M&C 2017 - International Conference on Mathematics & Computational Methods Applied to Nuclear Science & Engineering, Jeju, Korea, April 16-20, 2017, on USB (2017)

CFD modeling of the 37 rod bundle experiment

V.Yu. Volkov, L.A. Golibrodo, A.A. Krutikov, O.V. Kudryavtsev, A.P. Skibin,* L.L. Kobzar, D.A. Oleksyuk†

*OKB "GIDROPRESS", Podolsk
†National Research Center "Kurchatov institute", Moscow
skibin_ap@grpress.podolsk.ru, oleksuk_da@nrcki.ru*

Abstract - *This paper presents use of commercial software STAR-CCM+ for single-phase CFD analysis of conjugate heat transfer experimental facility "KS" located in National Research Center "Kurchatov institute". The facility used in this analysis represents 37 rod bundle with 16 spacer grids of the TVS-2M fuel assembly for water-cooled water-moderated nuclear power reactor (VVER). The main goal of the work is to study a coolant mixing in rod bundle with spacer grids. The computational investigation was carried out in OKB "GIDROPRESS". The experimental data and results of subchannel SC-INT code used to validate the obtained CFD results. The results of comparison of CFD with experimental and subchannel code data show that CFD data is able to obtain approximation functions and closure correlations for thermal-hydraulic.*

I. INTRODUCTION

A research of thermal-hydraulic characteristics of TVS-2M fuel assemblies (FA) was performed to validate enhancement of thermal power of operating VVER nuclear power plants [1].

Experimental and computational investigations of coolant mixing in rod bundles are the two ways to study heat transfer in FA.

An important characteristic of the experimental facility is the number of fuel rod simulators. A periphery in small bundles with a number of rods (7, 19) takes a significant part in the cross-section. It results to considerable uncertainty in the experimental data. To obtain representative data on heat transfer in spacer grid area it is necessary to carry out a research of the bundles with 37 rods or more. This allows one to get more realistic and representative experimental data for development of reliable computational methods [2].

CFD modeling and its corresponding post-processing may be used to obtain approximation functions and closure correlations for thermal-hydraulic subchannel codes [3, 4].

This paper describes a CFD simulation of conjugate heat transfer of the 37 rod bundle experimental facility. Experimental studies on facilities with rod bundle have additional objectives to validate thermal-hydraulic codes which are used in the design process of FA spacer grids [2, 5]. A comparison of CFD with subchannel code results was performed as well [2]. SC-INT code was developed by National Research Center "Kurchatov institute".

II. WORK DESCRIPTION

The main goal of the work is to perform computational investigation to study coolant mixing in fuel rod bundles with spacer grids. Calculations were performed with the commercial code STAR-CCM+ v10.02 [6] using Reynolds-averaged Navier-Stokes (RANS) based turbulence models. The CFD results were validated by comparison with experimental data and with results of subchannel code SC-INT [2] was done.

1. Facility description

An electrical heated test section of the experimental facility "KS" comprises 37 fuel rod simulators bundle encased in a hexagonal electrical insulation channel and a vertical tube. Fuel rods simulators were heated by means of rectified electric current. The heat generated is removed by means of water.

A fuel rod simulator comprises a stainless steel vertical tube with outside diameter of 9.00-9.07 mm. Simulators are arranged in a triangular lattice with a 12.75 mm pitch. The heated length of the simulators bundle is 2500 mm and the width across flats of hexagonal channel is 79.2 mm [2].

Spacer grid consists of 25 central and 12 peripheral cells [2]. Spacers are identical to TVS-2M fuel assemblies. The experimental tests were carried out under pressure ~12 MPa and temperature over the range 195-350°C.

The relative heat generation for fuel rods simulators is shown in Figure 1. The measurement system consists of 20 thermocouples placed in measurement section (see Figure 2).

2. Problem statement

The computational domain consists of test section (37 rod imitators and 16 spacer grids) and outlet section. Figure 2 shows the computational domain. First and second kind boundary conditions were set for inlet and outlet, respectively. Conjugate heat transfer was simulated[7]. Heat flux was expected to be circumferentially constant and set on inner pin boundary. Computational domain presented in Figure 2.

Solid model of the spacer grid with fuel rod simulators is shown in Figure 3.

