



ESDU 96029

Issued October 1996
With Amendments A and C
June 2010

Transonic data memorandum VGK method for two-dimensional aerofoil sections Part 2: User manual

Associated software: VGK

Endorsed by
The Royal Aeronautical Society

AVA Documents
Document Sharing Hub

ESDU DATA ITEMS

Data Items provide validated information in engineering design and analysis for use by, or under the supervision of, professionally qualified engineers. The data are founded on an evaluation of all the relevant information, both published and unpublished, and are invariably supported by original work of ESDU staff engineers or consultants. The whole process is subject to independent review for which crucial support is provided by industrial companies, government research laboratories, universities and others from around the world through the participation of some of their leading experts on ESDU Technical Committees. This process ensures that the results of much valuable work (theoretical, experimental and operational), which may not be widely available or in a readily usable form, can be communicated concisely and accurately to the engineering community.

We are constantly striving to develop new work and review data already issued. Any comments arising out of your use of our data, or any suggestions for new topics or information that might lead to improvements, will help us to provide a better service.

THE PREPARATION OF THIS DATA ITEM

The work on this particular Data Item was monitored and guided by the Transonic Aerodynamics Committee. This Committee first met in 1960 and now has the following membership:

Chairman	
Mr B. Probert	— British Aerospace Defence Ltd, Warton
Vice-Chairman	
Mr D.R. Stanniland	— Aircraft Research Association
Members	
Mr S.E. Allwright	— Avro International Aerospace, Woodford
Mr A. Dyke	— British Aerospace Airbus Ltd, Bristol
Mr M.C.P. Firmin	— Independent
Mr J.L. Fulker	— DERA Aerospace Division, Bedford
Mr G.R. Hargreaves	— Avro International Aerospace, Woodford
Mr M.D. Hodges	— Independent
Mr D.R. Holt	— Independent
Dr D.R. Philpott	— University of Hertfordshire, Hatfield
Mr G.R. Richards	— British Aerospace Defence Ltd, Farnborough
Mr P.W.C. Wong	— Aircraft Research Association.

The construction and subsequent development of the Data Item were undertaken by

Dr M.M. Freestone	— Senior Engineer.
-------------------	--------------------

The person with overall responsibility for the work in this subject area is Mr D.J. Mitchell, Head of Group.

VGK METHOD FOR TWO-DIMENSIONAL AEROFOIL SECTIONS PART 2: USER MANUAL

CONTENTS

	Page
1. INTRODUCTION	1
2. NOTATION	2
2.1 VGK File Name Conventions	5
2.1.1 VGK file name extensions and respective functions	5
2.2 Use of the Term 'Wake'	6
3. DESCRIPTION OF A VGK RUN	7
3.1 Definition and Types	7
3.1.1 Inviscid runs	7
3.1.2 Viscous runs	8
3.2 Unsuccessful Runs	8
3.3 Treatment of the Boundary-Layer	9
4. RUNNING VGK	10
4.1 Purpose and Use of VGKCON	10
4.1.1 Examples of VGKCON use	11
4.2 System Requirements and Computing Times	18
5. AEROFOIL COORDINATES DATA FILE	19
6. EXAMPLES OF VGK RUNS	21
6.1 New Inviscid Run (INVDEMO1)	21
6.2 New Viscous Run (VISDEMO1)	23
6.3 Continuation Viscous Run (VISDEMO2)	26
7. RESOLVING PROBLEMS ENCOUNTERED WITH VGK RUNS	30
7.1 VGK Fails to Complete Specified Number of Iterations	30
7.2 VGK Completes Specified Number of Iterations but the Flow Results Seem Suspect	31
8. RUNNING THE PROGRAMS VGKCON AND VGK IN A COMMAND WINDOW	32
8.1 VGKCON	32
8.2 VGK	32
9. PRODUCING GRAPHICAL OUTPUT	33
10. REFERENCES	33

