Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 147 (2016) 44 – 49

Procedia Engineering

www.elsevier.com/locate/procedia

11th conference of the International Sports Engineering Association, ISEA 2016

Flow visualization of downhill ski racers using computational fluid dynamics

Takeshi Asai[a*], Sungchan Hong[a], Koichi Ijuin[b]

[a]*University of Tsukuba, Institute of Health and Sports Sciences, Tsukuba, 305-8574, Japan*

[b]*Exa Japan Inc., Yokohama Mitsui Bldg 23F, Yokohama, 220-0011, Japan*

Abstract

In downhill alpine skiing, racers often exceed speeds of 120 km/h, with air resistance substantially affecting the overall race times. To date, studies on air resistance in alpine skiing have used wind tunnels and actual skiers to examine the relationship between the gliding posture and magnitude of drag, as well as for the design of skiing equipment. However, these studies have not revealed the flow velocity distribution and vortex structure around the skier. In the present study, we used computational fluid dynamics with the lattice Boltzmann method to derive the relationship between flow velocity in the full tuck position (the downhill racer's speed) and total drag. Furthermore, we visualized the flow around the downhill racer and examined its vortex structure. The results show that the total drag force in the downhill racer model is 27.0 N at a flow velocity of 15 m/s, increasing to 185.8 N at 40 m/s. Moreover, the visualization of the flow field indicates that the primary drag locations at a flow velocity of 40 m/s are the head, upper arms, lower legs, and thighs (including the buttocks).

Peer-review under responsibility of the organizing committee of ISEA 2016

Keywords: Ski; Drag; Fluid; Downhill; CFD

1. Introduction

In alpine skiing events such as downhill and super-giant slalom, racers often exceed speeds of 120 km/h [1]. Air resistance significantly affects the competition timings. To date, research on air resistance in alpine skiing has considered actual racers in wind tunnels to examine the relation between the gliding posture and the magnitude of drag, and for designing skiing equipment such as the racers' suits [2-4].

Although the total drag on a racer can be calculated by wind tunnel experiments, it is extremely difficult to measure the drag distribution across each part of the body. A more effective way to visualize the flow around the racer and examine the aerodynamic characteristics is to use Computational Fluid Dynamics (CFD) along with Experimental Fluid Dynamics (EFD) in a wind tunnel [5]. In particular, it is important to estimate the drag distribution for each part of the racer's body, and CFD makes such visualization possible. Revealing the drag distribution based on an understanding of the flow field around the racer will also provide the basis for new gliding postures and novel designs for skiing equipment.

In the present study, the relationship between the racer's speed and total drag in the full tuck posture is identified by combining EFD using the wind tunnel and CFD using the lattice Boltzmann method. Moreover, the flow around the racer is visualized in the CFD model to reveal the drag distribution of each part of the racer by studying the vortex structure.

2. Methods

2.1. CFD using lattice Boltzmann method

* Corresponding author. Tel.: +81-29-853-2711; fax: +81-29-853-2711;.
E-mail address: asai@taiiku.tsukuba.ac.jp

1877-7058
Peer-review under responsibility of the organizing committee of ISEA 2016
doi:10.1016/j.proeng.2016.06.187

A three-dimensional downhill racer model (including skis, poles, boots, and helmet) was constructed (Fig. 1) using data obtained from a 3D laser scanner (AICON 3D; Breuckmann GmbH) applied to a real downhill racer. A Cartesian grid form was adopted to generate a spatial grid of size 20 m × 20 m × 40 m (W × H × L), comprising nearly 500 million cells (Fig. 2). A sectional grid scale technique was employed for this study, with a minimum scale of 1 mm and a maximum scale of 4 mm (Fig. 3). This grid structure could not represent the detailed vortex formations perfectly, but was employed because of computational resource constraints [6-7]. The flow speed at the velocity inlet was set to either 15, 20, 25, 30, 35 or 40 m/s. The pressure outlet was defined for 1013.25 hPa. The boundary wall of the downhill racer was assumed to obey a no-slip condition, and the outer walls (including the ground surface) were defined as slip walls.

Fig. 1 Three-dimensional CFD downhill racer model.

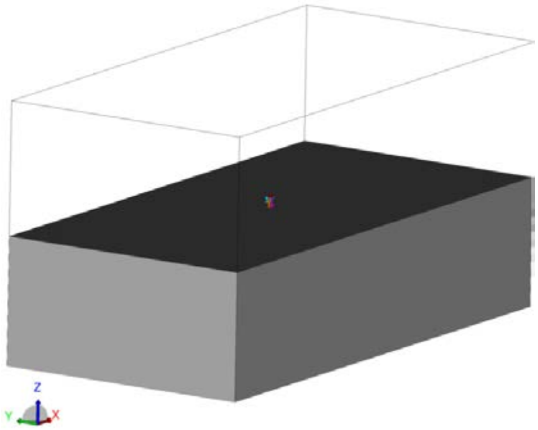

Fig. 2 Cartesian grid adopted to generate the spatial grid (W 20 m × H 20 m × L 40 m) for CFD.

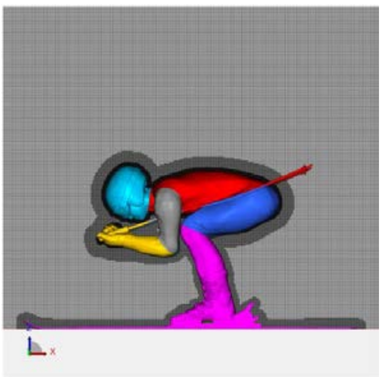

Fig. 3 Sectional grid scale technique for CFD.