The following assumptions were set for CFD modeling:

- influence of the inlet part is not considered;
- non-slip wall boundaries;
- coolant is Nutonian fluid;
- turbulent flow;
- steady-state simulation;

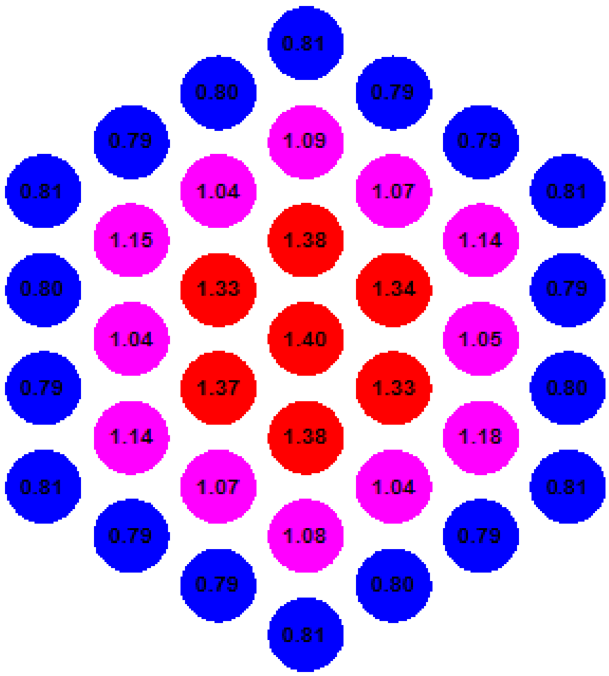

Fig. 1. Fuel rods imitators relative heat flux

- thermophysical properties of fluid depend on pressure and temperature and in agreement with the STAR-CCM+ IAPWS-97 standard [6, 8]. Thermophysical properties of material of spacer grid and simulator depend on temperature.

A computational simulation of three experimental cases was carried out [2]. Test parameters and boundary conditions values for the cases are presented in Table I.

Case number	1	2	3
Pressure at the outlet, bar	115.8	115.7	116.4
Inlet temperature, °C	195	198.7	257.5
Flow rate, kg/(m^2·s)	2177	3909	1054
Electrical power, kW	2323	3742	540
Maximum heat flux, MW/m^2	1.243	2.002	0.288

TABLE I. Test parameters for experiment cases

3. CFD mesh

The mesh model, based on polyhedral control volumes, was created using the pre-processing capabilities of the code STAR-CCM+. The extrusion was used for creation of mesh at the bundle. The mesh was generated for three separate regions (fluid, simulators and spacer grid) simultaneously. Prismatic boundary layers were generated for fluid region. A Thin Mesher was used to generate a prismatic type volume mesh for the rod bundle and spacer grid with 3 and 2 cells

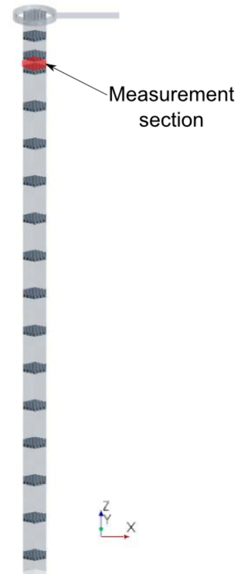

Fig. 2. Computational domain (Z scale=0.5)

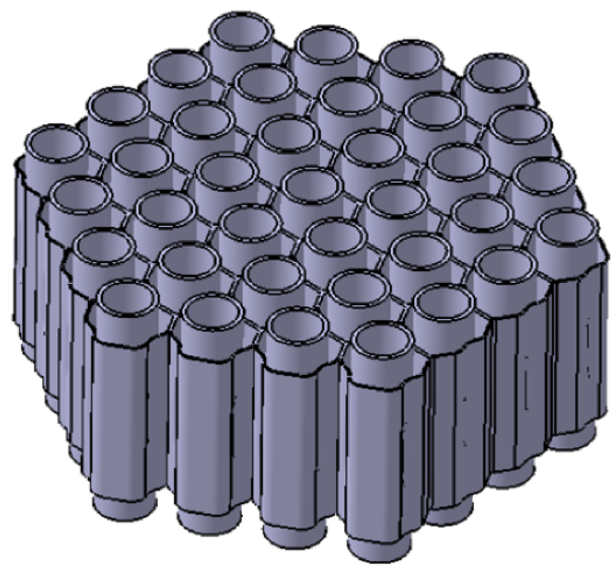

Fig. 3. Spacer grid with fuel rod simulators

layers, respectively.

