Thin Solid Films 518 (2010) 6676–6681

Contents lists available at ScienceDirect

Thin Solid Films

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

Design evaluation of the semi-prototype for the ITER blanket first wall qualification

Dong Won Lee [a], Young Dug Bae [a], Suk Kwon Kim [a], Sun Ho Kim [a], Bong Guen Hong [a], In Cheol Bang [b,*]

[a] Korea Atomic Energy Research Institute, Daejeon, Republic of Korea
[b] Ulsan National Institute of Science and Technology, Ulsan, Republic of Korea

ARTICLE INFO

Available online 8 April 2010

Keywords:
ITER
First Wall
Hypervapotron
ANSYS-11

ABSTRACT

For the second qualification of the First Wall (FW) procurement of the International Thermonuclear Experimental Reactor (ITER), a semi-prototype of the FW has been designed with increased local surface heat flux up to 5 MW/m^2. With the given conditions, the new semi-prototype design was simulated with the commercial CFD code, the ANSYS-11. The results show that the semi-prototype temperature exceeds the melting temperature of Be, and the current design is required to be modified. In order to enhance cooling, a hypervapotron was added in the design and an analysis with the same code was performed. The results show that the temperature with the hypervapotron reduced by around 100 °C but it was still higher than the melting temperature of Be. The hypervapotron mock-up was fabricated and tested with a variance of inlet coolant flow rates and heat fluxes of up to 1.75 MW/m^2 using the second Korea Heat Load Test (KoHLT-2) facility, in which heat was loaded by a graphite heater through radiation heating. Wall and coolant temperatures were measured and compared with the simulation results. So far, there is a large difference between the experiments and the simulation, and a next experiment is being prepared.

1. Introduction

The First Wall (FW) of the International Thermonuclear Experimental Reactor (ITER) is an important component that directly faces the plasma and therefore, it is subjected to a high heat and neutron loads, and composed of a beryllium (Be) layer as a plasma facing material, a copper alloy (CuCrZr) layer as a heat sink and type 316L authentic stainless steel (SS316L) as a structure material. In the initial design of the FW, the normal and peak heat loads were assumed to be 0.3 and 0.5 MW/m^2, respectively, but the peak value was changed to 5 MW/m^2, recently. Therefore, the FW panel design has been changed for enhancing the cooling, which was provided by ITER Organization.

In the present study, the proposed design was evaluated with the Computational Fluid Dynamics (CFD) code, ANSYS-11, especially for the materials' temperature such as Be, Cu-alloy, and SS316L. It shows that the temperature of the Be surface increases over the melting point and the design should be modified. Therefore, hypervapotron was adopted to enhance the cooling in the FW design and evaluation was performed with the same code. Moreover, the performance of the hypervaptron is being verified experimentally with the KoHLT-2 (Korea Heat Load Test facility 2) at KAERI (Korea Atomic Energy Research Institute).

2. Performance analysis with the semi-prototype design

Analyses with ANSYS-11 used a 3-dimensional model which were performed for the new FW design, especially for the semi-prototype of the 2nd qualification program [1]. Fig. 1 shows the 3-dimensional modeling of it and a selected finger, in which the latter was used for analysis in this study. According to the ITER operation condition, heat flux was loaded with 5 MW/m^2 in the local region (here, about 26 mm) and coolant conditions were used as follows; 100 °C of temperature and 3 MPa of pressure, 6.914 kg/s of flow rate. Fig. 2-(a) shows the overall temperature distribution without hypervapotron and the maximum temperature of Be surface at 1775 °C, which is higher than the melting point (1283 °C). Even though using hypervapotron, the temperature exceeds the melting point for Be but the enhancement of cooling when using hypervapotron can be observed as shown in Fig. 2-(b). Therefore, the design of the current semi-prototype should be modified and some device for enhancing the cooling should be considered as well.

3. Experimental work

In order to verify the hypervapotron function and analysis capability of ANSYS-11, experiment with a hypervapotron mock-up was performed with KoHLT-2, which was developed for ITER blanket FW [2]. The mock-up was designed and fabricated as shown in Fig. 3; 500 mm in length, 49.2 mm in width, 30 mm in height. It was made from Cu and the flanges from stainless steel were welded to be installed at the facility. The maximum power of the facility was 80 kW

* Corresponding author.
E-mail addresses: dwlee@kaeri.re.kr (D.W. Lee), icbang@unist.ac.kr (I.C. Bang).