FIGURES 1-3	34-36
APPENDIX A LIBRARY OF AEROFOIL COORDINATES DATA FILES	37
A1. FILES	37

VGK METHOD FOR TWO-DIMENSIONAL AEROFOIL SECTIONS

PART 2: USER MANUAL

1. INTRODUCTION

VGK is a CFD (computational fluid dynamics) method coded in Fortran for predicting the aerodynamic characteristics of a single-element aerofoil in a subsonic freestream, including the effects of viscosity (boundary-layers and wake) and shock waves. VGK uses an iterative approach to solve coupled finite-difference equations for the inviscid flow region (assumed to be potential) and the viscous flow region (represented by integral equations). The aerofoil boundary-layers must be attached for VGK to produce results that correspond to the real flow, but the code has been calibrated against experimental data to provide a method of estimating a separation boundary* – see Ref. 2 and Part 3. VGK was developed over a period of years at RAE/DERA (now QinetiQ, Farnborough) and is made available by IHS ESDU under the terms of an agreement with QinetiQ.

The disk containing the files relating to VGK also contains a 'library' of files of aerofoil coordinates – see Appendix A for the list of aerofoils provided.

The Data Items dealing with VGK, of which this is the second, are:

- Part 1: Principles and results,
- Part 2: User manual,
- Part 3: Estimation of a separation boundary in transonic flow,
- Part 4: Use of VGK to calculate excrescence drag at subsonic speeds,
- Part 5: Design to a specified upper-surface pressure distribution,
- Part 6: Aerofoil with simple hinged flaps,
- Part 7: VGK for Windows TM.

This Item describes the use of the variant of the VGK program suite that can be run in Microsoft Windows from a Command Window. Sufficient information is given in this Item to obtain results from VGK, but Part 1 should be referred to for the principles and performance of the method, so that the results obtained may be properly assessed. Typical computing time for a single run of VGK using a 80486 DX processor is 90 seconds and using a Pentium 90 processor is 30 seconds.

The file name convention adopted for VGK runs is given in Section 2.1. A description of the various types of VGK run is given in Section 3. The functions of the files and the way they are used and produced by the programs VGKCON (the VGK 'control' program) and VGK are presented in Section 4.

The specification of the file providing the coordinates of the aerofoil of interest is given in Section 5. Results of example VGK runs are given in Section 6. Section 7 contains a description of some of the problems that may be encountered with VGK runs and how they might be overcome.

Detailed commands for using the programs are given in Section 8. The production of graphical output is dealt with briefly in Section 9.

* VGK has been subsequently extended (as BVGK) to deal additionally with the effects of low Reynolds numbers and with boundary-layers with (small) regions of separation, see Part 1, Section 6.