Internal boundary-based conformal interfaces were used to create a full computation domain [6]. A contact between pins and spacer grids was considered during the simulations.

The overall mesh size of the computational domain is about 232.5 mln. cells; fluid region mesh size is 155.2 mln. cells; the bundle mesh size is 41.8 mln. cells; spacer grid mesh size is 35.5 mln. cells.

Figure 4 shows volume mesh of the experimental facility outlet section. A spacer grid volume mesh and mesh in a spacer mid-section are presented in Figure 5 and Figure 6, respectively.

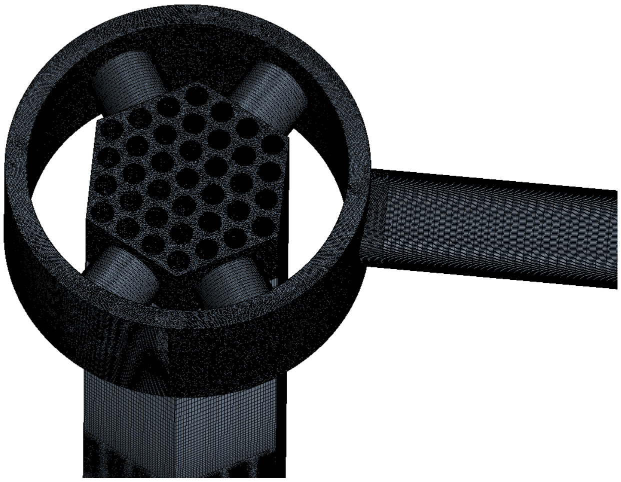

Fig. 4. Volume mesh of outflow experimental facility section

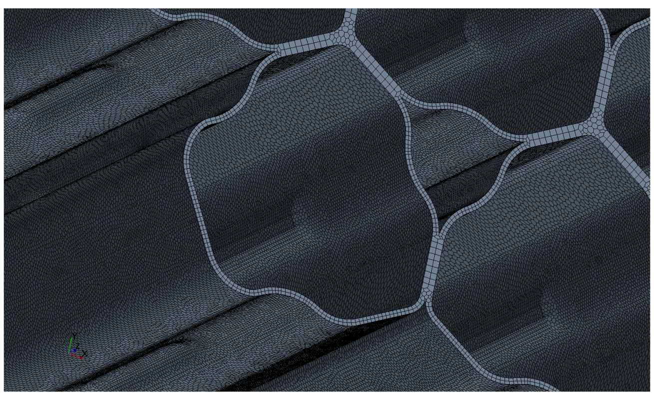

Fig. 5. Spacer grid volume mesh

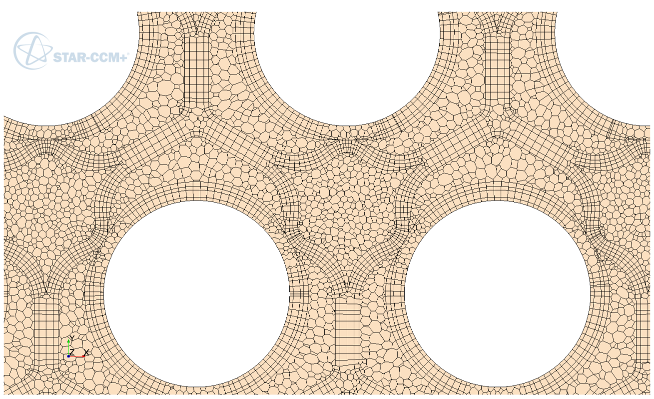

Fig. 6. Mesh in spacer grid mid-section

III. RESULTS

Temperature, velocity, density and pressure distributions in the experimental facility were calculated using the developed CFD model. The steady-state simulation of the discussed 232.5 million cells mesh required 20 h of computation to reach convergence, using 300 cores cluster.

Figure 7 shows temperature distribution in the bundle. Temperature and velocity magnitude distributions at the outlet section are shown in Figure 8 and Figure 9, respectively. Figure 10 shows temperature distribution at the at the measurement section.

