure 6.10: Domain decomposition for Test Case 4

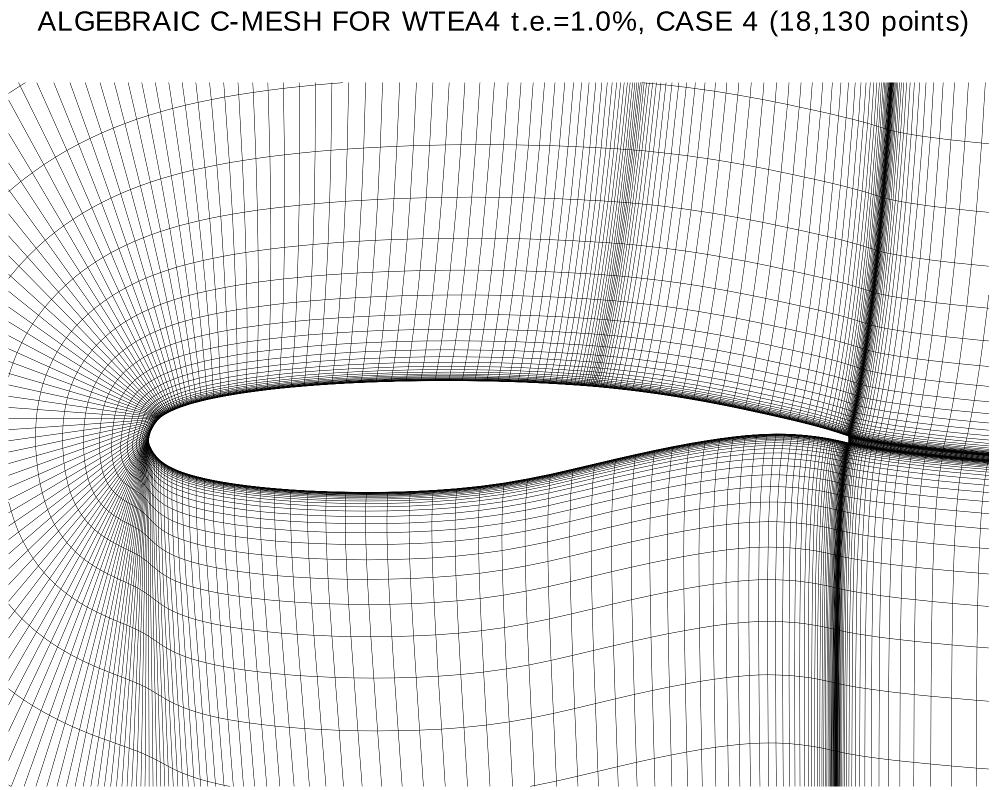

Figure 6.11: Algebraic grid for Test Case 4

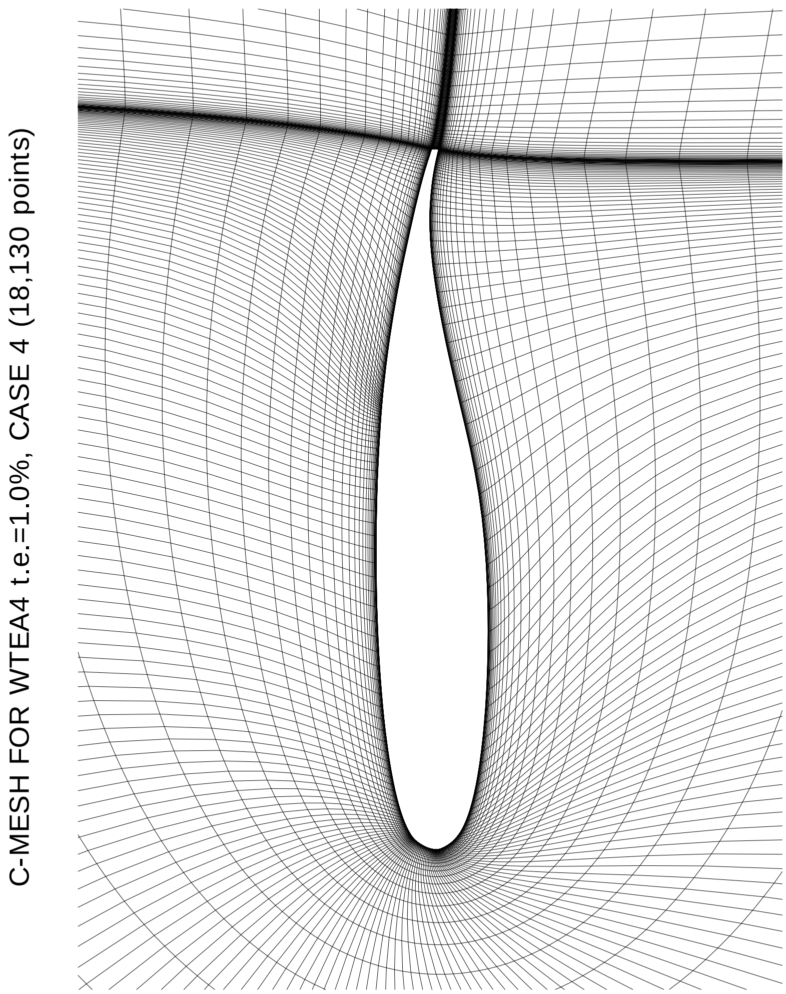

Figure 6.12: Smoothed grid for Test Case 4

Appendix E : WTEA4 t.e.=0.5%c / -20deg

This Appendix contains example input and output for this single element DeHavilland divergent blunt trailing edge WTEA t.e.=0.5%c / -20 degrees test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.13)
5. Algebraic H-grid plot (figure 6.14)
6. Elliptically smoothed H-grid plot (figure 6.15)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! DHC WTEA 0.5% BLUNT -20deg T.E. Test Case
!
 &INPUT
! AIRFOIL COORDINATES :
       FOIL='/usr/people/andrew/airfoils/wteaXX/wtea-20.dat'
! CONTROL PARAMETERS :
       ALPHA  = 0.0,             ! Angle of attack
       ROTATE  = 0.0,            ! Rotation angle
       X0 = 0.5,                 ! X coordinate of rotation centre
       Y0 = 0.0,                 ! Y coordinate of rotation centre
       LEPHI  = FALSE,           ! PHI-line from L.E. (t/f)
       CMESH  = TRUE,            ! C-MESH option (t/f)
       AUTOMESH = TRUE,          ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,            ! DECOMPose domain (t/f)
       MESHSEQ = 0,              ! max. number of MESH SEQuencing levels
! OUTPUT FILES :
       SPLFILE = 'wtea-20.spl',  ! SPLine file (spline coefficients)
       DECFILE = 'wtea-20.dec',  ! DEComposition file (block definitions)
       CONFILE = 'wtea-20.con',  ! CONnectivity file (block connections)
       DSBFILE = 'wtea-20.dsb',  ! Define Sides of Block file
       CTLFILE = 'wtea-20.ctl',  ! ConTroL file (control fn' decay param.)
       P3DFILE = 'wtea-20.p3d',  ! Plot3D file for block boundaries
       STGFILE = 'wtea-20.stg',  ! STaGnation point descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -8.00000,           ! Left Hand Boundary location
       XRB =  9.00000,           ! Right Hand Boundary location
       YLB = -8.00000,           ! Lower Boundary location
       YUB =  8.00000,           ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 100,100,         ! No. PoinTS on AirFoil(s) in order from t.e.
                                 !   to any cluster points to t.e. going c.w.
       NTE = 29,                 ! No. points on the blunt Trailing Edge
       NFFBX = 49,               ! No. points to the Far-Field Boundary
       NFFBY = 49,               ! No. points to the Far-Field Boundary
       NINT = 89,                ! No. vertical points IN-beTween the sections
       NGAP = 32,                ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 2.0E-05,            ! normal off Wall spacing
       DNFFB = 1.0,              ! normal spacing at the Far-Field Boundary
       DSSTG = 1.0E-03,          ! surface spacing at the STaGnation point
       DSTE = 2.0E-04,           ! surface spacing at the Trailing Edge
       DSLE = 5.0E-04,           ! surface spacing at any L.E. ctl line
```