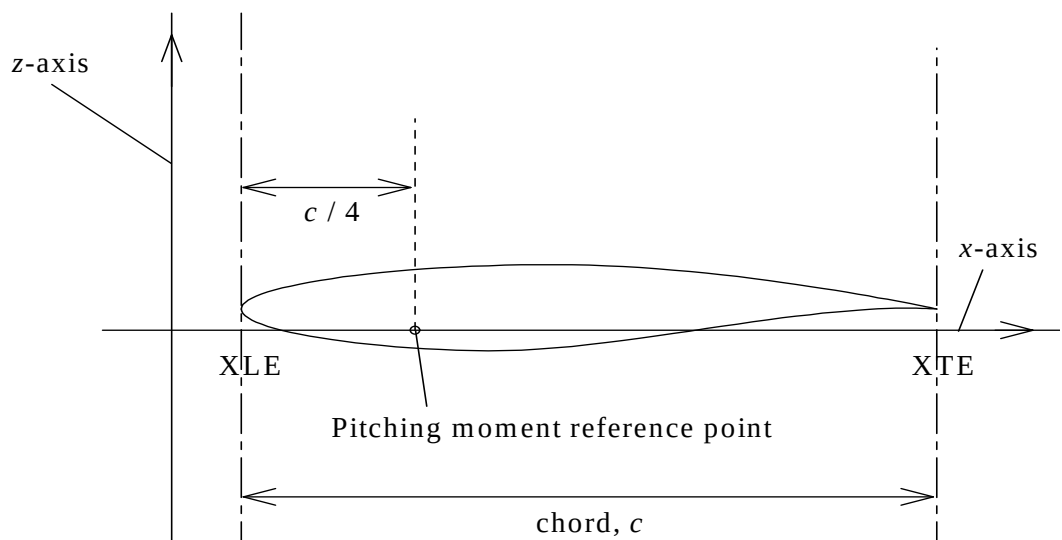
2. NOTATION

Algebraic notation is printed in italics or has Greek symbols. Notation used in VGK files is given in Roman type. Notation sequencing is based on the algebraic terms (except where none exists); some VGK notation therefore appears out of alphabetical order. All linear quantities made non-dimensional are based on aerofoil chord, c .

C_D	CD	overall (<i>i.e.</i> total) aerofoil drag coefficient (see also C_{DP} , CDP)
C_{DF}	CDF	contribution to C_D determined from integration of surface shear stresses
C_{DP}	CDP	contribution to C_D determined from integration of surface pressures (denoted by C_D (CD) for inviscid flow)
C_{DV}	CDV	viscous drag coefficient, determined from momentum thickness of wake far downstream
C_{DW}	CDW	wave drag coefficient
C_{D1}	CD1	wave drag coefficient determined from 'first-order method' (see Part 1, Section 3.4), context denotes whether for an individual shock or for all shocks
C_{D2}	CD2	wave drag coefficient determined from 'improved method' (see Part 1, Section 3.4), context denotes whether for an individual shock or for all shocks
C_L	CL	aerofoil lift coefficient
C_{Lpot}	CLPOT	value of C_L determined from jump in potential at trailing-edge
C_{Lspec}		value of C_L specified to VGK (only possible for viscous runs)
C_M	CM	pitching moment coefficient $\left(= M_{1/4} / \left(\frac{\gamma}{2} p_{\infty} M_{\infty}^2 c^2 \right) \right)$
C_p	CP	pressure coefficient $\left(= (p - p_{\infty}) / \left(\frac{\gamma}{2} p_{\infty} M_{\infty}^2 \right) \right)$
C_p^*		pressure coefficient for local Mach number of unity
c	C	aerofoil chord (see Sketch 2.1)
c_f	CF	local skin-friction coefficient
$d\zeta/d\xi$	DZ/DX	slope of aerofoil surface expressed in non-dimensional coordinates
$d^2\zeta/d^2\xi$	D2Z/DX2	second derivative of aerofoil surface expressed in non-dimensional coordinates
$\bar{H}$	HT	transformed boundary-layer velocity-profile shape parameter
	ISYM	integer denoting symmetrical aerofoil when equal to 1, and non-symmetrical aerofoil when equal to 0

	ITYP	integer denoting type of step in VGK calculation, when read in conjunction with NS
	IVP	control integer used by VGK, 0 for inviscid run, 4 for viscous run
j	NSG	number of radial lines in computing plane (see Part 1, Sketch 3.1)
k	K	number of circumferential grid lines in computing plane (see Part 1, Sketch 3.1)
k_c	NVISC	number of inviscid-flow iterations between each occasion viscous-flow boundary conditions are updated (in coarse mesh)
k_f	NVISC	number of inviscid-flow iterations between each occasion viscous flow boundary conditions are updated (in fine mesh)
M	EML	local Mach number (at edge of boundary-layer) based on isentropic flow relations
	GVISC	ω_c or ω_f in output files (context indicates which parameter is denoted)
M_{Lsh}		value of M just upstream of shock wave
$M_{1/4}$		pitching moment about quarter-chord reference point (see Sketch 2.1)
M_∞	EM	freestream Mach number
m, n		integers denoting point in grid (see Part 1, Sketch 3.1)
	NL	number of (x, z) coordinate pairs specifying aerofoil lower surface, see Section 5
	NS	integer denoting either NSC or NSF (context indicates), or when equal to -1 or when ITYP = -1 denotes type of step in VGK calculation
	NSC	number of iterations of inviscid-flow element using coarse mesh
	NSF	number of iterations of inviscid-flow element using fine mesh
	NU	number of (x, z) coordinate pairs specifying aerofoil upper surface, see Section 5
	NVISC	k_c or k_f in output files (context indicates which parameter is denoted)
p	P	local static pressure
p_0	P0	freestream stagnation pressure
p_∞		freestream static pressure
R	R	Reynolds number based on chord
	RES	maximum residual in inviscid flow element
	RTHETA	local Reynolds number based on momentum thickness and local conditions at edge of boundary-layer
	S	parameter related to conformal mapping calculation
	VRES	residual corresponding to trailing-edge location in viscous flow element