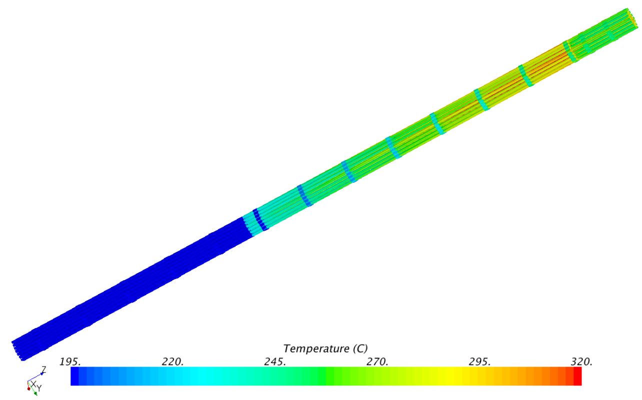

Fig. 7. Temperature distribution in the rod bundle (Z scale=0.5)

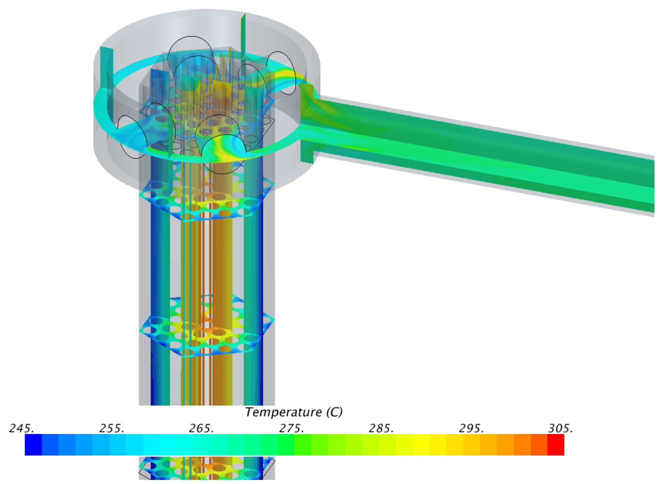

Fig. 8. Temperature at the outflow section

To determine influence of turbulence models simulation with four different turbulence models were selected: κ-ω SST (Mentor), κ-ε realizable and anisotropic quadratic κ-ε [9, 10] model with two sets of constants (standard and formulation presented in [11]). Comparison was performed for subchannel code cells at the heated part outlet. However, different turbulence models showed similar results and the difference for each section was less than 0.5°C.

An influence of thermal contact between simulators and spacer was estimated. Thermal contact was considered as ideal (without thermal resistance). For test case 1 with heat flux 2.323 MW through thermal contact between the fuel rod simulators and spacer grids only 5.144 kW (~0.2%) was conducted. However, the presence of nonideal thermal resistance decreases heat exchange between the bundle and spacer grid. Thus, thermal contact is small and it is not necessary to take it into consideration in further calculations.

