CFDSC Bulletin

CFD Society of Canada | Société Canadienne de CFD

Nov. 2005 | Issue No. 16

In this Issue

2

President's Message

I have known many of you since my involvement in the CFD Society for over a decade. Most recently I was elected as the president of the CFD society of Canada in July 2005…

3

CFDSC | Objectives

The Computational Fluid Dynamic Society of Canada (CFDSC) promotes the application of computational fluid dynamics (CFD) to provide a better understanding of fluid dynamic…

4

Brief Summary of CFD 2005

The 13th Computational Fluid Dynamics Conference of CFDSC was held in Fairmount Newfoundland Hotel, St. John's, Newfoundland and Labrador during July 31 and August 2, 2005…

4

Lifetime Achievement Award

The inaugural Lifetime Achievement Award of the Computational Fluid Dynamics Society of Canada was presented to Dr. G.D. Raithby at CFD2005 in…

5

CFD 2006 Conference | Queen's University

July 16-18, 2006 | The Computational Fluid Dynamics Society of Canada is pleased to announce that its 14th annual conference will be held on the beautiful campus of **Queen's University**…

6

Affiliation with International Journal of Computational Fluid Dynamics

The CFD Society of Canada is pleased to announce that each year, following the annual conference, a selected number of…

6

CFDSC Graduate Scholarship!

The CFD Society of Canada, in accordance with its aims promoting research in computational fluid dynamics, would like to announce the creation of a yearly…

6

The Direct Design Solution of Inverse Problems

The discussion of inverse problems receives very little attention in our educational system, yet many important technical problems, and indeed many…

9

Large-Eddy Simulation of Low-Reynolds-number Flows past Airfoils

This paper presents an investigation of low-Reynolds-number flows past airfoils. In particular, results of experimental measurements and …

4 Dr. G.D. Raithby, receiving Lifetime Achievement Award from the President of the CFD Society of Canada; Dr. Mahmood Khalid.

PRESIDENT'S MESSAGE
by
Mahmood Khalid

I have known many of you since my involvement in the CFD Society for over a decade. Most recently I was elected as the president of the CFD Society of Canada in July 2005 for the second time in St. John's Canada. Currently, I am employed at the Institute for Aerospace Research of the National Research Council as the Head of the fixed Wing Aerodynamics having served as head of IAR's CFD Group and Upland Aerodynamics facilities for almost 7 years. Those of you who have been our members since 1990's, would recall that your Society communicated regularly with its membership through this Bulletin which was circulated to membership at least a couple of times a year. However, the practice has been dormant for a while, and the current board has decided to restore this mechanism as an established and recognized means of approaching our membership.

It is clear that our Society's conferences are becoming increasingly more global with participation from diverse disciplines in fluid sciences. As a measure of our appeal, during the annual conference in Ottawa in 2004 we had received a total of some 157 abstracts from nations as far a point on the globe as Malaysia and South Korea with as many as 116 papers being received after a due review process. The St. John's conference witnessed a similar level of participation from domestic and international participants.

In 2004, for example we hosted four good lectures by scientists of international standings with an across distance on line presentation from non other than Frank Harlow himself, the Guru of Turbulence Modeling. During the conference in St. John's in Newfoundland organized by Dr. Pengfei Liu we witnessed some very outstanding lectures by world renowned scientists and CFD practitioners. The next conference is due to take place next year in Queens University, Kingston, Ontario. The organizing committee led by Prof. Pollard is well on its way and by all indications it has the makings of memorable event.

I would also like to inform you that IJCFD in collaboration with the CFD Society has agreed to issue an annual edition focusing on the papers presented to the CFD Society's annual conference. The technical committee will select about 6 to 10 papers worthy of meeting the journal's standard each year, which will be submitted as special issue after a thorough peer review process.

As many of you witnessed in St. John's the Society has initiated a new recognition award for a scientist who has rendered exceptional services to the discipline as a researcher, as an educator, as an entrepreneur. A committee selected from the peers scrutinizes the contributions and achievements of the candidates in consultation with referees of international recognition and fame. This year in St. John's this award was given to Prof. George Raithby.

CFDSC Bulletin

CFD Society of Canada | Société Canadienne de CFD

Nov. 2005 | Issue No. 16

CFD Society of Canada

Objectives ● The Computational Fluid Dynamic Society of Canada (CFDSC) promotes the application of computational fluid dynamics (CFD) to provide a better understanding of fluid dynamic processes in all applicable areas and thereby supports the competitiveness of Canadian industry and advances knowledge in the field. These objectives are met by *(1)* establishing a network of CFD practitioners and developers from industry, government and universities, *(2)* identifying national needs and priorities for research in CFD, *(3)* promoting research in CFD and related areas, *(4)* enhancing industry capabilities in CFD by the education and training of professionals, *(5)* promoting the use and increasing the availability of computational tools for CFD, *(6)* promoting communication and exchange within CFD and related disciplines (e.g. environment, biology and chemistry), *(7)* organizing conferences and seminars, and *(8)* representing Canada at international forums.