	XLE	value of x coordinate at leading-edge
	XM	relaxation factor applied to supersonic flow regions
	XS	relaxation factor applied to subsonic flow region
	XTE	value of x coordinate at trailing-edge
x		‘chordwise’ coordinate in physical plane (see Sketch 2.1)
z		‘normal’ coordinate in physical plane (see Sketch 2.1)
α	ALP	aerofoil incidence, <i>i.e.</i> angle between x -axis and freestream direction, positive nose-up
$\Delta\theta_L$	DD22	increment in non-dimensional momentum thickness at transition on lower surface
$\Delta\theta_U$	DD21	increment in non-dimensional momentum thickness at transition on upper surface
δ	DELTA	non-dimensional local boundary-layer thickness
δ^*	DELSTAR	non-dimensional local boundary-layer displacement thickness
ε	EP	artificial-viscosity parameter
γ		ratio of specific heat capacities (taken as 1.4)
λ	QCP	partially-conservative parameter
θ	THETA	non-dimensional local boundary-layer momentum thickness
θ_c	T	angular coordinate in computing plane (see Part 1, Sketch 3.1)
ω_c	GVISCC	relaxation factor associated with viscous effects using coarse mesh
ω_f	GVISCF	relaxation factor associated with viscous effects using fine mesh
ζ	X	non-dimensional chordwise coordinate, $\xi = 0$ at leading-edge and $\xi = 1$ at trailing-edge
ζ_{sh}	X(SHOCK)	value of ξ corresponding to shock location
ξ_{TL}	XTL	value of ξ corresponding to boundary-layer transition location on lower surface
ξ_{TU}	XTU	value of ξ corresponding to boundary-layer transition location on upper surface
ζ	Z	non-dimensional normal coordinate in physical plane
Subscripts		
	(L)	relates to lower-surface portion of wake (see Part 1, Section 3.1)
	(U)	relates to upper-surface portion of wake (see Part 1, Section 3.1)



Sketch 2.1 Aerofoil axes and notation

2.1 VGK File Name Conventions

The file names used with VGK consist of a part provided by the user, comprised of 1 to 8 alphanumeric characters (no spaces are allowed), followed by a . (period) and a three letter extension denoting the type (or function) of the file. The extensions used are listed in Section 2.1.1 with their respective functions. In this Item "ABCD1234" is sometimes used to denote the user-provided part of the file name. It is recommended that the user chooses file names that serve as a reminder of the work to which they refer and, more importantly, provide a unique identification of the particular VGK run to which they refer (see Section 3). It is also recommended that any letters used in file names be upper case.

2.1.1 VGK file name extensions and respective functions

File name extension	Function of file and comments
NAM	An input file for VGK, produced by the VGK 'control program', VGKCON. It contains mainly a list of file names. For a 'new' run (see Section 3) it contains the name of the aerofoil geometry file to be used in the VGK run.
VIN	An input file for VGK, produced by VGKCON. It contains, in cryptic form, the parameters that 'control' the VGK run. Even though some of these parameters may be easily identified, it is recommended that all .VIN files are produced by running VGKCON, and not by editing other .VIN files.
SIR	An input file for VGK, produced by VGKCON. It contains one file name.