```
       DSSHRP = 6.0E-04,          ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
       CLUSFL = 1,                ! CLUSter on FoiL # ..
       CLUSX = 0.0,               ! norm. arc length from LE (=0.0) to TE (=1.0)
       CLUSUL = 'u',              ! U for Upper surface, L for Lower surface
       CLUSDS = 2.0E-04,          ! CLUSter surface spacing (ds)
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
       SPCFFB = 0.2,              ! SPaCing ConTroL parameter for ffb
       SPCPSI = 0.01,             ! SPaCing ConTroL parameter for PSI lines
       SPCPHI = 0.02,             ! SPaCing ConTroL parameter for PHI lines
       ANGDECAY = 0.2             ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
wtea-20.dec                        ! Input decomposition file
wtea-20.spl                        ! Input spline file
wtea-20.dsb                        ! Input block side file
wtea-20.con                        ! Input connectivity file
wtea-20.agr                        ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                              ! graphics on/off
wtea-20.agr                        ! grid input file
plot3d                             ! grid format (tecplot or plot3d)
wtea-20.ctl                        ! boundary condition file
wtea-20.con                        ! block connectivity file
wtea-20.off                        ! offset file
wtea-20.g                          ! output file in tecplot format
plot3d                             ! output file format (tecplot or plot3d)
4                                  ! number of blocks
1.0                                ! SOR relaxation parameter (0<w<2)
2000                               ! Max iterations
1.0E-5                             ! Min residual for stopping
0.100000  0.1000000                ! Relaxation Factors for P & Q
0.100000  0.1000000                ! Limiting Factors for P & Q
```

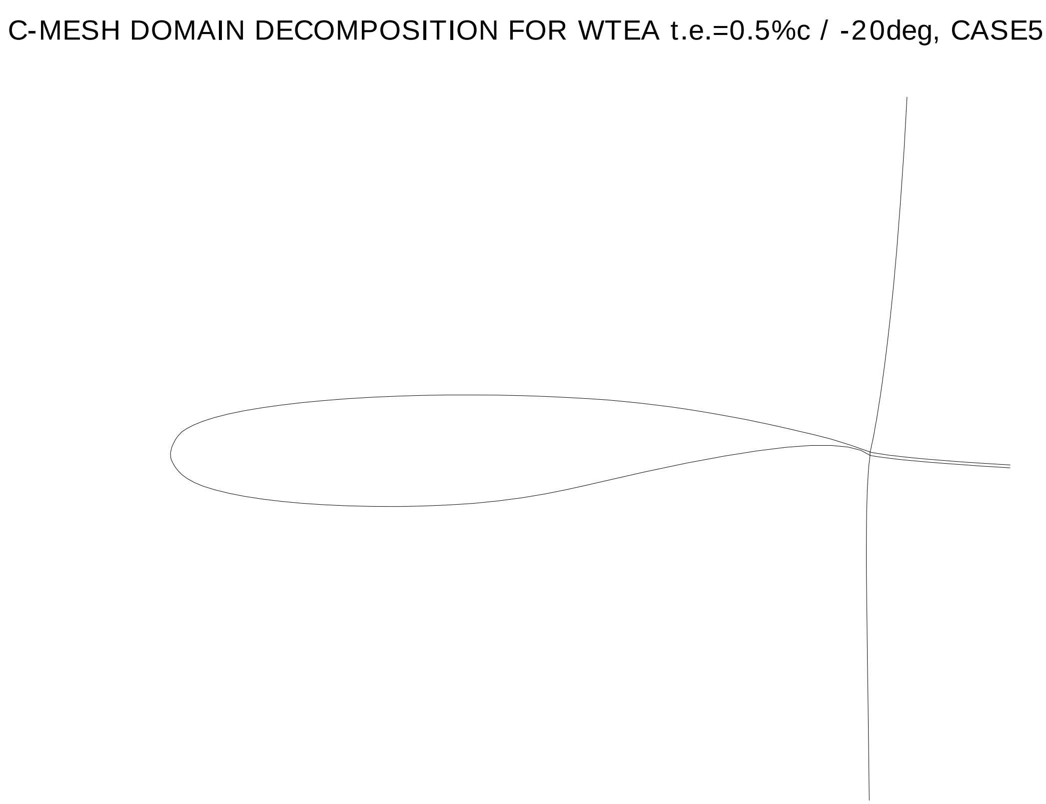

Figure 6.13: Domain decomposition for Test Case 5

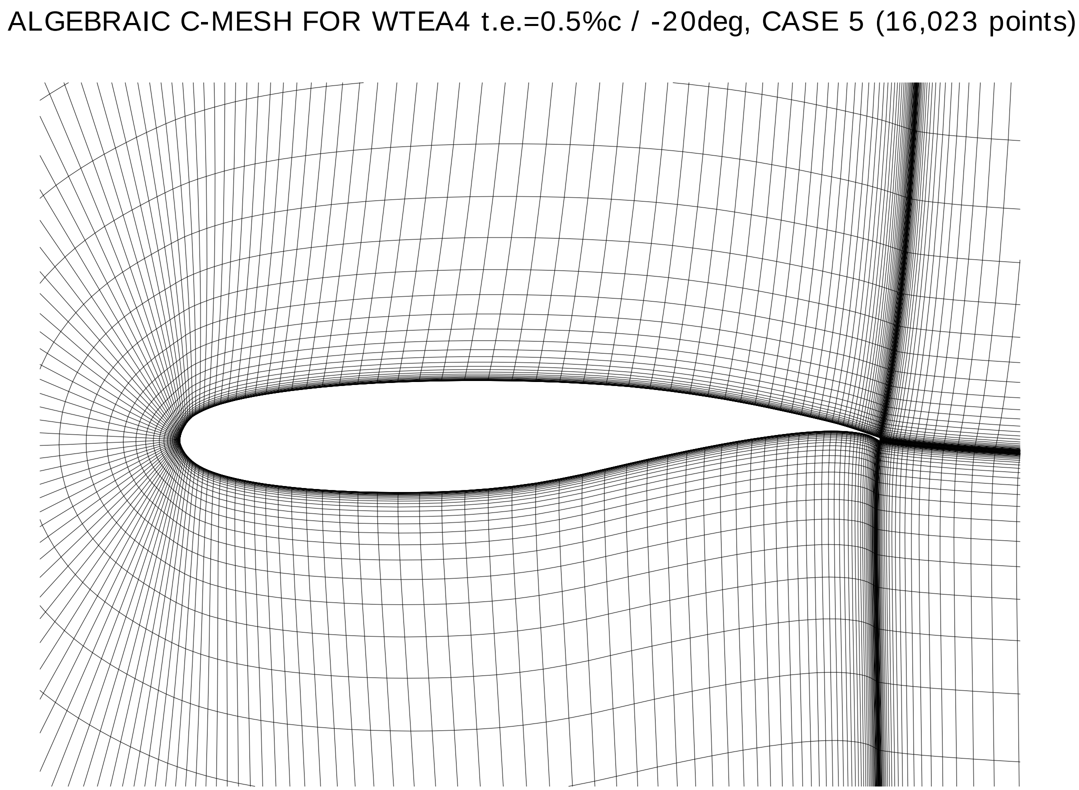

Figure 6.14: Algebraic grid for Test Case 5

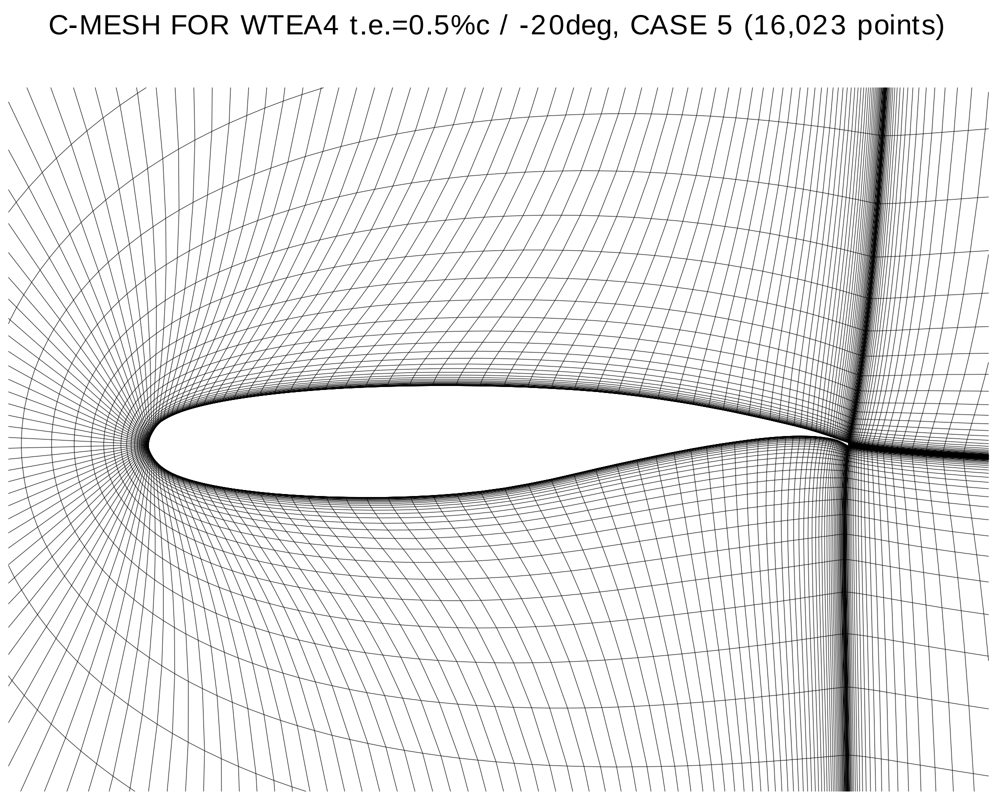

Figure 6.15: Smoothed grid for Test Case 5

Appendix F : DHC WTCT AUGMENTED LIFT AIRFOIL

This Appendix contains example input and output for the three element DeHavilland WTCT augmented lift airfoil test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Algebraic grid generation control file : *agrid.inp*
3. Multiblock elliptic smoother control file : *megrid.inp*
4. Domain decomposition plot (figure 6.16)
5. Algebraic H-grid plot (figure 6.17)
6. Elliptically smoothed H-grid plot (figure 6.18)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! DHC AUGMENTED LIFT SECTION
!
 &INPUT
! AIRFOIL COORDINATES :
!
  FOIL='/usr/people/andrew/airfoils/augwing/wtct.dat',
!
! CONTROL PARAMETERS :
!
  ALPHA  = 0.0,              ! Angle of attack