Membership Annual membership fees of the CFDSC are $50 for regular members, $20 for student members, and $25 for members who are retired.

World Wide Web | www.cfdsc.ca

Address

CFD Society of Canada
P. O. Box 4871, Station E
Ottawa, Ontario, Canada KIS 5J1

Bulletin The Bulletin of the CFDSC is published about three times a year to provide Members with information on current CFD issues. The published articles express the views and opinions of their respective authors and do not necessarily represent those of the CFDSC. The Managing Editors are:

Mahmood Khalid | NRC-IAR
Mobina Haider | Ottawa. Canada

mobina@inbox.com

Société Canadienne de CFD

Objectifs ● La Société canadienne de CFD (SCCFD) fait la promotion des applications de <<computational fluid dynamics>> afin de fournir une meilleure compréhension de la dynamique des fluides et contribue ainsi au développement de la compétitivité industrielle canadienne et à l'avancement des connaissances dans le domaine. Ces objectifs sont atteints en *1)* établissant un réseau d'utilisateurs et de développeurs de CFD dans les industries, les gouvernements et les universités, *2)* identifiant les besoins nationaux et les priorités de recherches en CFD, *3)* faisant la promotion de la recherche en CFD et les domaines reliés, *4)* rehaussant les capacités en CFD des industries par la formation et l'éducation des professionnels, *5)* faisant la promotion de l'utilisation et en augmentant la disponibilité d'outils informatiques en CFD, *6)* faisant la promotion de la communication et des échanges en CFD avec les disciplines reliées (par example, en environnement, en biologie et en chimie), *7)* organisant des conférences et des séminaires, *8)* représentant le Canada dans les forums internationaux.

Cotisation des membres Les frais annuels d'adhésion sont de 50$ pour les membres réguliers, 20$ pour les membres étudiants, et 25$ pour les membres à la retraite.

World Wide Web | www.cfdsc.ca

Adresse

Société canadienne de CFD
B. P. 4871, Station 'E'
Ottawa, Ontario,Canada K1S 5J1

Bulletin Le bulletin de la SCCFD est publié environ trois fois par année afin de fournir aux membres de l'information sur les grands courants reliés à la CFD. Les articles publiés représentent les opinions des auteurs respectifs et ne représentent pas nécessairement celles de la SCCFD. Les rédacteurs exécutifs sont:

Mahmood Khalid | CNRC-IAR
Mobina Haider | Ottawa. Canada

Brief Summary of CFD 2005

The 13th Computational Fluid Dynamics Conference of CFDSC was held in Fairmount Newfoundland Hotel, St. John's, Newfoundland and Labrador during July 31 and August 2, 2005. There were about 80 participants and 70 papers presented. About 40 percent of the participants were international.

There were four invited speakers and 14 parallel sessions. Current president of CFDSC, Dr. Mahmood Khalid opened the conference and introduced Dr. Pengfei Liu, the Chair of Local Organization Committee. Dr. F. Mary Williams, the Director General at Institute of Ocean Technology made a welcome address.

Dr. George Raithby received the first Life Achievement Award of CFDSC during the conference. The student paper award winners were: Mr. Johan Larsson from University of Waterloo and Ms. Xinfeng Gao from the University of Toronto.

The post conference workshop also went well. A total of 12 people attended the workshop with a great interest.

For more details, see **http://www.cfd2005.org**

CFDSC
Executive Committee & Directors

President
Mahmood Khalid - (2004-2007, G, NRC)
Executive Officer
Marilyn Lightstone - (2005-2008, U, McMaster)
Treasurer
François Lesage - (2003-2006, G, DND)
Communications Officer
Jon Pharoah - (2003-2006, U, Queens)
Membership Officer
Suzhen Chen - (2005-2008, G, NRC)
Secretary and Student Director
Jai Sachdev (2003-2006, Toronto)
Pengfei Liu (2004-2007, G, NRC)
Eric Thornhill (2005-2008, G, DRDC)
Jacek Szymanski (2004-2007, I, Nuclear Safety Solutions)
Clinton Groth (2005-2008, U, Toronto)
Dave Whitehouse (2005-2008, I, Martec)

U : University | G : Government | I : Industry

Lifetime Achievement Award

The inaugural Lifetime Achievement Award of the Computational Fluid Dynamics Society of Canada was presented to Dr. G.D. Raithby at CFD2005 in St. John's, Newfoundland. This award was created to recognize individuals who have made outstanding time-enduring contributions to computational fluid dynamics in Canada.