<i>File name extension</i>	<i>Function of file and comments</i>
IEH	An output file of VGK. It contains the ‘inviscid-element history’, <i>i.e.</i> for each iteration of the inviscid flow element, it gives the values of RES and CLPOT. It is used to judge whether or not a VGK run is sufficiently converged.
VEH	An output file of VGK. For any viscous run it contains the ‘viscous-element history’, <i>i.e.</i> for each iteration of the viscous flow element it gives the value of VRES. Together with the ABCD1234.IEH file it is used to judge whether or not a VGK run is sufficiently converged.
FUL	The main or ‘full’ output file of VGK, providing data for surface pressures, shock locations, boundary-layer and wake parameters, and overall force and moment coefficients.
BRF	The ‘brief’ output file of VGK, giving surface pressures and overall force and moment coefficients.
GRD	An output file of VGK, provided optionally. It contains grid node coordinates in the physical plane in a form suitable for most plotting programs.
UNF	The ‘dump’ output file of VGK, provided optionally. It contains the aerofoil title, grid data, and data defining the inviscid flow element parameters as determined at the end of the final iteration of the VGK run. It is written in binary or ‘unformatted’ form, thus facilitating accurate retrieval of the data by a subsequent ‘continuation’ – see Section 3.
SEP	An output file of VGK, giving data relating to the two separation criteria referred to in Part 3. This file is only utilised when a separation boundary is estimated by the method of Reference 2, in the way described in Part 3.

2.2 Use of the Term ‘Wake’

In places in this Item, and in Part 2, there is reference to the aerofoil wake, and to the variation of certain parameters along that wake. In real flows, the wake comprises the relatively thin region of the flow downstream of the trailing edge that is directly affected by viscosity, and is formed by the merging of the upper- and lower-surface boundary layers. In VGK calculations of viscous flows, however, the viscous wake parameters, such as momentum thickness, are evaluated using the approximation that streamwise variations of Mach number and pressure in the wake are the same as corresponding variations along the *grid line(s)* that run downstream from the trailing edge. The location of the viscous wake is not actually calculated. It is in the sense that the boundary layer and other data presented in the .FUL files (see Part 2) are to be interpreted.

For inviscid flows, the .FUL files still purport to present data for such parameters as Mach number and pressure in the wake, but in such cases the wake has no physical interpretation. The files then simply give variations of the parameters along the grid lines(s) that run downstream from the trailing edge.

3. DESCRIPTION OF A VGK RUN

3.1 Definition and Types

A VGK run is defined as a calculation using the VGK code for a specified aerofoil section under specified aerodynamic conditions, and with a particular selection of the parameters controlling the iterative process of calculation. The computational grids utilised in a VGK run are either calculated from the input aerofoil geometry, in which case the term *new run* is applied, or are retrieved from the 'dump'(.UNF) file produced by an earlier VGK run for the aerofoil of interest. In this case the term *continuation run* is used. The dump file contains not only the grid data, but also data defining the inviscid flow element as determined after the final iteration step of the earlier run. The continuation run takes these 'final step' data as initial values for its iteration process.

Although intended primarily for viscous flow calculations, VGK can be used to calculate an inviscid flow. Thus there are four types of VGK run: *new inviscid*, *continuation inviscid*, *new viscous* and *continuation viscous*. Note that the dump file produced by **any** of these four types is suitable for a subsequent continuation run of **either** viscous **or** inviscid type.