0040-6090/$ – see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.tsf.2010.03.149

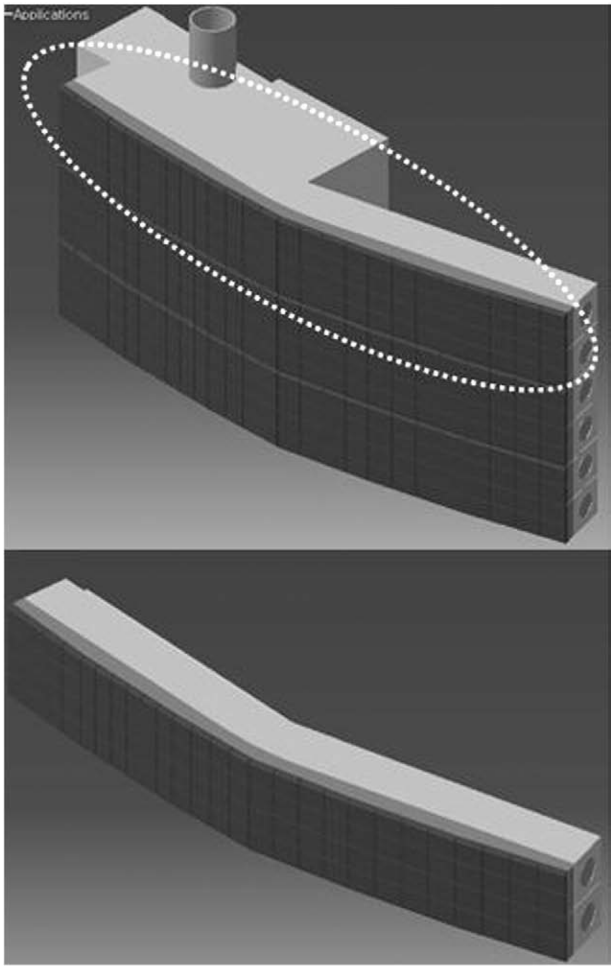

Fig. 1. Semi-prototype model of the new FW and its finger for analysis.

and it was able to produce the surface heat flux up to 1.0 ~ 3.0 MW/m^2. Its water supply system has the capability to supply water of 25 to 120 °C temperature and 3 MPa pressure. Fig. 4 shows the photograph and the overall schematic of the facility. Fig. 5 shows the graphite heater (80 mm × 80 mm) and installed Cu dummy as a calorimeter. On the opposite side of the Cu dummy, the hypervapotron mock-up was installed.

During the test, the absorbed power at the tested mock-ups was measured with a coolant temperature difference by thermocouples located at the inlet and outlet regions and the total power by a using a calorimeter, Cu dummy mock-up was compared to it. One thermocouple for the wall temperature measurement was attached on its backside. The mock-up was tested with a variance of inlet coolant flow rates and heat fluxes of up to 1.75 MW/m^2 heat flux. At a certain coolant flow rate, the heat fluxes were increased incrementally (0.25, 1.0, and 1.75 MW/m^2) and the power was kept to be constant about 2 min for reaching the steady conditions at each heat flux. Fig. 6 shows the measured temperature and calculated heat fluxes at 0.972 m/s of coolant inlet velocity. Other two cases with 0.505 and 0.762 m/s of coolant inlet velocities were performed.

Analyses with ANSYS-11 used a 3-dimensional model which were performed to compare with the test results and also to apply the virtual design for design optimization. Fig. 7 shows the meshes for the analysis from 3-dimensional model. The heated wall was modeled to be 49.2 mm × 80 mm as the same as the test conditions. The general k-e model for turbulent flow was used [3]. Fig. 8 shows the calculated temperature distributions of the mock-ups for the following case; 1.75 MW/m^2 heat flux and 0.972 m/s inlet coolant velocity. The maximum temperature in the surface is 320 °C and the temperature near the installed thermocouple is 76 °C. For all cases, simulation results with ANSYS-11 and experimental ones were compared as shown in Fig. 9. Higher heat flux was loaded, the discrepancy between the measured temperature and the analysis ones became larger. The error seems to be caused by wall temperature measurement not considering the radiation cooling at the back side of the mock-up and with bad contact of the thermocouples with it due to the thermal expansion. Therefore, the radiation effect will be considered in the