Dr. Raithby obtained his Ph.D. from the University of Minnesota in 1967 and subsequently held a post-doctoral position at a University in Germany. He joined the Department of Mechanical Engineering at the University of Waterloo in the 1969. In addition to his outstanding academic research, Dr. Raithby was renowned for his exemplary teaching and was awarded a 'Distinguished Teacher Award' by the university. His teaching style ensured that students would get a deep and profound understanding of the subject being taught. As a researcher he pioneered and advanced the technology and understanding of CFD in several key areas. This resulted in the development of software of sufficient robustness, accuracy and efficiency to tackle problems of substantial engineering interest. In the mid-1980's Dr. Raithby founded the company Advanced Scientific Computing (ASC) Ltd. to allow for commercialization of the software, thus bringing CFD into Canadian industry. This business is currently owned by ANSYS Inc., but is still run

as a separate business entity with head office in Waterloo. Dr. Raithby retired from the University of Waterloo in 1996 and is currently a 'Distinguished Professor Emeritus'.
In the development of the Lifetime Achievement Award, a procedure for the rigorous review of award nominees was put in place by the Computational Fluid Dynamics Society of Canada. International leaders in CFD reviewed the nominations. Excerpts of the reviews of Dr. Raithby's nomination include

"Professor Raithby is widely admired for his integrity and innovation. Through his research publications, his education of students, his consulting (notably on environmental problems), and his creation of a CFD software company, he has made pioneering contributions to CFD in Canada and the world."

"His [Prof. Raithby's] many contributions include his insightful work on upstream weighting, major contributions in Finite Volume and Finite Element Methods, pressure solution in primitive variable formulations, multigrid solvers, and many more. The early work of Stubley, Raithby and Strong (1980) on what is now known as the Error Transport Equation approach to error estimation has proven seminal to recent work."

"The nominee [Prof. Raithby] is one of the few pioneers in CFD in various ways: He performed and directed important fundamental developments leading to successful CFD methods which are still in use today; he had a great impact as a teacher of the methods and by training qualified researchers and developers and has written many pivotal papers through which the pioneering work was disseminated world wide; he also pioneered the transfer of CFD technology to industry, mainly by developing one of the first successful commercial CFD codes and by founding the CFD company ASC in Canada ..."

The CFD Society of Canada is very proud to have Dr. Raithby as the first recipient of the Lifetime Achievement Award. Through his roles as teacher, researcher, and businessman, he has had a significant impact in the field of CFD that certainly very few individuals worldwide can claim. The Society is particularly pleased that Dr. Raithby's wife Cathy was present at the award ceremony in St. John's.

Dr. G.D. Raithby, receiving Lifetime Achievement Award from the President of the CFD Society of Canada; Dr. Mahmood Khalid.

CFD 2006
Queen's University at Kingston
Ontario, CANADA

July 16-18, 2006 | The Computational Fluid Dynamics Society of Canada is pleased to announce that its 14th annual conference will be held on the beautiful campus of **Queen's University**, which is located in the historic city of **Kingston, Ontario**, Canada. Queen's University is one of Canada's finest institutions of higher learning and was founded in 1841. Kingston is in the heart of the 1000 Islands tourist area, and has been a magnet for tourists from around the world. Kingston, with a population of about 125,000 has many fine restaurants and picturesque buildings that house many "old-world" style accommodations.
The Conference will feature 4 keynote lecturers,

a CFD challenge exercise and a short-course, which will be held immediately following the conference. The Keynote speakers are:

Professor Bill Dawes | **Cambridge University, England**

Professor Ned Djilali | **University of Victoria, Canada**

Professor W.G. (Fred) Habashi | **McGill University, Canada**

Professor Gordon Mallinson | **University of Auckland, New Zealand**

There will be 4 parallel sessions. All papers will be peer reviewed. For further information, please contact one of the organizers:

Professor Andrew Pollard
pollard@me.queensu.ca

Professor Darko Matovic
matovic@me.queensu.ca

Professor Jon Pharoah
pharaoh@me.queensu.ca

Department of Mechanical and Materials Engineering
Queen's University at Kingston
ON Canada K7L 3N6
www.cfd2006.org

Affiliation with International Journal of Computational Fluid Dynamics

The CFD Society of Canada is pleased to announce that each year, following the annual conference, a selected number of papers would be submitted for publication in the International Journal of Computational Fluid Dynamics (Editor: Fred Habashi). The Society encourages its members to submit their journal articles for publication in this Journal. The Journal enjoys a high reputation amongst other CFD publications with meticulous reviewing policies and a clear focus on current topics

CFDSC Graduate Scholarship!

The CFD Society of Canada, in accordance with its aims promoting research in computational fluid dynamics, would like to announce the creation of a yearly Graduate Scholarship in the amount of $1000. The graduate scholarship will be a yearly competition awarded to a student in a Master's or Ph.D. level graduate program conducting research work related to CFD at a Canadian University. Details and application forms for the scholarship can be found at the society's website: www.cfdsc.ca. The deadline for submissions is May 1st of each year.

