Applied Thermal Engineering 84 (2015) 448–459

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research paper

Heat transfer and flow analysis of jet impingement on concave surfaces

Javad Taghinia*, Mizanur Rahman, Timo Siikonen

Aalto University, School of Engineering, FI–00076, Finland

HIGHLIGHTS

- This study assessed the newly developed RAST SGS model and DSM.
- Two common cases in industrial processes are modeled.
- LES performance was better than RNG k-epsilon model in predicting heat transfer.
- The RAST model produced more accurate results than DSM.

ARTICLE INFO

Article history:
Received 1 December 2014
Accepted 26 March 2015
Available online 4 April 2015

Keywords:
Concave surface
Jet impingement
RAST model
LES
Heat transfer
RANS

ABSTRACT

The current study evaluates the performance of three turbulence models in predicting the heat transfer and flow physics of jet impingement on concave surfaces. Two of the applied models are zero–equation subgrid-scale (SGS) models which belong to large eddy simulation (LES), namely the RAST and dynamic Smagorinsky model (DSM), and the third one is RNG k-$\in$ Reynolds Averaged Navier–Stokes (RANS) model. These models are utilized to analyze the heat transfer for two cases: (1) jet impingement on a curved surface with different jet–to–surface distances (2) jet impingement on a heated circular cylinder with varying nozzle–to–surface distances at two different Reynolds numbers. The predicted results are compared with the available experimental data in the literature. The findings revealed that RAST and DSM predictions are in better agreement with experiments than RNG k-$\in$ model. It is also concluded that at higher jet–to–surface ratios, all three models produced almost similar results, proving that the heat transfer distribution and the flow are more affected by the jet–to–surface distance than the magnitude of Reynolds number.

1. Introduction

Jet impingement plays a crucial role in many industrial applications due to its ability to provide localized cooling and heating mechanism in various processes such as surface coating and cleaning, electronics component cooling and paper production. Although, the flow domain may have a simple geometry, the impingement mechanism has a complex structure due to the presence of stagnation point, walls, high streamline curvature and development of shear layer at the free jet region. The jet impingement on a curved surface has significant effect on the flow structure because of a strong flow entrainment and a high streamline curvature in circumferential direction. The curved geometry also alters the heat transfer rate. The heat transfer rate in an impingement process on a concave surface shows an increase of 20% compared with that of a flat surface in similar conditions [1].

There are numerous studies in the literature concerning the impinging jet on flat surfaces with different computational fluid dynamics (CFD) approaches such as Reynolds–avaraged–Navier–Stokes (RANS) and large eddy simulation (LES) for different configurations. For example, El-Gabry and Kaminski [2] applied standard k-$\in$ and low-Reynolds number Yang-Shih models [3] to investigate the flow structure of an array of impinging jets on a flat surface. Both models could not produce accurate predictions of Nusselt number distributions. Miao et al. [4] studied the heat transfer rate of multiple jet impingements over a flat plate. Their study showed that realizable k-$\in$ model can predict acceptable results at higher jet–to–plate distances. Recently, Taghinia et al. [5] performed numerical

* Corresponding author.
E-mail address: javad.taghiniaseyedjalali@aalto.fi (J. Taghinia).

http://dx.doi.org/10.1016/j.applthermaleng.2015.03.064
1359-4311/

Nomenclature

C_μ	eddy–viscosity coefficient
$\overline{C}_s$	Smagorinsky coefficient
G	filter function
g	gravitational acceleration
k	total turbulent kinetic energy
L_{ij}	Leonard stress
Pr	molecular Prandtl number
Pr_{sgs}	sub–grid scale Prandtl number
Re	Reynolds number
$\overline{S}_{ij}$	mean strain–rate tensor
T	temperature
$\overline{u}_i$	grid–filter velocities
$\tilde{\overline{u}}_i$	test–filter velocities
$\overline{u}_\tau$	friction velocity
$\overline{W}_{ij}$	mean vorticity tensor
y^+	dimensionless wall distance $(\overline{u}_\tau y/\nu)$
β	thermal expansion coefficient
$\delta_{i,j}$	Kronecker's delta
Δt	time step
$\overline{\Delta}$	grid–filter width
$\tilde{\Delta}$	test–filter width
ν,ν_T	laminar and turbulent viscosities
$\overline{\theta}_i$	grid–filter temperature
$\tilde{\overline{\theta}}_i$	test–filter temperature
ρ	density
$\tau_{i,j}$	sub–grid scale stress tensor
LES	large eddy simulation
RANS	Reynolds averaged Navier–Stokes
RAST	Rahman–Agarwal–Siikonen–Taghinia
DSM	Dynamic Smagorinsky Model
SGS	sub–grid scale