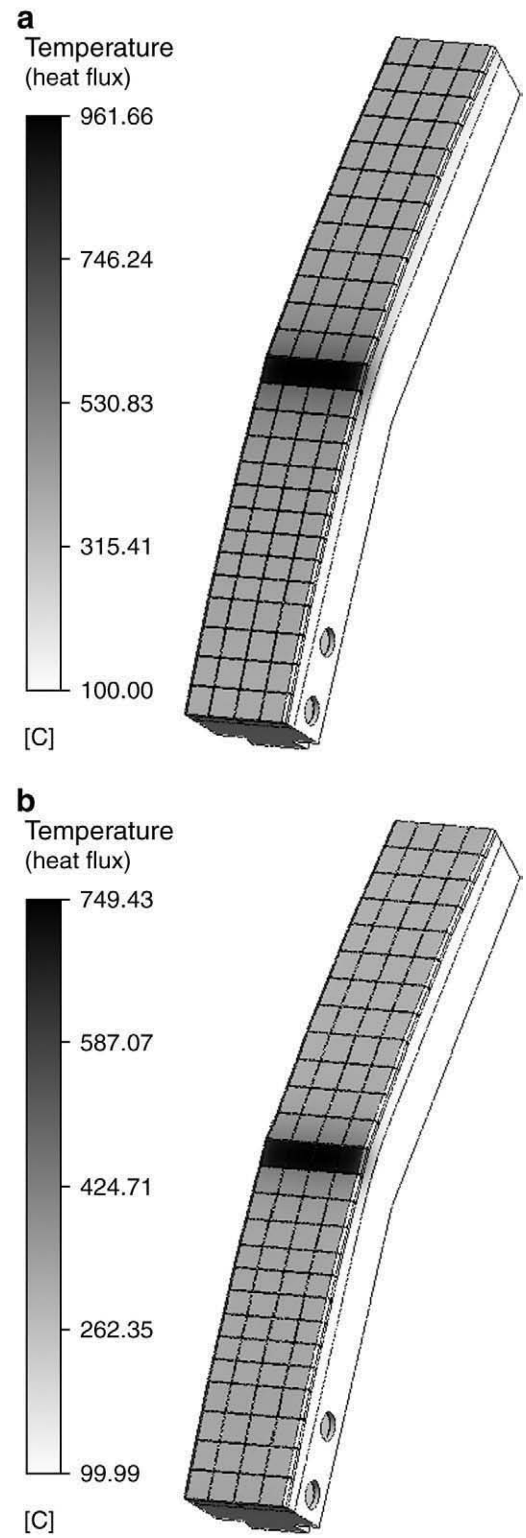

Fig. 2. Analysis results of the FW finger without/with hypervapotron.