The "Direct Design" Solution of Inverse Problems

A. Ashrafizadeh, G.D Raithby and G.D.Stubley
Dept. of Mechanical Engineering, University of Waterloo, Waterloo, ON, N2L 3G1
graith@mecheng1.uwaterloo.ca

Introduction

The discussion of inverse problems receives very little attention in our educational system, yet many important technical problems, and indeed many important problems in one's personal life, are inverse problems. This Note outlines the Direct Design Method for solving inverse problems involving fluid flow and heat transfer. The Method is simple to apply and, as far as we know, new. Being new, there are many issues that still need to be resolved. The material is a brief summary of the invited lecture by the second author (Raithby) at the CDF2005 conference in St. John's.

CFDSC Bulletin

Société canadienne de CFD Society of Canada

CFD Society of Canada | Société Canadienne de CFD

Nov. 2005 | Issue No. 16

What is an Inverse Problem?

C.W. Groetsch starts his wonderful book “Inverse Problems” [1] by observing “Inverse problems are hard to define. Yet every mathematician recognizes an inverse problem when she sees one”. Everyone learns to categorize by seeing examples. We start, then, by expanding on the example used by Groetsch in his opening paragraph.

The problem “what is the product of 2 and 6?” is a direct problem, while “what two integers when multiplied together yield 12?” is an inverse problem. One property of inverse problems is that they are generally more difficult than direct problems. But there are multiple answers to this inverse problem: Non-uniqueness is another feature of some inverse problems. Suppose we change the problem slightly to “what two integers multiply to give 13, ruling out the trivial solution 1 13=13?” There is clearly no solution - yet another possibility for inverse problems.

There are many inverse problems in fluid flow and heat transfer. For example, given the shape of an airfoil, the prediction of the surface pressure is a direct problem. The inverse problem is “given the surface pressure what is the airfoil shape?”. Another inverse problem is “given the pressure on a duct wall, what is the duct shape?”

Inverse problems are often called “design” problems and “direct” problems are referred to “analysis” problems. The terms “inverse” and “design” are used interchangeably, as are “direct” and “analysis”.

The “Direct Design” Method

A broad understanding of the Direct Design Method, and the basis for the name, will hopefully become clear through a simple example. Consider steady heat conduction in a uniform insulated rod extending from along the x axis. The end temperatures are specified. The inverse problem is to find the length of rod given the heat flux – using a finite volume method that is extendible to more complex problems. To solve this problem, the rod is subdivided into control volumes, with a node located in each volume, and with a node and half volume at each end. The unknowns are the location and temperature of each node i.

For internal nodes, the conservation of energy equation and a grid generation equation (e.g. the node i lies midway between its neighbours) provide the two required constraints. Unlike the analysis problem, where the node locations are fixed, the inverse problem is non-linear in x since the heat flux is proportional to the inverse of the distance between nodes. The boundary node location and the temperature at the node are known. For the boundary node at the other end, the temperature is known and the heat flux is also specified, so the energy balance for the control volume associated with this node is the constraint for its x location.

If there are N nodes in all, the grid generation constraints, the energy balances, and the boundary conditions provide 2N equations for the 2N values of and . Because the energy equation is non-linear in x, the Direct Design Method linearizes this equation is such a way that and both appear as dependent variables. A sequence of linear equation sets is solved that converges to the solution of the non-linear problem. The explicit, or “direct” appearance of the nodal locations in the linearized equations of the “design” problem is the origin of the name “Direct Design”.

Exactly the same principles applied in this extremely simple example apply to more complex problems. Our first experience with the method [2], before we realized we were solving an inverse problem, involved 2D problems governed by the Navier Stokes equations. This was followed by the solution of several 2D problems governed by Laplace’s equation [3,4,5]. 1D problems, like the one just described, have been invented to convey the basic idea. The following examples are 2D problems governed by Laplace’s equation.

Examples

Example 1 Flow in a converging nozzle: Consider the design of an elbow that accelerates an incompressible ideal flow through a 2:1 area ratio while turning it by 90^{o}. This is the classic Stanitz elbow problem [6]. Suppose the initial shape is guessed to be that in Fig. 1, top-left. The streamlines for this shape are shown, and Fig. 1 top- right gives the predicted distribution of wall velocity plotted as a function of dimensionless distance along each wall. For this (ideal) flow, the wall velocity is directly related to the surface pressure by Bernoulli's equation, so that the regions of decreasing velocity correspond to regions of adverse pressure gradient. These regions are prone to flow separation in a real viscous flow.

The wall velocities in Fig. 1 bottom-right would eliminate the regions of adverse pressure gradient and hence the propensity for flow separation. Specifying these wall velocities, the Direct Design Method calculates the duct shape in Fig. 1 bottom-left.

The main difference from the 1D example is that there are 3 unknowns per node in this problem: . In this case, two constraints are supplied by the grid generation algorithm and one from the governing equation. This method has been applied [3] to 2D ducts that are straight nozzles and diffusers, curved nozzles and diffusers and S-bends.