3.1.1 Inviscid runs

A new VGK run commences with a grid calculation. This is done using a conformal mapping function that maps the region exterior to the aerofoil on to the interior of a circle of unit radius. The computational grids correspond to a series of concentric circles and radial lines in this circle plane. Two grids are utilised, termed the fine grid and the coarse grid. The fine grid recommended corresponds to 160 radial lines, and 30 concentric circles, *i.e.* 160 grid lines emanating radially from the aerofoil perimeter, and 30 grid lines circumferentially around from the aerofoil – see Figure 3 and Part 1, Section 2. (There are 31 grid lines if the infinity line, corresponding to the centre of the circle, is counted.) Alternative choices may be made (see Section 6.1). The coarse grid is formed by removing alternate grid lines of the fine grid. The finite difference equations for the velocity potential are solved at the grid nodes using an iterative relaxation process, starting from an initial approximation (see Part 1). For a successful run, the iterative process produces a converged potential field satisfying both the far-field boundary conditions of specified freestream Mach number and direction, and the surface boundary conditions that the flow is everywhere tangential to the aerofoil surface and that it leaves the trailing edge smoothly.

Each of the iterations consists of a sweep of a relaxation operator through the grid, the purpose of which is to reduce the errors (or residuals) in the approximate solution of the velocity potential field. The relaxation scheme applied depends on two parameters: XS, which is the relaxation factor used where the local flow is subsonic, and XM, which applies to supersonic flow regions. The default value for XS is 1.9, and for XM it is 1. Although generally suitable, these values may need to be decreased in some cases, see Section 6.1. A VGK run generally consists of a number (usually 100) of coarse-grid iterations, followed by a larger number (usually 200) of fine-grid iterations. The use of two grids in this way usually greatly increases the efficiency of convergence.

As the iterations proceed, the value of the largest residual, RES, associated with the velocity potential field, and the current value of C_{Lpot}^* are written to the file ABCD1234.IEH (see Section 2.1), so that the convergence (or divergence) of the iteration process can be examined once the run has finished. Following the iterations, various surface and overall aerodynamic parameters are calculated and then written to results files. As an option, the mapping (*i.e.* grid) data and final potential field are written in binary form[†] to a separate output file. The run is then terminated. Further VGK runs for the same aerofoil (for other values

* Small differences between C_{Lpot} and C_L (found from integration of the chordwise pressure distribution) occur as a result of shock waves and boundary-layer effects. C_{Lpot} always gives at least a fair approximation to C_L , and less computation is required to determine it.

† Adopting binary or unformatted form has advantages for files to be read in for subsequent runs.

of M_∞ or α , for example) can either be carried out in exactly the same way (*i.e.* as new runs), or can be carried out as continuation runs. The advantages of carrying out continuation runs are that fewer iterations may be needed to achieve convergence, and that some combinations of M_∞ and α *can be calculated as continuation runs but not as new runs*.

3.1.2 Viscous runs

In the case of a viscous run, in addition to the data required for an inviscid flow, it is necessary to input values for the Reynolds number, the locations of laminar/turbulent transition on the upper and lower surfaces, and any increments in boundary-layer momentum thickness at these locations, see Section 4.3 and Part 1, Section 4. Since, for a viscous run, VGK has the facility for calculating the aerofoil flow having a specified value of C_L , it is necessary also to input this C_L value (denoted by C_{Lspec}) for those runs where the facility is utilised. It is also necessary to specify values for the parameters, ω_c , k_c , ω_f , k_f which govern the way the viscous effects are introduced in the coarse and fine grids. (The default values for these parameters are: $\omega_c = 0.15$, $k_c = 5$, $\omega_f = 0.075$, $k_f = 5$. See Part 1, Section 3.1 for the effects of utilising other values.) A VGK run to calculate a viscous flow utilises procedures for progressively modifying the surface boundary conditions and conditions corresponding to the aerofoil wake in the (inviscid) potential field so as to account for the effects of the boundary-layer and viscous wake. Calculations of the viscous flow element and the inviscid flow element interact, *i.e.* the boundary-layer and wake calculations depend on the current calculation of the inviscid-flow parameters, and the inviscid flow iterations depend on the current calculation of the boundary-layer and wake. Following each sequence of k_c or k_f successive potential flow iterations, a viscous-flow step is calculated. In addition to writing values of residuals and C_{Lpot} to the file ABCD1234.IEH, the parameter VRES, a residual associated with the viscous-flow element, is written to the file ABCD1234.VEH after each calculation of the boundary-layer and wake. VRES should progressively diminish as the iterations proceed. The output files written for a viscous run include data for the boundary-layer and wake, as well as overall and surface aerodynamic parameters. Again the mapping and final potential field are written (optionally) to an output file for possible use as initial conditions of subsequent runs.