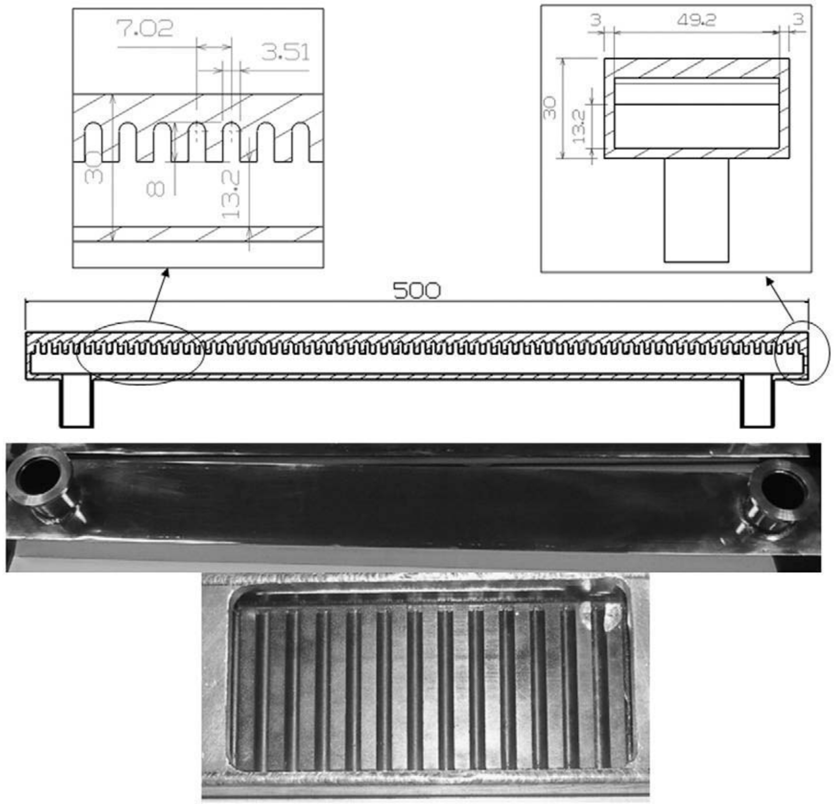

Fig. 3. Draft and photos of the fabricated hypervapotron mock-up.

analysis and more thermocouples will be installed in the mock-up, which located at nearer heating surface and with care for better contact.

4. Conclusion

In order to evaluate the proposed design for semi-prototype of ITER blanket FW, ANSYS-11 was used for simulation with the ITER operation conditions. The results show that the temperatures were too high so that the design should be modified and some devices such as hypervapotron should be adopted to enhance the cooling. For confirming the feasibility of using the hypervaptron, the mock-up was fabricated and tested in the KoHLT-2 as a first step in Korea with a variance of inlet coolant flow rates and heat fluxes of up to 1.75 MW/m^2. Simulation with ANSYS-11 was performed with the same model as the mock-up for the experiment. The results show a large difference in temperature between CFD code and measured ones. Further experiment and analysis will be done in the near future with care for better contact of thermocouples and with considering the radiation effect in the mock-up.

References

[1] ANSYS CFX-11, 2007, User Manual, ANSYS-CFX.
[2] Y.D. Bae, et al., Development of a high heat flux test facility for plasma facing component, Fus. Sci. Technol. 56 (2009) 91–95.
[3] S. Pascal-Ribot, et al., 3D numerical simulations of hypervapotron cooling concept, Fus. Eng. Des. 82 (2007) 1781–1785.