Example 2 2D heat conduction: Consider a cylinder, infinitely long in the z-direction, that is symmetric in y and has a heated strip of specified temperature along the x-axis over the interval. Heat flows from the outside surface of the cylinder, at constant temperature, to a fluid at uniform temperature, where the heat transfer coefficient h is constant (so the surface heat flux is also constant). The design problem [4] is to find the cylinder shape.

For the parabolic variation in shown in Fig. 2(a), with temperature at the ends of the heated strip, the calculated cylinder shape is shown in Fig. 2(b). The Direct Design Method obtained this solution in just a few iterations starting with a circular cylinder with radius, or 60 times the dimension of the final shape at.

Example 3 Direct design of an airfoil. The velocity along the surface of a Joukowski airfoil was specified and the Direct Design Method was applied [5] to see if the correct Joukowski shape was recovered. The calculation was started by guessing the symmetrical airfoil shown in Fig. 3. After a total of 13 iterations, the airfoil has changed position in the flow field and changed its shape, as shown in Fig. 3 by the n=N=2 plot. This shape is in excellent agreement with the correct Joukowski shape. The shape designated n=1 is an intermediate shape after 5 iterations.

The Future

In many design problems, the objective is to find a better surface shape. Sometimes optimization methods are used, but usually this is done by guessing a shape, analyzing the flow using a direct method, tweaking the shape and repeating the process until a satisfactory shape is found. It is often difficult to foresee how the shape should be changed. There is a more direct relation between surface pressure and the flow, so that designers would likely find design easier by tweaking the surface pressure, provided a good inverse problem solver is available. This is a big proviso. Much development and testing remain to be done, but it is hoped that the Direct Design Method may eventually provide a robust and efficient solver for 3D inverse problems governed by the Navier Stokes equations.

Acknowledgements

The basic research on the Direct Design Method has, until recently, been funded though the NSERC Discovery grant to the second author. This support is gratefully acknowledged.

References

[1] Groetsch, C.W., Inverse Problems, The Mathematical Society of America, 1999.
[2] Raithby, G.D., Xu, W.-X. and Stubley, G.D., "Prediction of Free Surface Flows with a Finite Volume Method", CFD Journal, Vol. 4, No. 1, pp. 353-371, 1995.
[3] Ashrafizadeh, A., Raithby, G.D., and Stubley, G.D., "Direct Design of Ducts", ASME J. Fluids Engineering, Vol. 125, pp158-165, 2003.

[4] Ashrafizadeh, A., Raithby, G.D., and Stubley, G.D., "Direct Design of Shape", J. Numerical Heat Transfer - Fundamentals, Vol. 41, pp501-520, 2002.

[5]Ashrafizadeh, A., Raithby, G.D., and Stubley, G.D., "Direct Design of Airfoil Shape with a Prescribed Surface Pressure", J. Numerical Heat Transfer - Fundamentals, Vol. 46, pp505-527, 2004.

[6] Stanitz, J.D., A Review of Certain Inverse Methods for the Design of Ducts with 2- and 3-Dimensional Potential Flow, Appl. Mech. Rev., Vol. 41, no. 6, pp217-238, 1988.

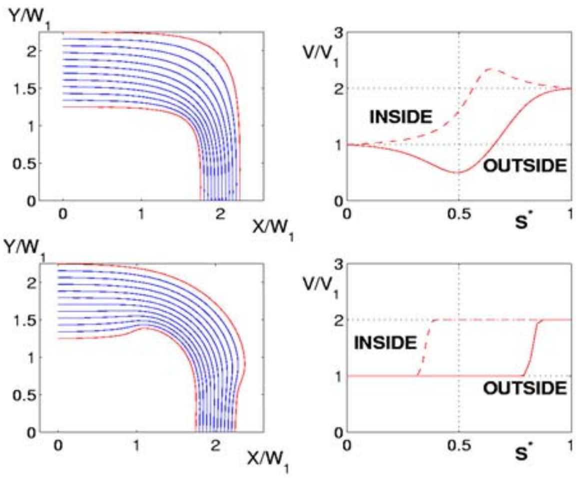

Fig. 1 Upper-left: Initial duct shape with streamlines; upper-right surface velocities for ideal flow; lower-right: prescribed wall velocities; lower-left: computed duct shape for prescribed wall velocities

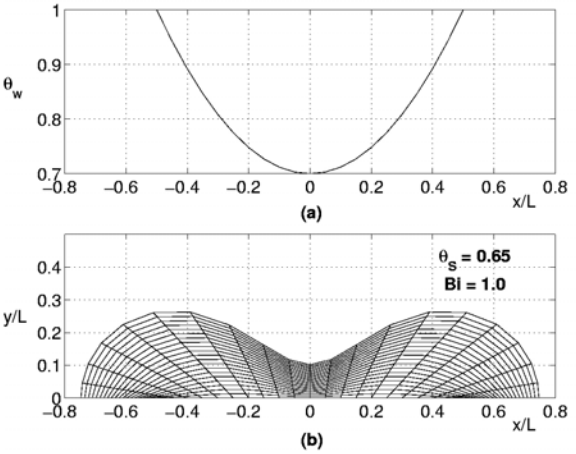

Fig. 2 Computed shape of cylinder for parabolic variation of strip temperature, T_w, at $y/L = 0$ and $-0.5 \le x/L \le 0.5$, for constant surface temperature, T_s, and constant heat transfer coefficient, *h*.