  ROTATE  = 0.0,             ! Rotation angle
  X0 = 0.5,                  ! X coordinate of rotation centre
  Y0 = 0.0,                  ! Y coordinate of rotation centre
  LEPHI  = FALSE,FALSE,
           FALSE,            ! Leading Edge PHI lines (t/f)
  CMESH  = FALSE,            ! C-MESH option (t/f)
  AUTOMESH = TRUE,           ! AUTOMESH Option (.dsb & .ctl files) (t/f)
  DECOMP = TRUE,             ! DECOMPose domain (t/f)
  MESHSEQ = 0,               ! max. number of MESH SEQuencing levels
!
! OUTPUT FILES :
!
  SPLFILE = 'wtct.spl',      ! SPLine file (spline coefficients)
  DECFILE = 'wtct.dec',      ! DEComposition file (block definitions)
  CONFILE = 'wtct.con',      ! CONnectivity file (block connections)
  DSBFILE = 'wtct.dsb',      ! Define Sides of Block file
  CTLFILE = 'wtct.ctl',      ! ConTroL file (control fn' decay param.)
  P3DFILE = 'wtct.p3d',      ! Plot3D file for block boundary definitions
  STGFILE = 'wtct.stg',      ! STaGnation point descriptors FILE
!
! BOUNDARY PLACEMENT :
!
  XLB = -10.00000,           ! Left Hand Boundary location
  XRB =  10.00000,           ! Right Hand Boundary location
  YLB = -10.00000,           ! Lower Boundary location
  YUB =  10.00000,           ! Upper Boundary location
!
! MESH PARAMETERS - NODES :
!
  NPTSAF = 209,209,209,       ! No. points on the AirFoil(s)
  NTE = 39,29,29,             ! No. points on the blunt Trailing Edge
  NFFBX = 59,                 ! No. points to the Far-Field Boundary
```

```
  NFFBY = 59,                 ! No. points to the Far-Field Boundary
  NINT = 49,                  ! No. vertical points IN-beTween the sections
  NGAP = 30,                  ! No. horiz. points in negative overlap (GAP)
!
! MESH PARAMETERS - SPACINGS :
!
  DNW = 4.0E-05,4.0E-05,
        4.0E-05,              ! normal off Wall spacing
  DNFFB = 2.0,                ! normal spacing at the Far-Field Boundary
  DSSTG = 2.0E-04,2.0E-04,
          2.0E-04,            ! surface spacing at the STaGnation point
  DSTE = 1.0E-04,1.0E-04,
         1.0E-04,             ! surface spacing at the Trailing Edge
  DSLE = 5.0E-04,             ! surface spacing at any L.E. ctl line
  DSSHRP = 2.5E-04,           ! surface spacing at any SHaRp points
!
! MESH PARAMETERS - CLUSTERING :
!
! CLUSFL =                    ! CLUSter on FoiL # ..
! CLUSX =                     ! (X/C) coordinate of CLUSter point
! CLUSUL =                    ! U for Upper surface, L for Lower surface
! CLUSDS =                    ! CLUSter surface spacing (ds)
!
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
!
  SPCFFB = 0.2,              ! SPaCing ConTroL parameter for ffb
  SPCPSI = 0.01,             ! SPaCing ConTroL parameter for PSI lines
  SPCPHI = 0.02,             ! SPaCing ConTroL parameter for PHI lines
  ANGDECAY = 0.2             ! ANGle ConTroL parameter
                                &END
```

