IMPACT OF TURBULENCE MODELING ON FLUID/SOLID HEAT TRANSFER INSIDE INDUSTRIAL AUTOCLAVES

ParCFD2022
$33^{rd}$ International Conference on Parallel Computational Fluid Dynamics
May 25-17 2022, Alba Italy

# IMPACT OF TURBULENCE MODELING ON FLUID/SOLID HEAT TRANSFER INSIDE INDUSTRIAL AUTOCLAVES.

**Luca BANETTA*, Luca CATTAROSSI+, Daniela TORDELLA* AND Daniele L. MARCHISIO***

* Department of Applied Science and Technology, Politecnico di Torino
Corso Duca degli Abruzzi 24, 10129 Torino(TO) Italy
e-mail: luca.banetta, daniela.tordella, daniele.marchisio@polito.it, web page: https://www.polito.it

+ Department of Mechanical and Aerospace Engineering, Politecnico di Torino
Corso Duca degli Abruzzi 24, 10129 Torino(TO) Italy
e-mail: s277928@studenti.polito.it, web page: https://www.polito.it

**Key words:** Detached Eddy Simulations, RANS , Chemical Engineering, Heat exchange

**Abstract.** This work is centred on the analysis of the impact of different turbulence modeling approaches on the fluid/solid heat exchange inside a commercial size autoclave. This project proposes itself to be a first step towards the optimization of the turbulent flow inside this kind of machinery to improve the curing treatment of Carbon-Fiber Reinforced Plastics (CFRP). The setup of the CFD simulations includes the presence of a metallic sample object inside the autoclave, where air will be recirculated with velocity, pressure and temperature typically adopted for this type of treatments. The analysis takes advantage of parallel CFD simulations, conducted by using the open-source software openFOAM v2106. Two turbulence models have been adopted: one is the well-known Reynolds-Average Navier-Stokes approach (RANS), which is currently used to model the turbulence inside this type of machinery. The second one is the Delayed Detached Eddy Simulations (DDES), which allows the full resolution of the majority of turbulent scales around the sample object. First, we propose the difference between the local heat flux distribution at the air/solid interface computed by using RANS and DDES, next we analyse the overall heat flux entering the sample object: the resolution of the turbulent scales does not influence the local heat flux only, but also the overall heat flux entering the object; an average increase of 35% is reported when the velocity fluctuations are neglected. Future steps of the research foresee the analysis of the heat flux and temperature distributions on the surface of realistic shapes and common-use CFRP. Afterwards, the autoclave design will be optimized by adding multiple inlets and aerodynamic devices to guarantee a more homogeneous heat flux distribution on the surface of realistic shapes of actual CFRP.

## 1 INTRODUCTION

Carbon-fiber reinforced plastics (CFRP) have been extensively used in a variety of fields, including the aviation industry, due to their advantages of high specific stiffness

and strength. Their mechanical performances are mainly designed by autoclave processing under high pressure and elevated temperature: the procedure is based on an heat transfer from a pressurized fluid under turbulent conditions to the surface of the material to be treated.
It is clear that fluid dynamics plays a pivotal role in this scenario, since the nature of the air flow influences the heat transfer from the fluid to the solid surface.
CFD studies centered on air flows inside autoclaves have been proposed over the last few years [1, 2]. Information derived from them has then been used to optimize the geometry and the location of the materials to be treated. We notice that all the studies are based on the pure RANS approach, in which the impact of the velocity fluctuations is entirely neglected. This project proposes itself to analyse the impact of the inclusion of different turbulent structures in determining the heat flux transferred from the technical fluid to a sample object in an industrial autoclave. What follows is the first step of the project, which involves the comparison between DDES and RANS in the evaluation of the heat flux exchanged by the air to a metallic sample object inside the chamber of a commercial-size autoclave.