$$\theta_w = (T_w - T_\infty)/(T_L - T_\infty)$$

$$\theta_s = (T_s - T_\infty)/(T_L - T_\infty),\ Bi = hL/k\,.$$

Large-eddy Simulation of Low-Reynolds-number Flows past Airfoils

Weixing Yuan and Mahmood Khalid
Institute for Aerospace Research (IAR)
National Research Council (NRC) Canada
Ottawa, Ontario, K1A 0R6, Canada

This paper presents an investigation of low-Reynolds-number flows past airfoils. In particular, results of experimental measurements and numerical calculations based on LES simulations are discussed.

Nomenclature

CL = lift coefficient
CD = drag coefficient
C_f = skin friction coefficient
C_p = pressure coefficient
c = chord
U_∞ = free-stream velocity
u = velocity
x,y,z = Cartesian coordinates

Introduction

Low-Reynolds-number aerodynamic performance of small sized air vehicles is an area of increasing interest. This case exemplifies a flow phenomenon called laminar separation bubble. The principal flow behavior of a laminar separation bubble is sketched in Figure 1. In the first stage of the transition process, external distortions, like freestream turbulence, acoustic waves or surface roughness, generate small, harmonic waves within the laminar boundary layer upstream of the separation. In the second stage, unstable waves grow exponentially while traveling downstream. For the low disturbance environment investigated in this paper, the primary instability mechanism encountered in this stage is initially of the “Tollmien-Schlichting” (TS)-type. Depending on the distances of the separated shear layer from the wall, a smooth shift over to the Kelvin-Helmholtz (KH) instability can take place, which might become dominant in the rear part of the second stage. In the third stage of transition, where the distortions become so large that saturation occurs and secondary instabilities can grow on the distorted boundary layer flow. This behavior is fraught with nonlinear interactions. Finally, the number of spatial and temporal modes grows rapidly and the ordered laminar structures break down to turbulence.

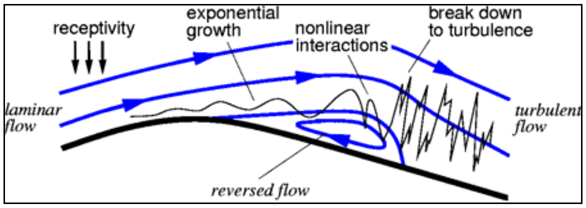

Figure 1. Instability and transition in a laminar separation bubble, with a sketch of time-averaged streamlines.

Description of the Simulations

Large-eddy simulations were performed at the NRC-IAR. As LES is a direct numerical simulation (DNS)-like method, it simulates large eddies and models the small ones using sub-grid-scale (SGS) models. Therefore, it is able to capture the transition process without pre-specifications of modelling individual stages. A major controversy for the computations of transitional flows is the artificial triggering of transition by a pointwise input of turbulence energy. Similar to recent successful LES and DNS investigations by other researchers, the current LES calculations have confirmed that the numerical forcing is not needed for this type airfoil flow. Transition occurred naturally on the airfoil suction side in the large-eddy simulation when the mesh was very fine. To resolve smaller eddies occurring in the transitioning and turbulent regions, very fine grids and very small timesteps are required.

Using the in-house code INSflow, investigations were performed for low-Reynolds-number flows past an SD7003 airfoil designed to explore low-speed viscous features. Large-eddy simulations at NRC-IAR were performed using the Smagorinsky SGS and the SMSM models. However, no significant difference was found between the results from these two SGS models. The turbulence intensity of the freestream was not considered in the large-eddy simulations. In the aforesaid simulations, the chord length of the airfoil was set to 202.6mm, which was the same as in the experiments at the Technical University of Braunschweig (TUBS). The fluid was water. The freestream Reynolds number based on the airfoil chord length was set to 60K. C-H grids were chosen for the calculations. The boundaries were located at least 25 chords away from the airfoil surface. The results illustrated here were obtained on a mesh with 737×65×17 grid points. Owing to large computational times and costs, only one case at 4° angle of attack was investigated. In the 3D LES, the span was assumed to be 6.4% of the chord and a periodic boundary condition was set for the spanwise direction, which allowed statistical averaging along the spanwise direction. The maximum value of y^+ in the computations was determined to be 0.4 at the wall. The timestep was set to be 10^{-4} seconds. The resulting CFL number was about 1-2 in the large-eddy simulations. Statistics were computed after the flow has reached a statistically stationary state. The time averaging for LES was made based on results of not less than 15000 timesteps, which corresponded to more than 2.2 units of flow through time (c/U_∞).