Algebraic grid generation control file: agrid.inp

```
wtct.dec                            ! Input decomposition file
wtct.spl                            ! Input spline file
wtct.dsb                            ! Input block side file
wtct.con                            ! Input connectivity file
wtct.agr                            ! Output file agrid file
```

Multiblock elliptic smoother control file: megrid.inp

```
false                                 ! graphics on/off
wtct.agr                              ! grid input file
plot3d                                ! grid format (tecplot or plot3d)
wtct.ctl                              ! boundary condition file
wtct.con                              ! block connectivity file
wtct.off                              ! offset file
wtct.g                                ! output file in tecplot format
plot3d                                ! output file format (tecplot or plot3d)
27                                    ! number of blocks
1.0                                   ! SOR relaxation parameter (0<w<2)
200                                   ! Max iterations
1.0E-5                                ! Min residual for stopping
0.100000  0.1000000                   ! Relaxation Factors for P & Q
0.100000  0.1000000                   ! Limiting Factors for P & Q
```

H-MESH DOMAIN DECOMPOSITION FOR DHC AUGMENTED LIFT AIRFOIL, CASE6

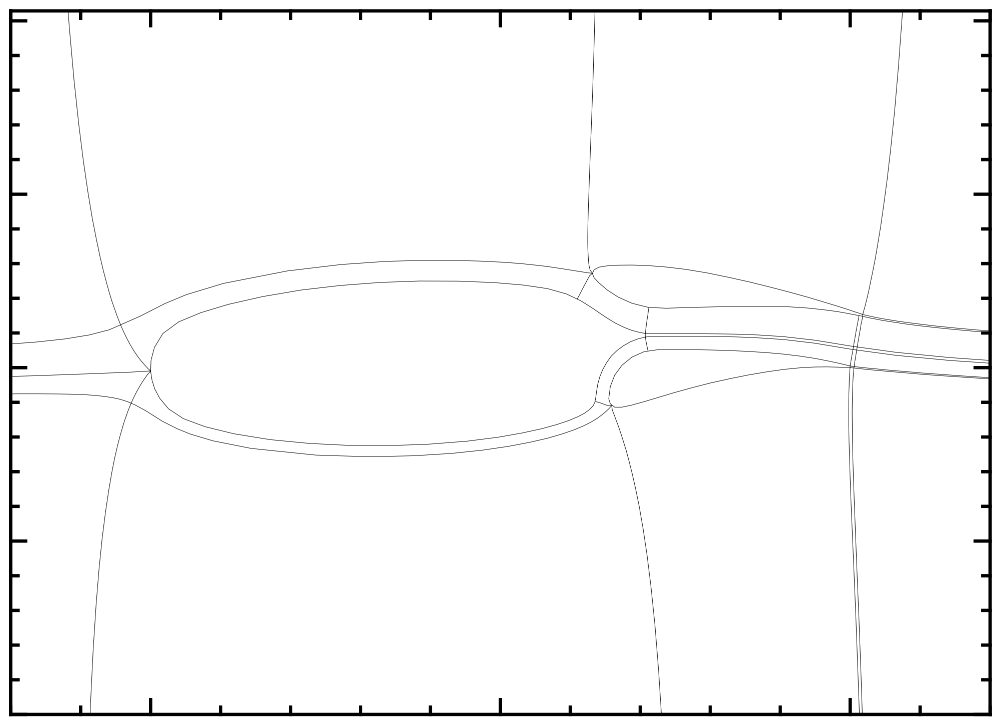

Figure 6.16: Domain decomposition for Test Case 6

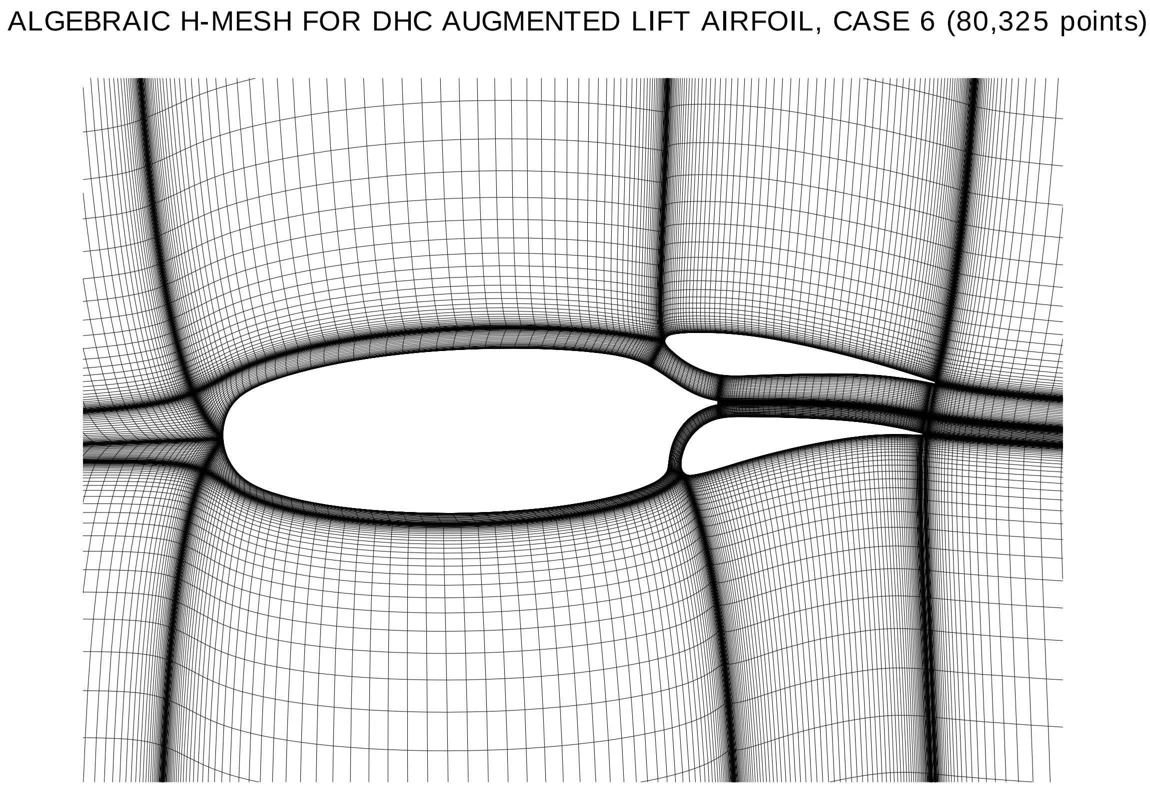

Figure 6.17: Algebraic grid for Test Case 6

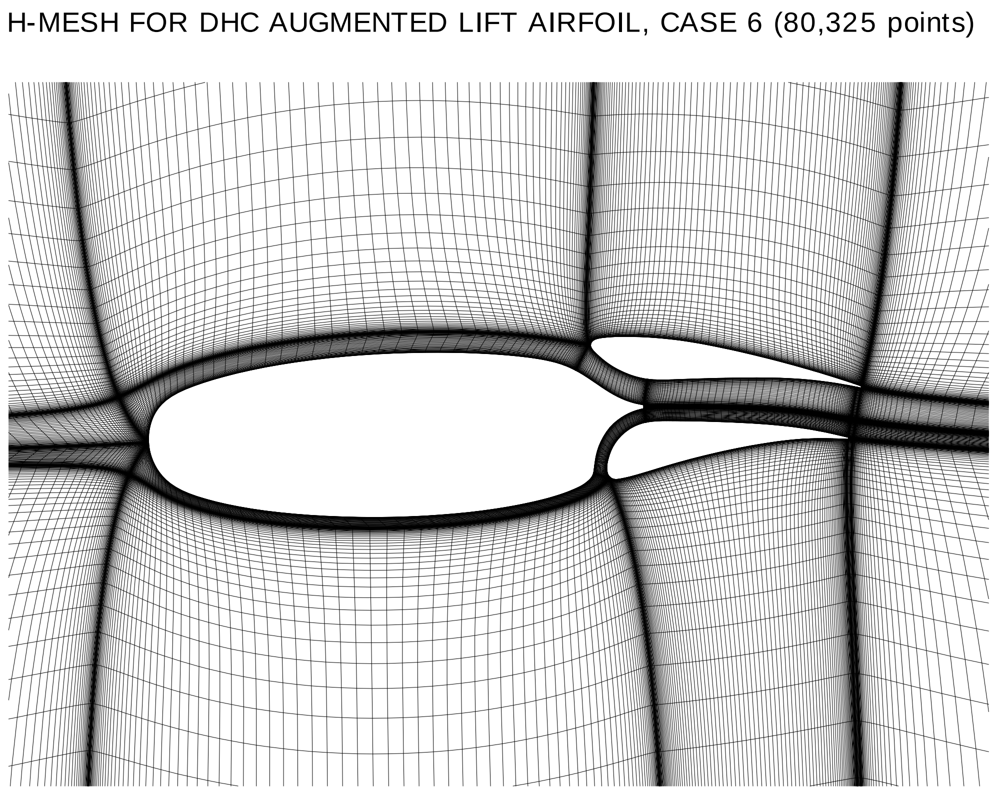

Figure 6.18: Smoothed grid for Test Case 6

Appendix G : WILLIAMS A

This Appendix contains example input and output for the double element Williams A test case. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Airfoil coordinate data file
3. Domain decomposition plots (figures 6.19 and 6.20)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! WILLIAMS TEST CASE A
!
 &INPUT
! AIRFOIL COORDINATES :
       FOIL='/usr/people/andrew/airfoils/williams/williams_a.dat',
! CONTROL PARAMETERS :
       ALPHA  = 2.0,              ! Angle of attack
       ROTATE  = 0.0,             ! Rotation angle
       X0 = 0.5,                  ! X coordinate of rotation centre
       Y0 = 0.0,                  ! Y coordinate of rotation centre
       LEPHI  = FALSE,TRUE,       ! PHI-line from L.E. (t/f)
       CMESH  = FALSE,            ! C-MESH option (t/f)
       AUTOMESH = TRUE,           ! AUTOMESH Option (.dsb & .ctl files) (t/f)
       DECOMP = TRUE,             ! DECOMPose domain (t/f)
! OUTPUT FILES :
       SPLFILE = 'willy_a.spl',  ! SPLine file (spline coefficients)
       DECFILE = 'willy_a.dec',  ! DEComposition file (block definitions)
       CONFILE = 'willy_a.con',  ! CONnectivity file (block connections)
       DSBFILE = 'willy_a.dsb',  ! Define Sides of Block file
       CTLFILE = 'willy_a.ctl',  ! ConTroL file (control fn' decay param.)
       P3DFILE = 'willy_a.p3d',  ! Plot3D FILE for block boundaries
       STGFILE = 'willy_a.stg',  ! SinGular poinT descriptor FILE
! BOUNDARY PLACEMENT :
       XLB = -10.00000,           ! Left Hand Boundary location
       XRB =  10.00000,           ! Right Hand Boundary location
       YLB = -10.00000,           ! Lower Boundary location
       YUB =  10.00000,           ! Upper Boundary location
! MESH PARAMETERS - NODES :
       NPTSAF = 249,149,29,       ! No. points on the AirFoil(s)
       NTE = 0,0,                 ! No. points on the blunt Trailing Edge
       NFFBX = 49,                ! No. points to the Far-Field Boundary in X
       NFFBY = 49,                ! No. points to the Far-Field Boundary in Y
       NINT = 40,                 ! No. vertical points IN-beTween the sections
       NGAP = 30,                 ! No. horiz. points in negative overlap (GAP)
! MESH PARAMETERS - SPACINGS :
       DNW = 4.0E-05,4.0E-05,     ! normal off Wall spacing
       DNFFB = 2.0,               ! normal spacing at the Far-Field Boundary
       DSSTG = 2.0E-04,2.0E-04,  ! surface spacing at the STaGnation point
       DSTE = 1.0E-04,1.0E-04,   ! surface spacing at the Trailing Edge
       DSLE = 2.0E-04,            ! surface spacing at any L.E. ctl line
       DSSHRP = 4.0E-04,          ! surface spacing at any SHaRp points
! MESH PARAMETERS - CLUSTERING :
```