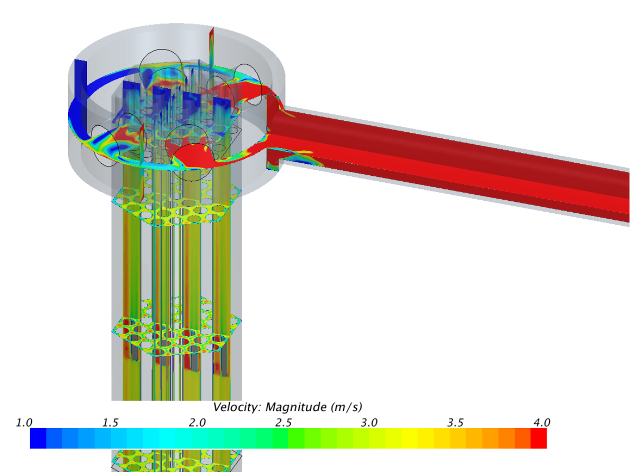

Fig. 9. Velocity magnitude at the outflow section

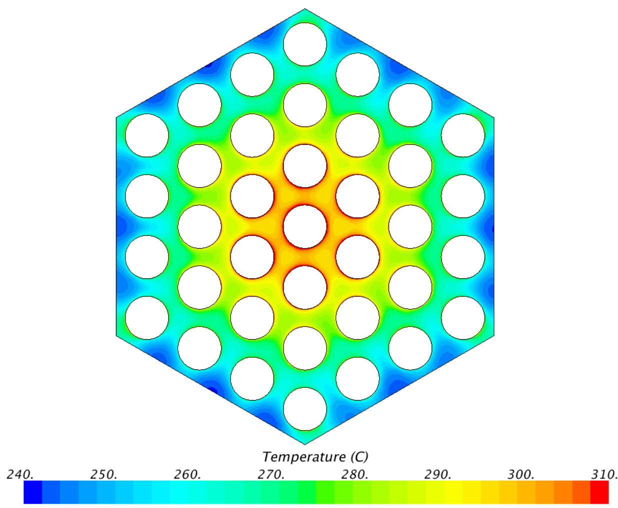

Fig. 10. Temperature at the measurement section

1. Post-processing of the CFD results

To compare CFD results with experimental data and subchannel code results, temperature distribution was integrated by plane sections according with subchannel model [2].

Figure 11 shows CFD mesh with plane sections for integration and probe points. Locations of probes coincide with the centers of thermocouples. Integrated CFD results correspond to subchannel results at the measurement section at the end of the heated part.

For all the test cases CFD results were compared with experimental and subchannel code data. The relative deviation of temperature of calculated CFD data from experimental (or subchannel code data) was determined by the follow equation:

$$\delta = \frac{T_{\exp} - T_{CFD}}{\overline{\Delta T}} \cdot 100\% \tag{1}$$

Mean deviation of calculated CFD results from the experimental data (or subchannel code data) was determined by the follow equation:

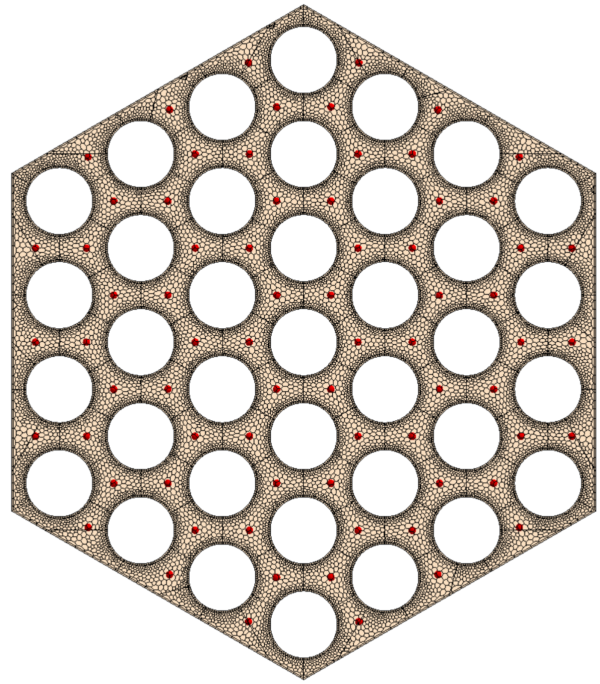

Fig. 11. Location of probe points and plane sections plotted on CFD mesh

$$\delta^{mean} = \frac{\sum_{i=1}^{N} \delta_i}{N} \tag{2}$$

The standard deviation of calculated CFD results from the experimental data (or subchannel code data) was determined by the follow equation:

$$\delta^{st} = \sqrt{\frac{\sum_{i=1}^{N} (\delta_i - \delta^{mean})^2}{N-1}} \tag{3}$$

2. Validation of the CFD results

Figure 12 and Figure 13 show the relative deviation of the CFD results from experimental data and from subchannel code data (SC-INT), respectively. In that Figures show results only for test case 1 by reason of the similar deviations for the rest cases.

A comparison of the CFD results with experimental and subchannel code data is presented in Table II. Mean deviation of calculated CFD results from experimental data do not exceed 7%, standard deviation didn't exceed 6%. A comparison between of CFD and subchannel code results shows that mean and standard deviations do not exceed 4% and 3%, respectively.