The size of the numerically predicted laminar separation bubbles is compared to the existing experimental data. As seen in the following table, the predicted locations of laminar-turbulent transition and turbulent reattachment were in reasonably good agreement with the experimentally measured data.

Table 1 Comparison among numerical results and experimental data for Re=60K and α=4°

Data set	Tu [%]	Separation [x_s/c]	Transition [x_t/c]	Reattachment [x_r/c]
Expt. (TUBS)	0.08	0.30	0.55	0.62
Expt. (AFRL)	~0.1	0.18	0.47	0.58
LES (IAR)	0	0.25	0.49	0.60

Figure 2 shows the mean pressure coefficient on the airfoil surface from the 3D LES simulation. At the airfoil leading edge, the strong adverse pressure gradient on the airfoil upper surface caused the flow separation of the boundary layer. The flattened region followed the adverse pressure gradient corresponded to the flow separation. The pressure plateau terminates when the transition of the shear layer caused a rapid increase in the surface pressure. After complete transition from laminar to turbulent flow, the large turbulent shear stress generated large momentum transport towards the airfoil surface resulting in reattachment.

Figure 3 shows the skin friction coefficient obtained from the mean flow on the upper surface of the airfoil. In the separation region between x/c=0.25 and 0.6, the skin friction coefficient was negative, indicating the laminar separation bubble. In accordance with the pressure changes shown in Fig. 2, the sudden decrease of c_f before reaching its negative peak at x/c=0.55 indicated the transition. The abrupt recovery of c_f from its negative peak to a positive value at x/c=0.6 corresponded to the reattachment of the separated flow.

The profiles of the mean streamwise velocity from 3D LES calculations and its root-mean-square (r.m.s.) fluctuation as a function of the dimensionless wall-normal coordinate at various streamwise locations are shown in Figure 4 and Figure 5. In Figure 5, the mean velocity profiles clearly show three different flow regimes: laminar attached flow at the leading edge, laminar boundary layer separation and turbulent reattachment. It can be seen in Figure 5 that flow unsteadiness already appeared at x/c=0.2. This indicated that the initial unsteadiness occurred before the laminar separation point of the mean flow. The fluctuation near the wall started to develop rapidly and violently at about x/c=0.5 and became dominant at x/c=0.6 showing strong turbulent activities in the near wall region.

It is well known that the necessary condition for a flow to be inviscidly unstable to small disturbances is that the velocity profile has an inflection point. This theorem was first formulated by Rayleigh. Later, Tollmien showed that the existence of an inflection point is also a sufficient condition for instability. The profiles in the separation bubble shown in Figure 4 had inflection points and hence were inviscidly unstable. The freestream shear layer in the separation bubble is likely to become inviscidly unstable via the Kevin-Helmholtz instability mechanism. Generally, the amplification is larger in the case of an inviscid instability than is in the case for viscous instabilities (Tollmien-Schlichting waves).

Figure 6 shows validation of the Reynolds shear stress against experimental data. The 3D LES prediction of the laminar separation bubble and the transition process compared reasonably to the two experiments. The predicted maximum values of $-\overline{u'v'}$ were reached above the reattachment point while its development was similar to the experimental observations, with a tendency to approach to the AFRL results. Both, the predicted LSB length and the transition point, lag comfortably in between the measured data from the two experiments. As mentioned earlier, the $\overline{u'v'}$ distribution was obtained from statistical averaging in the spanwise direction as well as in time. Therefore, the spanwise motion and fluctuations were taken into account. This provides an option to understand the 3D effects although it is still very time-consuming.

Figure 7 shows the instantaneous contours of spanwise vorticity near the mid-span location of the computation domain. The figure confirms that the instantaneous reattachment point was indeed highly variable in time. The evolution of vortex shedding from the separated shear layer as it rolled downstream can be clearly seen.

Figure 8 shows the three-dimensional isosurfaces of the instantaneous spanwise vorticity near the airfoil surface. The 2D structure of the separated shear layer, the distortion of the 2D structure, and the breakdown to 3D structure can be clearly seen in the figure. As no artificial forcing for the onset of transition was used in the simulations, the numerical noise replaced the action of freestream turbulence or surface roughness, which generated three-dimensional structures in practical application.

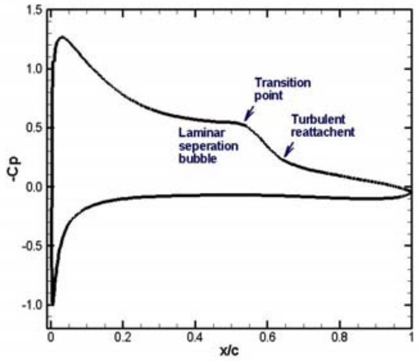

Figure 2. Mean pressure profile along the chord of the airfoil at Re=60K and α=4°.

