Ain Shams Engineering Journal (2015) 6, 281–288

Ain Shams University

Ain Shams Engineering Journal

www.elsevier.com/locate/asej
www.sciencedirect.com

MECHANICAL ENGINEERING

Design optimization of three dimensional geometry of wind tunnel contraction

A.S. Abdelhamed, Y.El-S. Yassen *, M.M. ElSakka

Mech. Eng. Dept., Faculty of Eng., Port-Said Univ., Port-Said, Egypt

Received 22 November 2013; revised 16 August 2014; accepted 10 September 2014
Available online 19 October 2014

KEYWORDS

Wind tunnel contraction;
Turbulence model;
Goal-driven optimization;
Screening method;
Uniformity;
Analytical technique

Abstract The aim of the present study is to redesign three dimensional geometry of existing open circuit wind tunnel contraction. The present work achieves the recommended contraction ratio, maximum uniformity at the working section mid-plane, without separation, no Gortler vortices in the contraction, and minimizing the boundary layer thickness at entrance to the working section. Using CFD along with optimization tools can shorten the design optimization cycle time. Moreover CFD allows insight into the minute flow details which otherwise are not captured using flow bench tests. The design exploration algorithm is used to optimize the profile of the contraction in an automated manner. The optimization is based on using screening method to choose the best design set and verified by the CFD solver. The new contraction, compared to the old design contraction is confirmed using CFD. The new design is manufactured in full scale. The optimized contraction is investigated computationally and experimentally.

1. Introduction

Contraction is an important part of a wind tunnel. The main effects of a contraction are to reduce both mean and fluctuating velocity variations to a smaller fraction of the average velocity and further to increase the corresponding mean velocity. Generally to design of a subsonic and supersonic wind tunnel, the contraction portion should not have adverse pressure gradient in the stream-wise and further the effect of adverse pressure gradient at the exit of the contraction must be minimal. Whenever a converging duct segment is attached to constant-area segments, regions of adverse pressure gradient will occur along the wall, at its inlet and exit that may cause boundary layer separation. If separation occurs, it will degrade the flow uniformity and steadiness, both of which are essential in a test facility. Separation is usually avoided if the adverse pressure gradients are minimized which is done by making the contraction sufficiently long. The contraction can be divided into two parts. The first part has walls of concave shape and it is very important to elongate this part as much as possible to avoid wall boundary layer separation. The streamline curvature effects on the pressure gradient in the boundary layer promote the risk of separation. Along a fair part of this section, there will be a positive pressure gradient. The second part of the contraction

---

* Corresponding author. Tel.: +20 10 1246 1182.
E-mail addresses: sh_ahmed99@yahoo.com (A.S. Abdelhamed), y_yassen70@yahoo.com (Y.El-S. Yassen), elsakka_mohamed@yahoo.com (M.M. ElSakka).

Peer review under responsibility of Ain Shams University.

ELSEVIER Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.asej.2014.09.008

Nomenclature

a	concave-straight line deviation relative to the contraction inlet half height, h
b	convex-straight line deviation relative to the contraction inlet half height, h
c	axial position of the inflection point relative to the contraction length, L
D_h	test section hydraulic diameter
Uni_{ex}	flow uniformity at the exit plane of the contraction
τ_{min}	minimum wall shear on the contraction, N/m^2
β	porosity
C_2	pressure jump coefficient
k	pressure drop coefficient