```
       CLUSFL = 1,                ! CLUSter on FoiL # ..
       CLUSX = 0.75,              ! (X/C) coordinate of CLUSter point
       CLUSUL = 'u',              ! U for Upper surface, L for Lower surface
       CLUSDS = 2.0E-04,          ! CLUSter surface spacing (ds)
! MESH PARAMETERS - CONTROL FUNCTION DECAY PARAMETERS :
       SPCFFB = 0.5,              ! SPaCing ConTroL parameter for ffb
       SPCPSI = 0.005,            ! SPaCing ConTroL parameter for PSI lines
       SPCPHI = 0.01,             ! SPaCing ConTroL parameter for PHI lines
       ANGDECAY = 0.2             ! ANGle ConTroL parameter
                                &END
```

Airfoil coordinate data file:

```
!    AIRFOIL DATA FOR WILLIAM'S TEST CASE A
!
*NAME=MAIN
!   X          Y          CP
   1.00000   0.00590   1.00000
   0.99931   0.00612  -0.04022
   0.99417   0.00748   0.23687
   0.98434   0.00903   0.50061
   0.96975   0.00941   0.67369
   0.94998   0.00766   0.75477
   0.92461   0.00361   0.77689
   0.89358  -0.00236   0.76914
   0.85728  -0.00965   0.74766
   0.81639  -0.01771   0.72058
   0.77169  -0.02612   0.69205
   0.72396  -0.03451   0.66437
   0.67396  -0.04261   0.63898
   0.62240  -0.05018   0.61699
   0.56993  -0.05699   0.59936
   0.51716  -0.06287   0.58708
   0.46466  -0.06766   0.58115
   0.41297  -0.07214   0.58264
   0.36259  -0.07350   0.59271
   0.31398  -0.07438   0.61252
   0.26759  -0.07386   0.64327
   0.22381  -0.07194   0.68596
   0.18304  -0.06866   0.74118
   0.14563  -0.06409   0.80830
   0.11190  -0.05833   0.88385
```

```
   0.08214  -0.05151   0.95724
   0.05663  -0.04378   0.99969
   0.03560  -0.03530   0.93296
   0.01927  -0.02625   0.53705
   0.00783  -0.01681  -0.79779
   0.00143  -0.00714  -4.20249
   0.00017   0.00264  -8.34989
   0.00409   0.01242  -8.73166
   0.01311   0.02211  -7.14534
   0.02707   0.03155  -5.73037
   0.04582   0.04056  -4.73940
   0.06914   0.04898  -4.05084
   0.09681   0.05663  -3.55471
   0.12857   0.06335  -3.18166
   0.16414   0.06902  -2.88974
   0.20321   0.07352  -2.65315
   0.24543   0.07678  -2.45564
   0.29044   0.07875  -2.28674
   0.33785   0.07942  -2.13965
   0.38724   0.07881  -2.00992
   0.43814   0.07700  -1.89477
   0.49010   0.07408  -1.79260
   0.54258   0.07019  -1.70272
   0.59507   0.06550  -1.62525
   0.64697   0.06020  -1.56109
   0.69769   0.05453  -1.51204
   0.74656   0.04870  -1.48106
   0.79290   0.04293  -1.47265
   0.83597   0.03743  -1.49342
   0.87501   0.03232  -1.55225
   0.90929   0.02762  -1.65810
   0.93815   0.02323  -1.80900
   0.96122   0.01893  -1.95727
   0.97850   0.01458  -1.95374
   0.99043   0.01041  -1.60169
   0.99753   0.00718  -0.02119
   1.00000   0.00590   1.00000
*EOR
*NAME=FLAP
!    X          Y          CP
   1.31389  -0.20363   1.00000
   1.31360  -0.20335   0.61683
   1.31121  -0.20083   0.62318
   1.30635  -0.19598   0.64997
```

1.29886	-0.18893	0.67827
1.28864	-0.17996	0.70420
1.27564	-0.16939	0.72668
1.25995	-0.15765	0.74560
1.24177	-0.14518	0.76122
1.22146	-0.13243	0.77401
1.19948	-0.11982	0.78457
1.17640	-0.10766	0.79358
1.15285	-0.09619	0.80184
1.12944	-0.08553	0.81022
1.10676	-0.07572	0.81976
1.08535	-0.06674	0.83159
1.06565	-0.05854	0.84699
1.04799	-0.05105	0.86722
1.03263	-0.04423	0.89341
1.01972	-0.03807	0.92597
1.00930	-0.03258	0.96308
1.00134	-0.02781	0.99520
0.99572	-0.02381	0.98154
0.99226	-0.02065	0.71658
0.99073	-0.01835	-1.17476
0.99087	-0.01686	-5.75997
0.99242	-0.01604	-2.85918
0.99508	-0.01571	-1.43049
0.99864	-0.01569	-0.89891
1.00295	-0.01582	-0.70367
1.00797	-0.01598	-0.68332
1.01372	-0.01607	-0.77990
1.02027	-0.01609	-0.96299
1.02768	-0.01606	-1.20757
1.03600	-0.01610	-1.48844
1.04527	-0.01631	-1.78052
1.05548	-0.01684	-2.06103
1.06658	-0.01785	-2.31124
1.07852	-0.01946	-2.51744
1.09119	-0.02181	-2.67093
1.10447	-0.02499	-2.76751
1.11824	-0.02909	-2.80664
1.13235	-0.03415	-2.79067
1.14667	-0.04020	-2.72396
1.16103	-0.04725	-2.61232
1.17532	-0.05525	-2.46235
1.18938	-0.06416	-2.28112
1.20310	-0.07391	-2.07571