## 2 THEORETICAL BACKGROUND

The theory behind CFD simulations is based on the application of the Reynolds decomposition and average to the Navier-Stokes equations; since we deal with heat transfer phenomena, the balance equation for the enthalpy $h$ is solved as well. The resulting Reynolds stress tensor in the momentum conservation equations is then correlated to the average velocity $\bar{\mathbf{U}}$ by the Boussinesq hypothesis, and the closure models the turbulent viscosity $\mu_{\mathrm{t}}$ and thermal conductivity $\lambda_{\mathrm{t}}$ as a function of the turbulent features of the system. The closure for the pure RANS system is the well known k-$\omega$ SST approach [3], in which $\mu_{\mathrm{t}}$ and $\lambda_t$ are modeled as a function of the turbulent kinetic energy (TKE) $k$ and its specific dissipation rate $\omega$:

$$\begin{cases} \dfrac{\partial(\rho_{\mathrm{f}}\ k)}{\partial t} + \nabla\cdot(\rho_{\mathrm{f}}\ k\ \bar{\mathbf{U}}) = \tilde{P}_k - \beta^*\rho_{\mathrm{f}}k\omega + \nabla\cdot\left[\left(\mu_{\mathrm{f}} + \sigma^*\mu_{\mathrm{t}}\right)\nabla k\right] \\ \dfrac{\partial(\rho_{\mathrm{f}}\ \omega)}{\partial t} + \nabla\cdot(\rho_{\mathrm{f}}\ \omega\ \bar{\mathbf{U}}) = \alpha\dfrac{\omega}{k}\tilde{P}_k - \beta\rho_{\mathrm{f}}\omega^2 + \nabla\cdot\left[\left(\rho_{\mathrm{f}} + \sigma_\omega\mu_{\mathrm{t}}\right)\nabla\omega\right] + 2(1-F_1)\dfrac{\sigma_{\omega 2}\rho_{\mathrm{f}}}{\omega}\nabla k\nabla\omega, \end{cases} \tag{1}$$

where $\rho_{\mathrm{f}}$ is the fluid density, $\mu_{\mathrm{f}}$ its viscosity, $\tilde{P}_k$ the generation of TKE and, finally, $\alpha$, $\beta^*$, $\beta$, $\sigma$, $\sigma_\omega$, $F_1$ and $\sigma_{\omega 2}$ are model-dependent coefficients.
DDES is, instead, an extension of the RANS approach, in which the dissipation term of the equation for $k$ is modified, so that eventual turbulent structures are rigorously solved if the local grid size $\Delta$ multiplied by a weighting factor $C_{\mathrm{DES}}$ is smaller than the local turbulent characteristic length $l_{\mathrm{RANS}} = \sqrt{k}/(C_\mu\omega)$; the final equation for TKE becomes

[4]:

$$\begin{cases} \dfrac{\partial(\rho_\mathrm{f}\ k)}{\partial t} + \nabla\cdot(\rho_\mathrm{f}\ \bar{\mathbf{U}}\ k) = \tilde{P}_k - \beta^*\rho_\mathrm{f}\dfrac{k^{3/2}}{d} + \nabla\cdot\left[\left(\mu_\mathrm{f} + \sigma^*\mu_\mathrm{t}\right)\nabla k\right]; \\ d = l_\mathrm{RANS} - f_d \max\left(0, l_\mathrm{RANS} - C_\mathrm{DES}\Delta\right), \end{cases} \tag{2}$$

where $f_d$ is an empirical shield function, which smooths the transition from modeled to solved turbulent structures to prevent grid-induced separations.