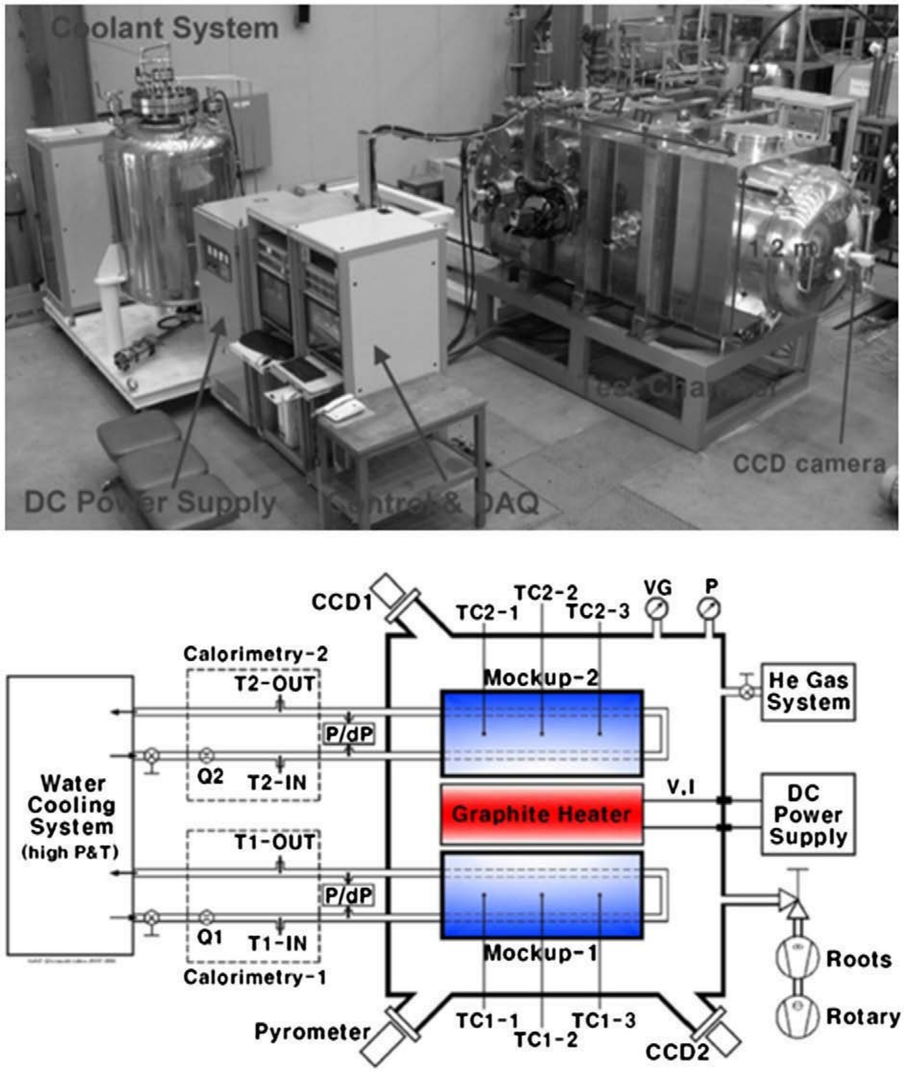

Fig. 4. Photo and schematic of the KoHLT-2 facility.

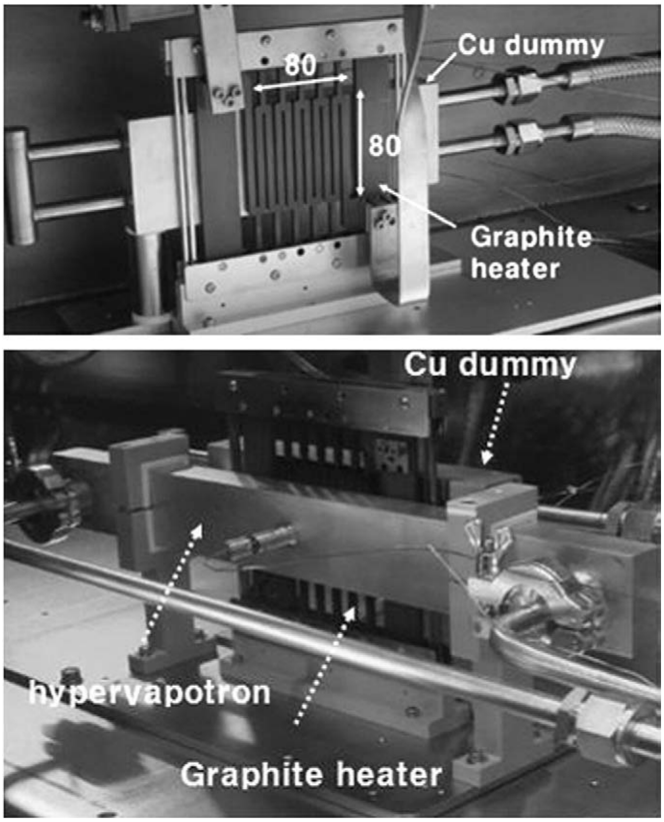

Fig. 5. Photo of the installed graphite heater and mock-ups.

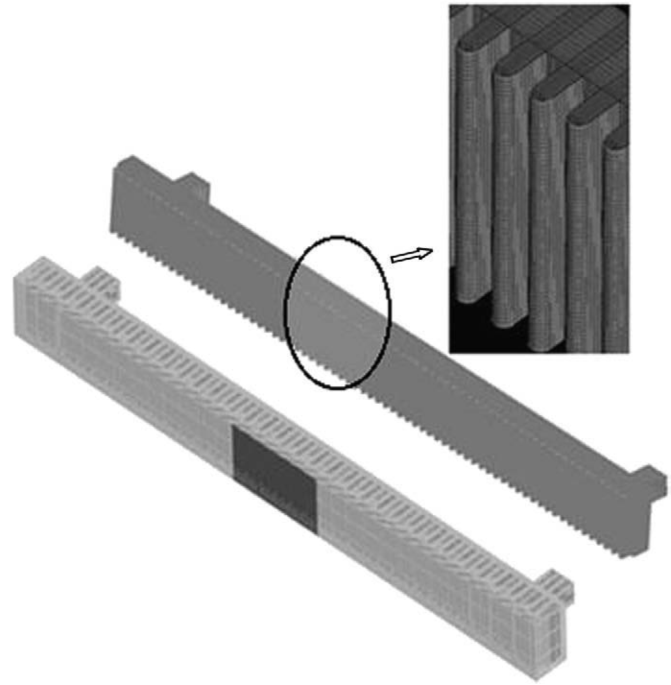

Fig. 7. Generated meshes for analysis of hypervapotron mock-up.