```
   1.21637  -0.08439  -1.85307
   1.22907  -0.09548  -1.61970
   1.24112  -0.10704  -1.38159
   1.25245  -0.11891  -1.14405
   1.26298  -0.13090  -0.91168
   1.27267  -0.14281  -0.68829
   1.28147  -0.15440  -0.47687
   1.28934  -0.16544  -0.27961
   1.29624  -0.17566  -0.09789
   1.30214  -0.18476   0.06777
   1.30697  -0.19245   0.21788
   1.31064  -0.19840   0.35456
   1.31303  -0.20226   0.48483
   1.31389  -0.20363   1.00000
*EOF
```

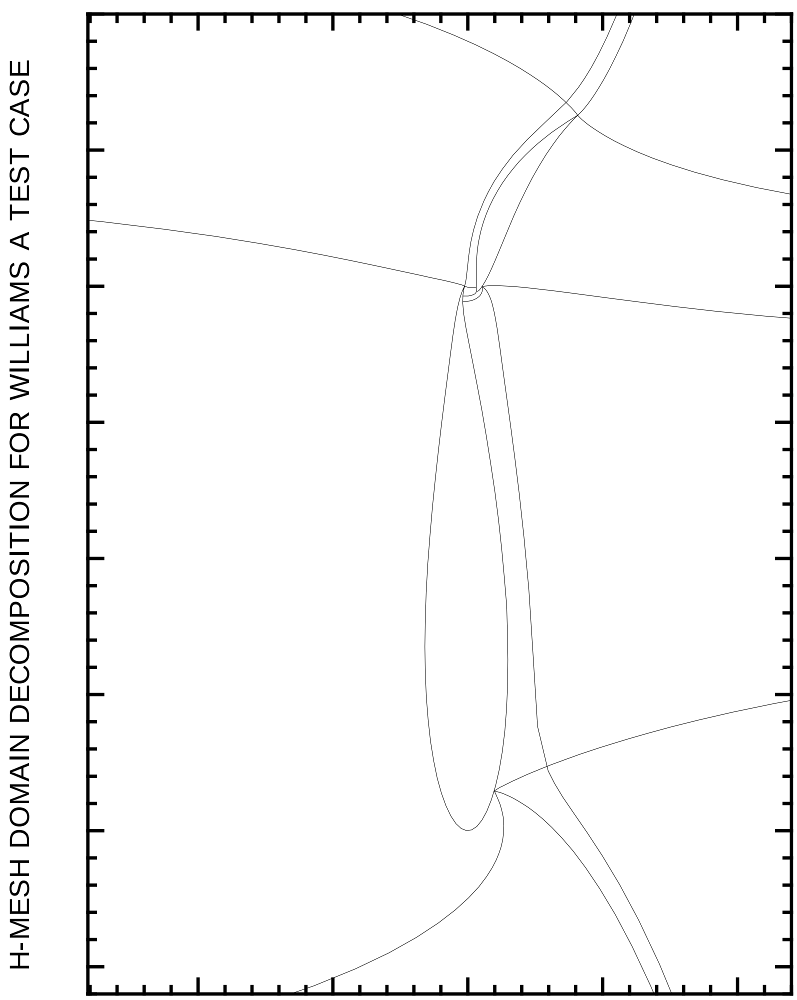

Figure 6.19: Domain decomposition for Test Case 7

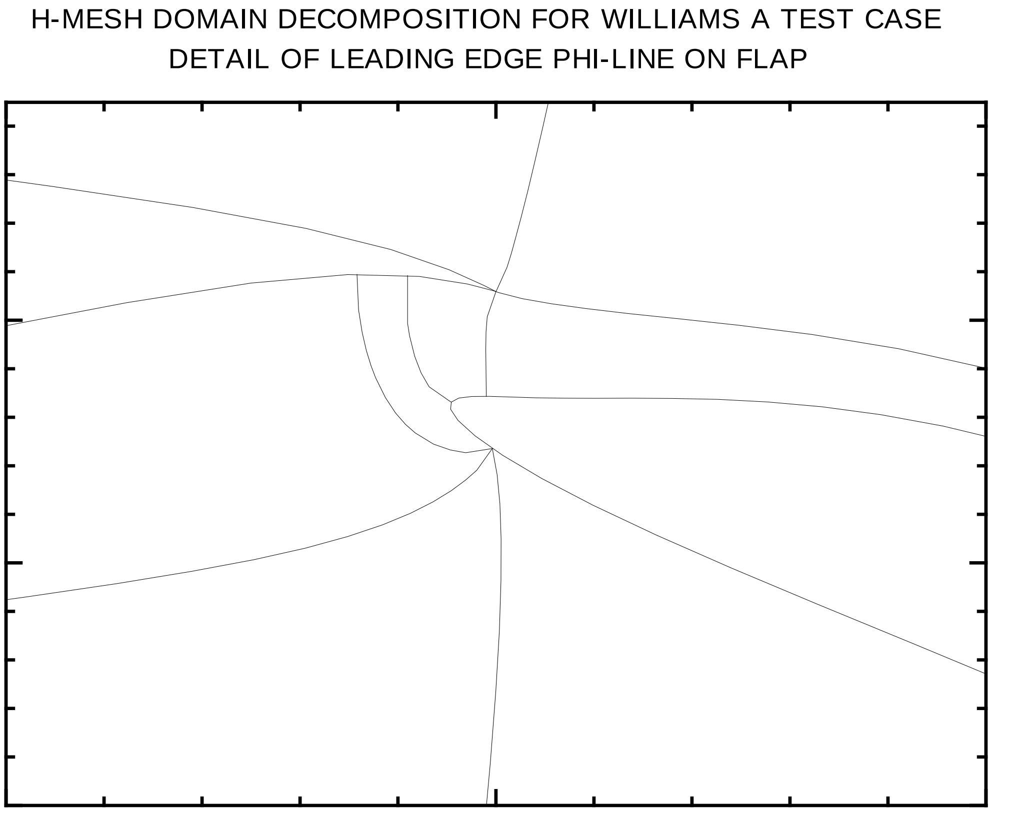

Figure 6.20: Domain decomposition for Test Case 7 - detail of flap

Appendix H : NACA0012 WITH CONCAVE T.E.

This Appendix contains example input and output for the single element NACA 0012 test case with a blunt concave trailing edge. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Airfoil coordinate data file
3. Domain decomposition plots (figures 6.21 and 6.22)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! NACA 0012 AIRFOIL WITH BLUNT CONCAVE TRAILING EDGE
!
 &INPUT   FOIL='/usr/people/andrew/airfoils/naca0012/n0012te.dat',
! OUTPUT FILES :
          SPLFILE = 'b12.spl',
          DECFILE = 'b12.dec',
          CONFILE = 'b12.con',
          DSBFILE = 'b12.dsb',
          CTLFILE = 'b12.ctl',
          P3DFILE = 'b12.p3d',
          STGFILE = 'b12.stg',
! BOUNDARY PLACEMENT :
          XLB = -5.00000,
          XRB =  6.00000,
          YLB = -5.00000,
          YUB =  5.00000       &END
```

Airfoil coordinate data file:

```
!  Airfoil coordinates for NACA 0012
!
*NAME=NACA12
1.0000    -0.00126
0.9500    -0.00807
0.9000    -0.01448
0.8000    -0.02623
0.7000    -0.03664
0.6000    -0.04563
0.5000    -0.05294
0.4000    -0.05803
0.3000    -0.06002
0.2500    -0.05941
0.2000    -0.05737
0.1500    -0.05345
0.1000    -0.04683
0.0750    -0.04200
0.0500    -0.03555
0.0250    -0.02615
0.0125    -0.01894
0.0000     0.00000
```

```
0.0125     0.01894
0.0250     0.02615
0.0500     0.03555
0.0750     0.04200
0.1000     0.04683
0.1500     0.05345
0.2000     0.05737
0.2500     0.05941
0.3000     0.06002
0.4000     0.05803
0.5000     0.05294
0.6000     0.04563
0.7000     0.03664
0.8000     0.02623
0.9000     0.01448
0.9500     0.00807
1.0000     0.00126
!
! Coordinates of the blunt trailing edge :
!
*TE=NACA12
1.0000     0.00126
0.9991     0.00089
0.9987     0.00000
0.9991    -0.00089
1.0000    -0.00126
*EOF
```