## 3 SIMULATIONS SETUP

A simplified geometry adopted of the actual autoclave is proposed in Figure 2. This

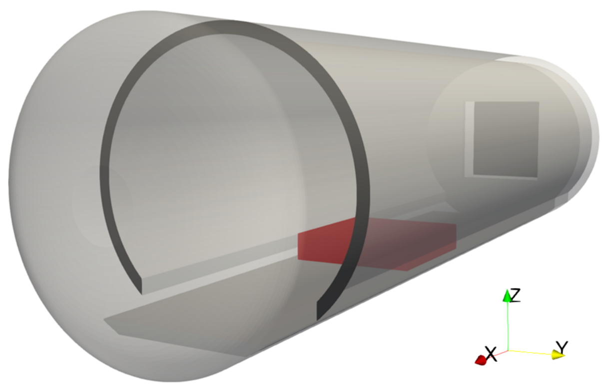


**Figure 1**: 3D image of the geometry of the computational domain.

kind of machinery is a closed system, where the technical fluid is recirculated, but for simplicity we will consider it as an open equivalent: the black circular zone represents the inlet, from which the fluid enters the chamber, meanwhile the exit is highlighted as a black square located at the far end. Finally, The design foresees the presence of a support, over which the metallic sample has been located.

At the beginning, the air inside the chamber is stagnant and pressurized at $P_0$, while the temperature of both the fluid $T_{\mathrm{f},0}$ and solid phase $T_{\mathrm{s},0}$ is equal to the environment conditions. Hot air subsequently enters from the inlet with temperature $T_{\mathrm{a,in}}$ and velocity with module $U_0$ parallel to the longitudinal axis. It will subsequently flow over the sample inside the chamber and finally go through the exit. Due to proprietary reasons, neither the the structural features of the autoclave nor its operative conditions can be disclosed. The transport properties of air, $\rho_\mathrm{f}$, $\mu_\mathrm{f}$, $\lambda_\mathrm{f}$ and specific heat $C_P$ will be modeled as a function of the temperature inside the chamber; on the other hand, the transport properties of the solid phase will be considered as constant.

A slice of the adopted computational grid is proposed in Figure 2: it is composed of 22 million hexahedral cells, the majority of them used to model the turbulent boundary layer at the fluid/ solid interface. Simulations have been run adopting 32 processes in parallel, using two High-Performance Computing clusters: one is *Galileo100* provided by CINECA, the second one is *Legion* provided by HPC@POLITO.

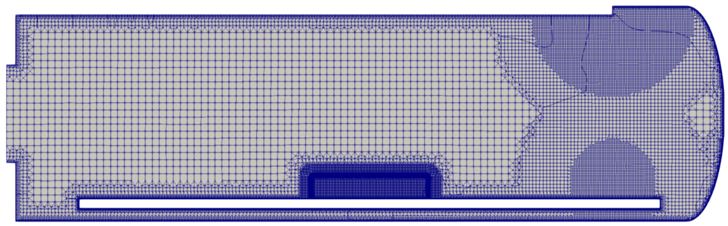

**Figure 2**: Longitudinal section of the adopted computational grid built by using the openFOAM tool *snappyHexMesh*: different levels of refinement have been applied around the inlet area and the sample object.

## 4 DISCUSSION

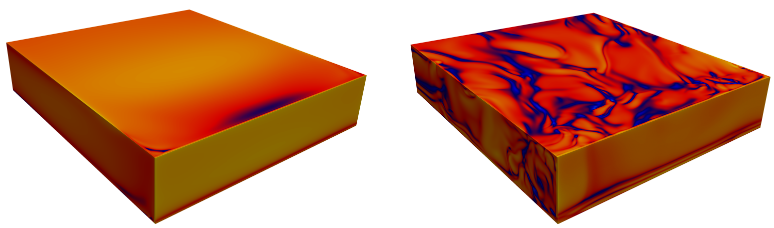

**Figure 3**: Spatial distribution of $J_{\text{fluid/solid}}$ after 10 s of simulation using RANS (left) and DDES (right).