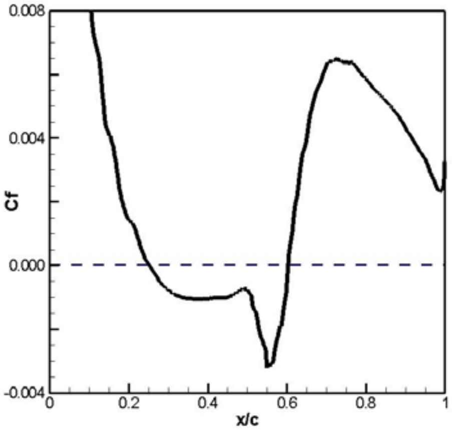

Figure 3. Mean skin-friction coefficient on the airfoil suction surface at Re=60K and α=4°.

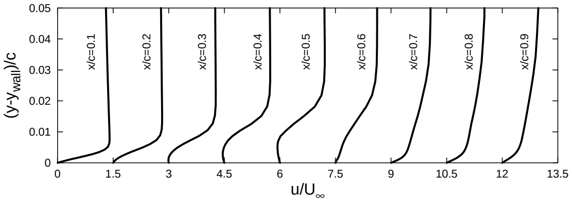

Figure 4. Profiles of mean streamwise velocity at several streamwise locations in function of normalized wall normal distance; individual profiles are separated by horizontal offset of 1.5 with the corresponding zero lines located at 0, 1.5, …, 12.

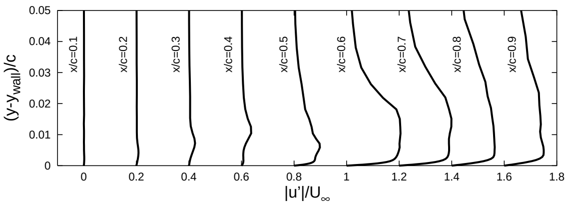

Figure 5. Profiles of the r.m.s. streamwise velocity fluctuation; individual profiles are separated by horizontal offset of 0.2 with the corresponding zero lines located at 0, 0.2, …, 1.6.

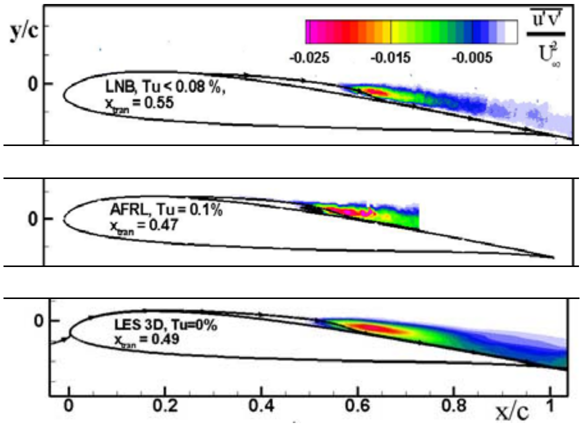

Figure 6. $\overline{u'v'}$ contour over the SD7003 airfoil at Re=60K and α=4°.

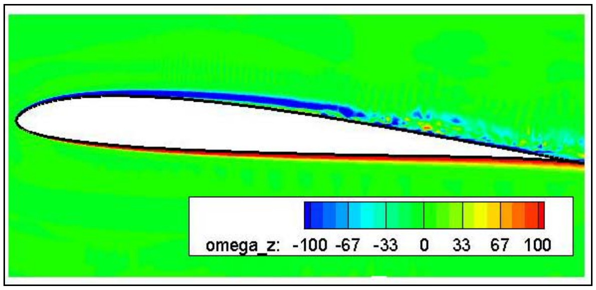

Figure 7. Instantaneous spanwise vorticity near the mid-span.

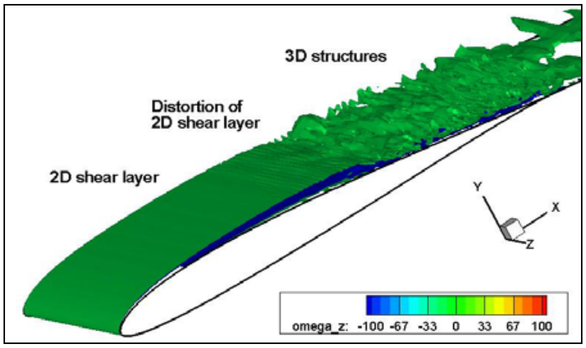

Figure 8. Three-dimensional isosurfaces of instantaneous spanwise vorticity.

Conclusions

The paper has addressed large eddy simulations of incompressible flows at low Reynolds numbers. By and large, large-eddy simulation was able to predict laminar separation bubbles including the corresponding transition process in good agreement with the observation in the experiments. The computed Reynolds shear stress, one of the critical parameters on which to base the accuracy of viscous flows, compared favorably with the measured data. Vortical structures (2D, 3D vortices) have been demonstrated at different stages of transition. It is believed that the transition process initiates with 2D instability of the large-scale separated shear layer in the laminar separation bubble. There is evidence of 3D flows which coupled with the separated shear layer constitutes distinct rolling up vortices leading to the eventual breakdown to turbulence.