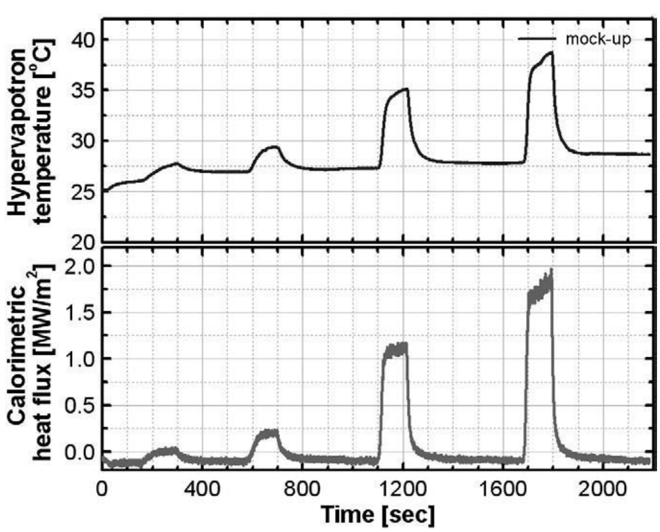

Fig. 6. Raw data at 0.972 m/s of inlet coolant velocity.

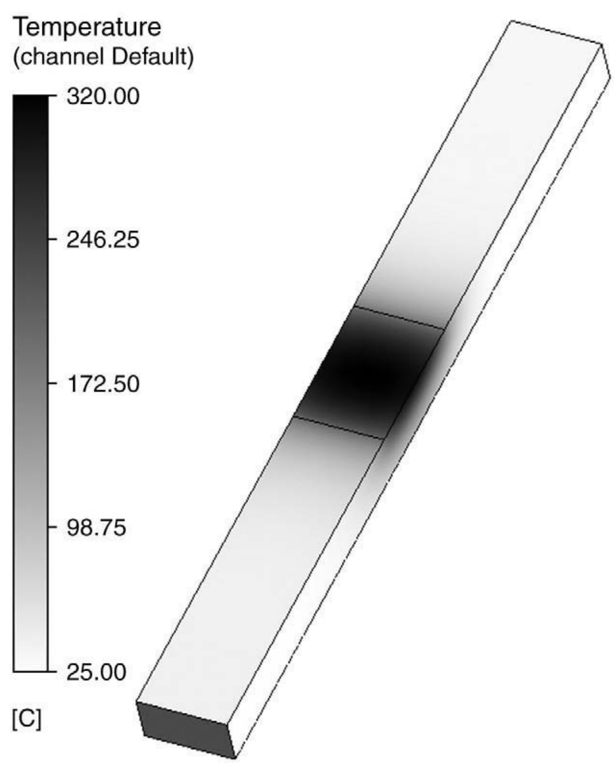

Fig. 8. Analysis results with 0.972 m/s velocity and 1.75 MW/m^2 heat flux.

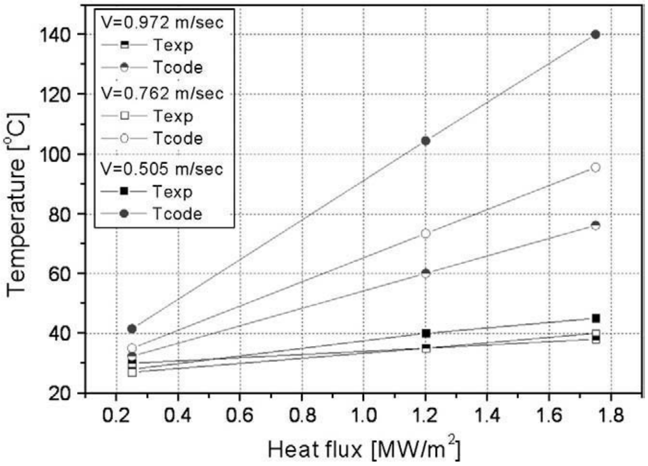

Fig. 9. Comparison of experimental data with analysis ones.