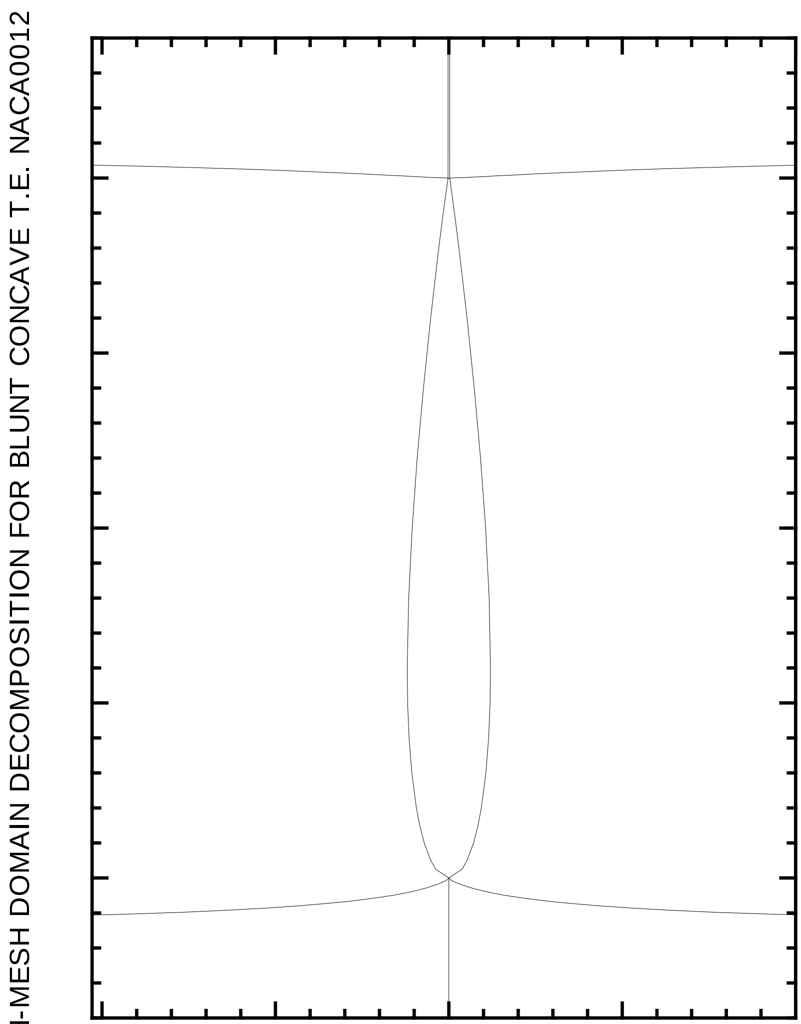

Figure 6.21: Domain decomposition for Test Case 8

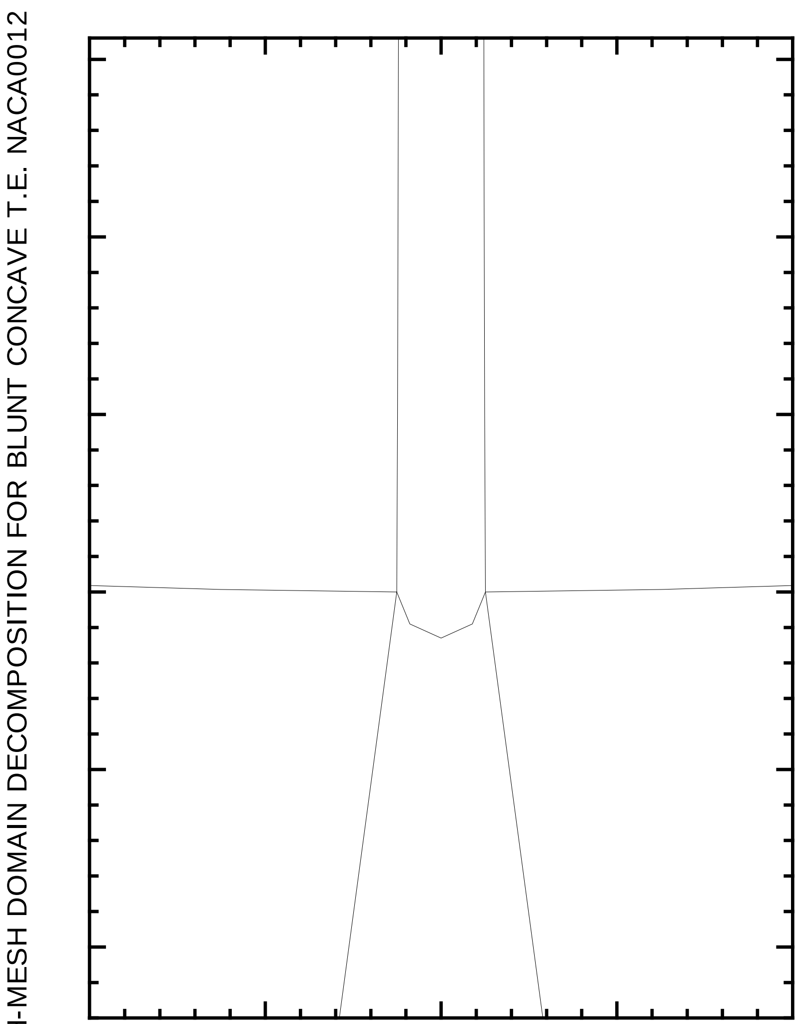

Figure 6.22: Domain decomposition for Test Case 8 - detail of trailing edge

Appendix I : NACA0012 WITH CUT OUT

This Appendix contains example input and output for the single element NACA 0012 test case with a blunt trailing edge and flap cut out. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Airfoil coordinate data file
3. Domain decomposition plot (figure 6.23)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! NACA 0012 AIRFOIL WITH BLUNT TRAILING EDGE and a flap cutout
!
 &INPUT   FOIL='/usr/people/andrew/airfoils/naca0012/n12co.dat',
!
         SHRPFL =  1, 1,
         SHRPPT = 5, 10,
! OUTPUT FILES :
         SPLFILE = 'n12co.spl',
         DECFILE = 'n12co.dec',
         CONFILE = 'n12co.con',
         DSBFILE = 'n12co.dsb',
         CTLFILE = 'n12co.ctl',
         P3DFILE = 'n12co.p3d',
         STGFILE = 'n12co.stg',
! BOUNDARY PLACEMENT :
         XLB = -10.00000,
         XRB =  10.00000,
         YLB = -10.00000,
         YUB =  10.00000       &END
```

Airfoil coordinate data file:

```
!  Airfoil coordinates for NACA 0012
!
*NAME=NACA12
1.0000    -0.00126
0.9500     0.00107
0.9000     0.00448
0.8000     0.01223
0.7500     0.01464
0.7400     0.00438
0.7300    -0.00587
0.7200    -0.01613
0.7100    -0.02638
0.7000    -0.03664
0.6000    -0.04563
0.5000    -0.05294
0.4000    -0.05803
0.3000    -0.06002
0.2500    -0.05941
0.2000    -0.05737
0.1500    -0.05345
0.1000    -0.04683
0.0750    -0.04200
0.0500    -0.03555
0.0250    -0.02615
0.0125    -0.01894
0.0000     0.00000
0.0125     0.01894
0.0250     0.02615
0.0500     0.03555
0.0750     0.04200
0.1000     0.04683
0.1500     0.05345
0.2000     0.05737
0.2500     0.05941
0.3000     0.06002
0.4000     0.05803
0.5000     0.05294
0.6000     0.04563
0.7000     0.03664
0.8000     0.02623
0.9000     0.01448
0.9500     0.00807
1.0000     0.00126
```

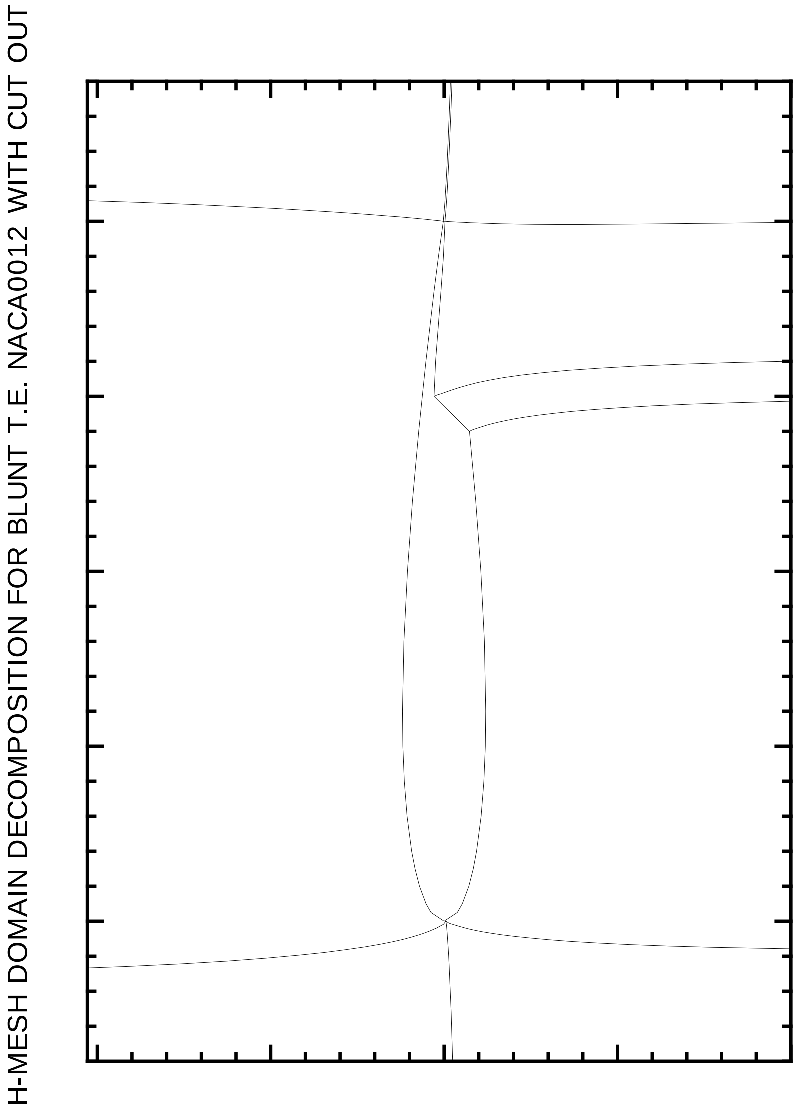

Figure 6.23: Domain decomposition for Test Case 9

Appendix J : NACA0012 WITH UPPER STEP

This Appendix contains example input and output for the single element NACA 0012 test case with a blunt trailing edge and an upper surface step. More specifically, the following are given:

1. Domain decomposition control file : *kmS.inp*
2. Airfoil coordinate data file
3. Domain decomposition plots (figures 6.24 and 6.25)

kmS.inp:

```
! CONTROL FILE FOR K-M DOMAIN DECOMPOSITION
! NACA 0012 AIRFOIL WITH BLUNT TRAILING EDGE and an upper surface step
!
 &INPUT   FOIL='/usr/people/andrew/airfoils/naca0012/n12step-.dat',
!
         SHRPFL =  1, 1,
         SHRPPT = 31, 32,
! OUTPUT FILES :
         SPLFILE = 'n12step.spl',
         DECFILE = 'n12step.dec',
         CONFILE = 'n12step.con',
         DSBFILE = 'n12step.dsb',
         CTLFILE = 'n12step.ctl',
         P3DFILE = 'n12step.p3d',
         STGFILE = 'n12step.stg',
! BOUNDARY PLACEMENT :
         XLB = -10.00000,
         XRB =  10.00000,
         YLB = -10.00000,
         YUB =  10.00000       &END
```

Airfoil coordinate data file:

```
!  Airfoil coordinates for NACA 0012
!  Step at 70% of 0.1% outward
!
*NAME=NACA12
1.0000    -0.00126
0.9500    -0.00807
0.9000    -0.01448
0.8000    -0.02623
0.7000    -0.03664
0.6000    -0.04563
0.5000    -0.05294
0.4000    -0.05803
0.3000    -0.06002
0.2500    -0.05941
0.2000    -0.05737
0.1500    -0.05345
0.1000    -0.04683
0.0750    -0.04200
0.0500    -0.03555
0.0250    -0.02615
0.0125    -0.01894
0.0000     0.00000
0.0125     0.01894
0.0250     0.02615
0.0500     0.03555
0.0750     0.04200
0.1000     0.04683
0.1500     0.05345
0.2000     0.05737
0.2500     0.05941
0.3000     0.06002
0.4000     0.05803
0.5000     0.05294
0.6000     0.04563
0.7000     0.03664
0.7010     0.03564
0.8000     0.02556
0.9000     0.01415
0.9500     0.00790
1.0000     0.00126
*EOF
```

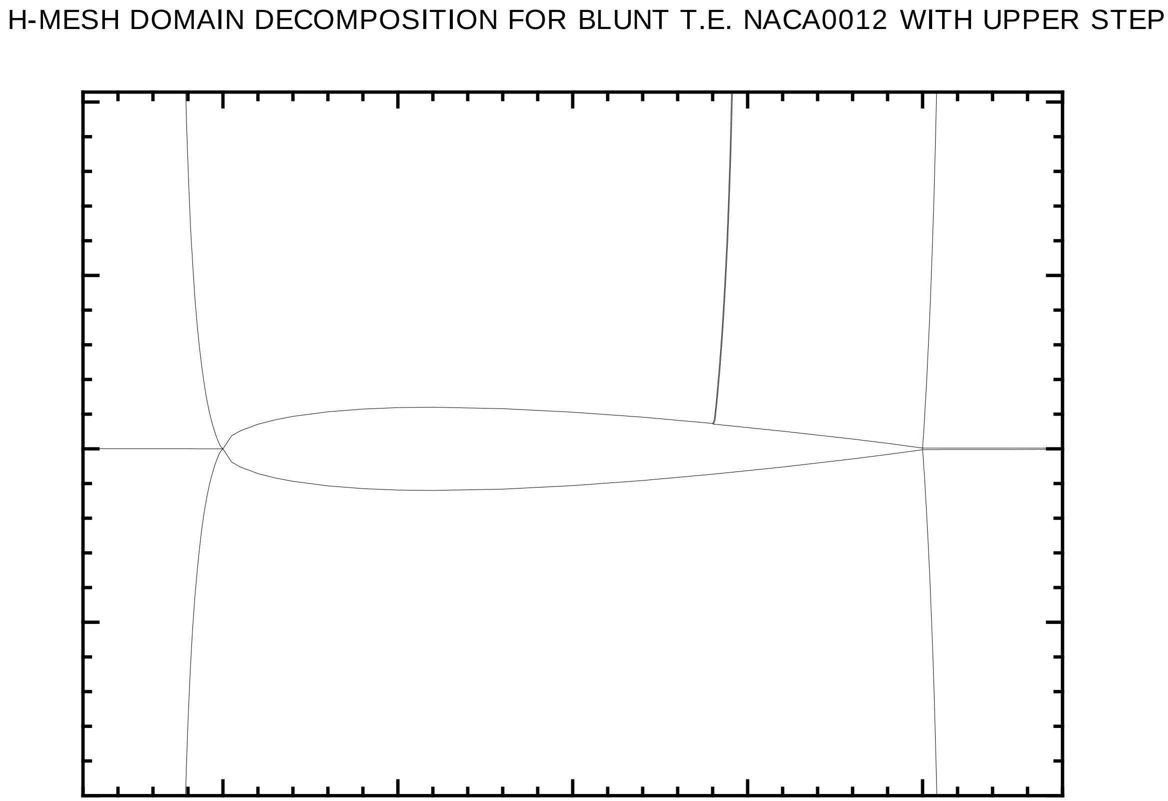

Figure 6.24: Domain decomposition for Test Case 10

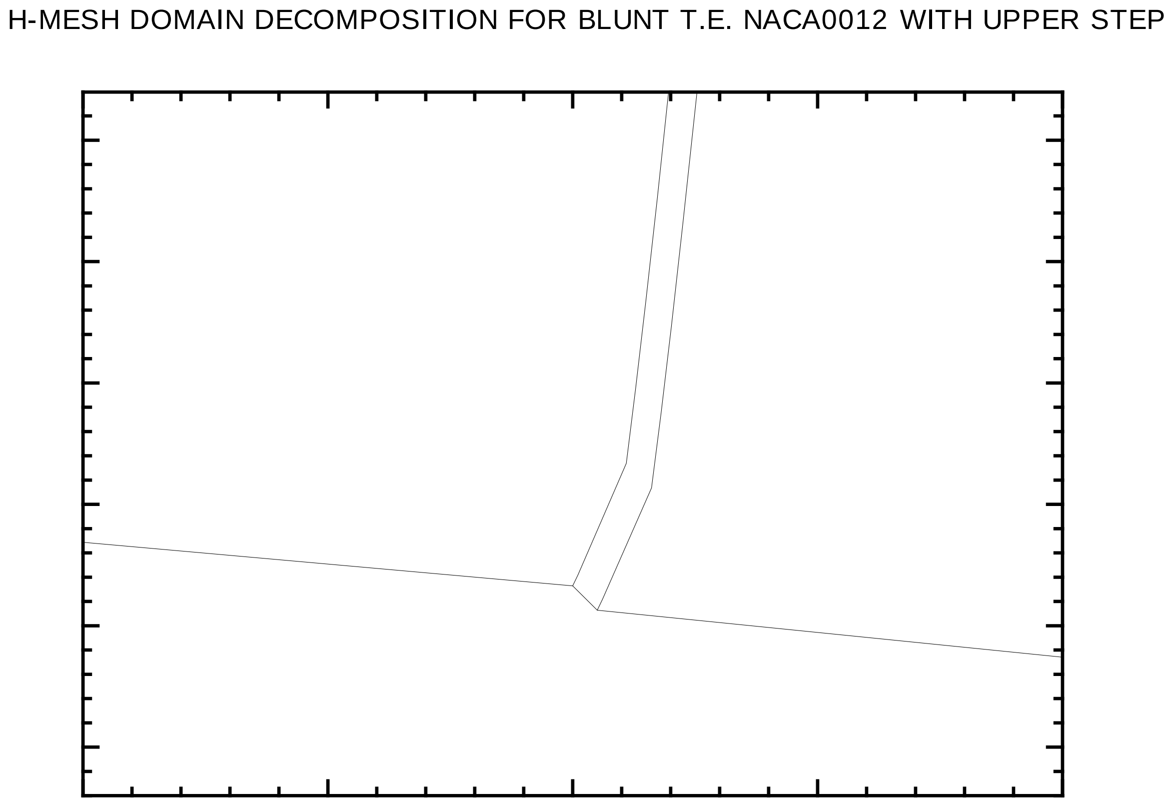

Figure 6.25: Domain decomposition for Test Case 10 - detail of step