Subscript

i,j	variable numbers
in	inlet condition
out	outlet condition

simulation of twin-jet impingement on a flat surface with an hybrid LES-RANS model. They found satisfactorily predictions in terms of mean flow parameters such as pressure and velocity distributions at higher jet–to–plate ratios. By considering the above mentioned studies, there are not many documents available on heat transfer and flow structure of impinging jet on concave surfaces. Most of the existing works deal with experiments of flow parameters in impingement process showing that the impinging characteristics on curved surfaces are different from those of flat plates [6,7]. There are a few reported studies on numerical investigation of impinging jets on concave surfaces. Most of these studies applied an RANS modeling approach in analyzing the flow behavior for impinging jet. Souris and Liakos [8] numerically investigated the flow field from an impinging jet on a semi–cylindrical surfaces by applying the standard k-$\in$ and Reynolds stress transport model (RSM). They found relatively good agreement between model predictions and experimental results of Choi et al. [9]. In an another attempt, Kayansayan and Kucuka [10] carried out numerical simulations for a confined jet impingement on a concave surface for various Reynolds numbers and compared the results with their own experimental data. They found a maximum deviation of 28% between the predictions and measurement at the impinging zone. Frageau et al. [11] studied the heat transfer of an array of jets on a concave surface of an airfoil for various nozzle–to–surface spacings. They applied Spalart-Allmaras turbulence model and found satisfactory results for high nozzle–to–surface heights. The heat and flow field from a row of round jets on a concave semi–cylinder was investigated by Craft et al. [12]. In their study, linear and non–linear k-$\in$ model along with a wall function were employed. They concluded that the applying a standard wall function leads to under–prediction in Nusselt number values. Sharif and Mothe [1] applied RSM, standard k-$\in$, RNG k-$\in$ and SST k-ω RANS models to simulate the impingement on a curved surface. All these models failed to accurately predict the Nusselt number distribution along the impinging surface. However, RSM showed relatively better performance in reproducing the flow features. In a more recent effort, Singh et al. [13] conducted a numerical study of heat transfer of a jet impingement on a circular cylinder with various RANS turbulence models. They used different nozzle-to–surface distances and nozzle diameters in their study. They reported that all the applied turbulence models in all configurations over–estimated the Nusselt number at the impinging region.

According to the above literature review as mentioned earlier, the heat transfer and flow field analysis of impinging jets on concave surfaces are less investigated. Apart from this fact, all the available literature concerning the numerical investigation of this subject are based on RANS methods which could not produce accurate results. To the authors' knowledge, there are no reported investigations on this phenomenon based on LES. Therefore, establishing a numerical study based on this method is essential for such a demanding case.

With the recent development in computational power, large eddy simulation (LES) has became a powerful approach to simulating the flows with separation and recirculation. The under-lying mechanism in an LES gives this method a special ability to capture even the smallest turbulent structures affecting the flow topology. The LES decomposes the flow field into two scales based on eddy sizes, namely the large scale and sub–grid scale (SGS). Large eddies are solved directly while the small ones are modeled. The SGS eddies are nearly isotropic and independent of the flow geometry having universal characteristics. Therefore, the SGS models require a fewer empirical coefficients than those of RANS models. The main differences among the available LES approaches lie in the applied SGS models embedded in them. Different SGS models have been introduced since the last three decades among which Smagorinsky [14] and dynamic Smagorinsky model (DSM) [15] are the most popular ones. The Smagorinsky model benefits from a constant eddy–viscosity coefficient which is simple and robust, but it is not suitable for complex flows in which the coefficient changes with time and space. On the other hand, the DSM computes the eddy–viscosity coefficient locally varying in time and space.

This paper aims at investigating the heat transfer and flow field of an impinging jet on concave surface by applying an LES with different SGS models, namely RAST (Rahman–Agarwal–Siikonen–Taghinia) [16] and DSM [15]. The RAST model utilizes a variable eddy–viscosity coefficient which responses to the anisotropic characteristics of turbulence and non–equilibrium. This feature allows the model to adjust itself with a rapid change in flow structure, particularly close to the solid boundaries. These models are applied for two cases in