IV. CONCLUSIONS

CFD model of the 37 rod bundle experimental facility was developed, a number of CFD calculation was performed. CFD simulation results were obtained and compared against the

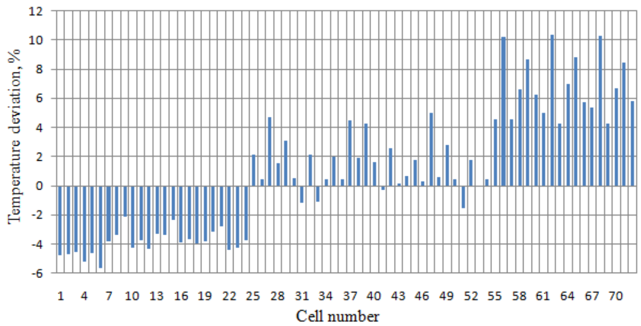

Fig. 12. Relative deviation of the CFD results from experimental data

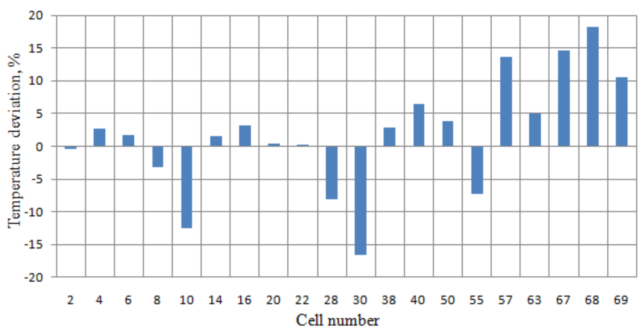

Fig. 13. Relative deviation of the CFD results from SC-INT results

State	Experiment		SC-INT	
	δ^{mean}, %	δ^{st}, %	δ^{mean}, %	δ^{st}, %
1	6.64	5.74	3.71	2.53
2	6.21	5.58	3.54	2.54
3	7.08	4.76	4.04	2.52

TABLE II. Mean and standard deviation of calculated CFD results

experimental and subchannel code data. Standard deviation of calculated CFD results from experimental data and from subchannel code data does not exceed 6% and 3%, respectively.

NOMENCLATURE

T_{exp} – experimental coolant cell temperature, °C;
T_{CFD} – CFD coolant cell temperature, °C;

$\overline{\Delta T}$ – average heat-up, °C;
δ – deviation of calculated CFD data from experimental (or subchannel code) data;
δ^{mean} – mean deviation of calculated CFD results from the experimental (or subchannel code) data;
δ^{st} – standard deviation of calculated CFD results from the experimental (or subchannel code) data.

REFERENCES

1. "VVER-1200 (V-491) (VVER-1200 (V-491))," *IAEA Status report* (2011).
2. L. L. KOBZAR, D. A. OLEKSYUK, and Y. M. SEMCHENKOV, "Experimental and computational investigations of heat and mass transfer of intensifier grids," *KERNTECHNIK*, **80**, *4*, 349 (2015).
3. V. KRIVENTSEV and H. NINOKATA, "Numerical method for simulation of fluid flow and heat transfer in geometrically disturbed rod bundles," *Journal of Nuclear Science and Technology*, **37**, *8*, 646–653 (2000).
4. L. GOLIBRODO, N. STREBNEV, M. KURNOSOV, ET AL., "CFD simulation of turbulent flow structure in a rod bundle array with the split-type spacer grid," *Proceedings of CFD4NRS-4 OECD / NEA&IAEA Workshop*, p. 105 (2010).
5. S. TOTH, "Analysis of coolant mixing in VVER-440 fuel assemblies with the code CFX: Ph.D. thesis / Sandor Toth," *Budapest University of Technology and Economics* (2010).
6. *USER GUIDE STAR-CCM+*, CD-adapco Inc., 10.02 ed. (2015).
7. J. H. FERZIGER and M. PERIC, *Computational Methods for Fluid Dynamics*, Springer, 3rd ed. (2002).
8. R. FERNANDEZ-PRINI and R. DOOLEY, *The International Assosiation for the Properties of Water and Steam. Release on the IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam.*, Erlangen. Germany (1997).
9. D. C. WILCOX, *Turbulence modeling for CFD*, DCW Industries, Inc, 1st ed. (2000).
10. F. MENTER, "Zonal two equation k-turbulence models for aerodynamic flows," *Proc. 24th Fluid Dynamics Conf* (1996).
11. E. BAGLIETTO and H. NINOKATA, "A turbulence model study for simulating flow inside tight lattice rod bundles," *Nuclear Engineering and Design*, , *235*, 773–784 (2005).