3.2 Unsuccessful Runs

A VGK run to calculate either a viscous or an inviscid flow may fail to produce an adequately converged result. Such unsuccessful runs may occur in a number of ways as considered below.

- (a) A run-time error may occur due to a function argument being outside the legal domain – for example, the argument of the SQRT function may become negative. With such a failure the output files for the run are absent or incomplete. This type of failure occurs when the input for the VGK run produces a rapidly diverging iteration sequence, or when even the first iteration in the sequence fails to be computed. Attempts to compute some flows (for example a flow with an unrealistically high value of C_L) may be expected to be always unsuccessful, but it is not always easy to judge whether or not a particular unsuccessful run would have succeeded if a different iteration sequence, defined by different values of the relaxation factors, had been adopted. Guidance on the choice of relaxation factors is given in Part 1, Section 3.3.
- (b) A VGK run will stop before the specified number of iterations has been completed if it is classed as having diverged according to the criteria embodied in the code. This is likely to happen when attempting to calculate flows with large local gradients of velocity. When it happens, the output file ABCD1234.FUL contains the message that the iteration procedure has diverged. Changes to the relaxation factors defining the iteration sequence may result in a successful run, see Part 1, Section 3.

- (c) A VGK run may produce results files that reveal that the degree of convergence achieved is unsatisfactory. A further run (or runs) utilising the dumped potential from the previous run (and using unchanged values for the relaxation factors) may result in satisfactory convergence, although it might be necessary to change the relaxation factors to achieve this. However, it may turn out that no successful VGK run for the particular combination of aerofoil and aerodynamic parameters is possible, whatever values are chosen for the relaxation factors.

3.3 Treatment of the Boundary-Layer

For a viscous run, VGK requires the non-dimensional coordinates of the boundary-layer transition points on the upper and lower surfaces, respectively XTU and XTL, to be specified. These are the points where the boundary-layer changes relatively suddenly from the laminar to the turbulent state. For information on specifying transition locations for VGK calculations, relevant to flight or wind-tunnel tests, see Part 1, Section 4. For over-fixed transition in wind-tunnel tests, the momentum thickness will undergo an increment at the transition location. In VGK, increments in the non-dimensional momentum thickness may be specified at the upper and lower surface transition locations, so as to model over-fixing – see Part 1, Section 4 for details.

During a VGK run, the calculated flow field at one or more of the iteration steps may include a relatively large adverse pressure-gradient region over part of the surface such that no computed attached boundary-layer could correspond to it. In VGK, where the calculated value of the boundary-layer shape-factor H indicates that the local skin friction coefficient, c_f , is very close to zero at a grid point on the aerofoil surface (where the boundary-layer is locally turbulent), then c_f is artificially raised (to 2×10^{-6}), so permitting the boundary-layer calculation to be continued downstream to the trailing edge. If the boundary-layer is laminar at a point where c_f becomes critically small, the code imposes transition at that point. This again has the effect of allowing the boundary-layer calculation to be continued downstream. (The imposition of transition by VGK in this way may be thought of as a simple model for the short-bubble laminar separation/turbulent re-attachment that often occurs in practice, see Part 1, Section 3.1.) Using this procedure enables some iteration processes to converge that otherwise could not be calculated. A disadvantage is that it is necessary to examine the results file ABCD1234.FUL to see if the calculated aerofoil flow has any locations where c_f has been raised artificially. If it has, but there are just one or two surface grid-point locations where c_f has been raised, then it is unlikely that the calculation would be significantly misleading; however, the presence of more than a small number of such locations must be taken as an indication that the calculation is unreliable. Examples are presented in Part 1, Section 6.