has convex walls that may cause flow separation in the vicinity of the test section due to a positive pressure gradient. To delay separation, it is better that a longer contraction's length be chosen, but this will increase the cost and thickening the boundary layer that may enhance boundary layer and risk of separation. Furthermore, if the length is reduced, the contraction costs will reduce and it will fit into a smaller space. In addition, the boundary layer will generally be thinner due to the combined effects of increase in the favorable pressure gradients and decrease in the length of the contraction. Furthermore, it may increase the possibility of flow separation. Thus the length most be optimized. The contraction area ratio is another dominant factor that affects the extent of flow uniformity, flow separation, and downstream turbulence level. The characteristics of the flow were investigated in a square contraction numerically and experimentally [1]. Their Measurements included the cross-sectional velocity profiles and longitudinal pressure distributions along the wall of a contraction of a wind tunnel. Boundary layer was studied in the contraction [2]. The calculations showed that the minimum-length contraction shapes can provide fully attached boundary-layer flow. The results showed that exist regions of separated flow along the wall, in the inlet and exit of the contraction furthermore; conclusions were drawn that the existing adverse pressure gradient is the essential condition for the separation. Flow study shows that a three-dimensional separation occurs in the contraction surface. In a proposed conceptual model of this phenomenon, the separation process begins with small non-uniformities in the boundary-layer flow merging from the screens upstream of the contraction. On entering the contraction, the non-uniformities are amplified by a combination of Gortler instability, lateral pressure gradient and adverse streamwise pressure gradient to form a strong counter-rotating streamwise vortex pair (Gortler vortex) that detaches from the surface [3,4].

Another contraction parameter that has to be selected, a priori, is the cross-sectional shape. In order to avoid cross-flows and boundary layer separation in the corners, the ideal cross-sectional shape is circular. However, in the absence of separation, the secondary flows in the corners tend to remain localized, without any significant effect on the test section flow quality [5]. The cross-sectional shape for modern day contractions is, therefore, almost always chosen to match the other tunnel components which are normally square or rectangular.

An iterative design procedure was developed for the contraction to be installed on the mixing layer wind tunnel [6,7]. The procedure consisted of first computing the potential flow field and hence the pressure distributions along the walls of a contraction of given size and shape using a three-dimensional numerical panel method.

Currently, more flexibility in the design of wind tunnel contractions can be exhibited, with the use of CFD to enable rapid testing of designs to optimize contractions of arbitrary cross-section and wall profile. The use of CFD allows for the use of design exploration algorithm to optimize the profile of the contraction in an automated manner. However, the performance of the contraction still requires testing after construction, as the level of CFD used for this application is typically insufficient to detect the development of longitudinal vortices through the working section such as were measured by [8].

A great effort has been made to set the guidelines for the design of wind tunnels [9,10]. However, the recent development in optimization techniques encourages the researchers to develop optimized design for the wind tunnels especially the wind tunnel contraction that has a strong influence on the flow quality in the test section. An optimization is performed for a 2D contraction profile described by a six order polynomial and the results are validated experimentally [11]. The effective Global Optimization algorithm is used to optimize the profile of a 2D contraction described by a two parameters Bézier curve modeled by a three- dimensional potential flow solver using Kriging Meta model to predict the values of the objective function [12].

The present work contains a detailed account of the redesign of the existing contraction that based on the analytical technique according to the method given by [13,14], assembly and calibration of a wind tunnel specifically redesigned for maximum flow uniformity at the working section, prevention of separation and no Gortler vortices in the contraction, controlling the turbulence level and minimizing the boundary layer thickness at entrance to the working section. The wind tunnel consists of two separate legs which are driven independently by centrifugal blowers connected to variable speed motors. The two streams are allowed to merge in the test section. The construction of the wind tunnel was motivated by a strong interest in the study of three dimensional contraction profiles. The wall contraction profile is described by three dimensional delineated curves and optimized using the design exploration algorithm based on screening method. The flow in the wind tunnel is modeled using a Reynolds-averaged Navier–Stokes (RANS) solver. The aerodynamic performance improvement of the optimized contraction, compared to the old design contraction based on analytical technique, is confirmed using CFD. The new design is manufactured in full scale. Although, investigation by using CFD analysis is less expensive in general, the results obtained from CFD calculations should be validated by means of experimental results. In addition to validation, in cases like simulating three dimensional contraction profiles, performing CFD simulations can