The target variable is the local heat flux entering the sample object at time t and at every surface location $\mathbf{x}$, which is computed as a function of the enthalpy gradient next to the surface of the sample object:

$$J_{\text{air/solid}}(\mathbf{x}, t) = \frac{\lambda_{\text{f}}}{C_P} \nabla h(\mathbf{x}, t), \tag{3}$$

The comparison between the heat flux distribution over the sample object is proposed in Figure 3. It can be seen quite clearly that the resolution of the local turbulent structures leads to an in homogeneous heat flux distribution, meanwhile the use of the RANS approach leads to a levelling of the local values of $J_{\text{air/solid}}$ across the surface $S$ of the sample object. The impact of the resolution of the detached turbulent structures is not limited to the local values of the heat flux only. In Figure 4 we propose the temporal trend of the whole heat flux $J^{\text{int}}_{\text{fluid/solid}}(t)$ obtained by the two turbulence models; it is computed by integrating $J_{\text{fluid/solid}}(\mathbf{x}, t)$ over the whole surface S. Once the hot air circulating in the chamber starts heating the sample object, the flux computed by the different methodologies differs quite significantly. Once a pseudo steady state has been reached, the standard RANS approach evaluates an overall heat flux from 30%to 40% higher than the one obtained by resolving the turbulent scales by using DDES.
Concluding, this work has proved that neglecting the local velocity fluctuations can lead to an overestimation of the overall heat exchange over the surface of a sample object. This clearly can lead to an erroneous evaluation of the temperature distribution inside the solid phase. Future steps on this research will include an experimental validation, in

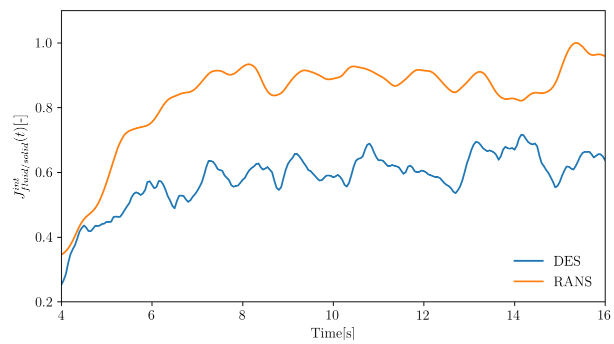


**Figure 4**: Temporal trend of the normalized integral of the module of the wall heat flux at the fluid/solid interface obtained using the different turbulence models; the values have been normalized due to proprietary reasons.

which the impact of the divergence of the results between RANS and DDES is compared with empirical measurements of the temperature distribution on actual CFRP with realistic shapes. The ulterior step forward will be the optimization of the turbulence of the air flow entering the chamber by using additional inlets and active grids to make the turbulence more isotropic to guarantee a uniform heat flux over the fluid/solid interface.

## 5 ACKNOWLEDGMENTS

The authors acknowledge AlmaFLUIDA project, done in collaboration with *Italmatic group*, and CINECA for the computational resources. Additional resources were provided by HPC@POLITO, a project of Academic Computing within the Department of Control and Computer Engineering at the Politecnico di Torino (http://www.hpc.polito. it).

## REFERENCES

[1] T. Bohne, T. Frerich, J. Jendrny, J.-P. Jürgens, and V. Ploshikhin, "Simulation and validation of air flow and heat transfer in an autoclave process for definition of thermal boundary conditions during curing of composite parts," *Journal of Composite Materials*, vol. 52, no. 12, pp. 1677–1687, 2018.

[2] J. Zhu, T. Frerich, and A. S. Herrmann, "Cfd modeling and validation of heat transfer inside an autoclave based on a mesh independency study," *Journal of Composite Materials*, vol. 55, no. 18, pp. 2469–2487, 2021.

[3] F. R. Menter, "Influence of freestream values on k-omega turbulence model predictions," *AIAA journal*, vol. 30, no. 6, pp. 1657–1659, 1992.

[4] M. S. Gritskevich, A. V. Garbaruk, J. Schütze, and F. R. Menter, "Development of ddes and iddes formulations for the k-$\omega$ shear stress transport model," *Flow, turbulence and combustion*, vol. 88, no. 3, pp. 431–449, 2012.