IRSTI 61.13.03

D.M. Kenzhebekov[1] – *main author*,
A.E. Khussanov[2], I. Iristaev[3], A. Zholshybek[4], D. Janabayev[5]

*[1,4]PhD student, [2]Candidate of Technical Sciences, Associate Professor,
[3]Engineer of the laboratory, [5]PhD*

ORCID *[1]https://orcid.org/0000-0002-6367-5975 [2]https://orcid.org/0000-0002-1563-6437
[3]https://orcid.org/0009-0008-3255-4602 [4]https://orcid.org/0000-0003-4535-9730
[5]https://orcid.org/0000-0002-6522-0536*

*[1,2,3,4]M. Auezov South Kazakhstan University, Shymkent, Kazakhstan
[5]Shymkent University, Shymkent, Kazakhstan*

@ *[2]khusanov_1975@inbox.ru*

https://doi.org/10.55956/PTUL4611

GENERATION AND CONFIGURATIONOF A HEAT EXCHANGER MODEL GRID FOR CFD MODELING IN COMSOL MULTIPHYSICS

Abstract. The stages of creating a three-dimensional computational domain and generating a grid model for CFD modeling of flow hydrodynamics in a pipe-in-pipe heat exchanger are considered in detail. The sequence of actions for grid generation in COMSOL multiphysics is described. A description is given of the method of thickening the grid by dissecting the volume with a surface and constructing grids with the required interval in each of the resulting connected volumes. A model grid with triangular and square elements was used for the heat exchange model. The construction and adjustment of the model grid plays an important role in multiphysical modeling. The accuracy of the final results depends on the correctness of the model grid setting. During modeling, it is necessary to pay attention to the size of the model grid, the quality and type of the grid.

Keywords: numerical modeling, hydrodynamics, heat transfer, heat exchanger, multiphysical modeling, generation, grid.

Kenzhebekov D.M., Khussanov A.E., Iristaev I., Zholshybek A., Janabayev D. Generation and configurationof a heat exchanger model grid for CFD modeling in comsol multiphysics //Mechanics and Technology / Scientific journal. – 2025. – No.1(87). – P.279-285. https://doi.org/10.55956/PTUL4611

Introduction. The scope of application of heat exchangers is wide and covers technological processes in the oil refining, chemical, refrigeration, gas, energy and other industries. The use of heat exchangers makes it possible to increase the efficiency of the device for which they are used [1].

The rapid development of computer technology and methods for numerically solving problems of heat transfer and hydrodynamics using multiphysical modeling programs has led to the fact that in many fields of science and technology, the results of multiphysical modeling of heat transfer and mass transfer processes become an essential element [2].

The results obtained using modeling allows not only to correctly comprehend and understand the physical effects observed on experimental devices, but also in certain cases a completely natural experiment on computer modeling

[3]. Currently, CFD packages for calculating heat exchange, mass transfer and hydrodynamics are widely used for engineering calculations and scientific research. All CFD packages consist of preprocessors, a solver and a postprocessor [2,3].

CFD modeling *(Computational Fluid Dynamics modeling)* is one of the subsections of continuum mechanics. This package is designed to calculate the characteristics of process flows using physical, mathematical and computational methods. Modeling using CFD packages provides an opportunity to evaluate the temperature and simulation of the flow of liquid and gas in the designed experimental equipment [2,3].

The creation and generation of a grid of the computational domain is an essential component of every engineering calculation where software packages based on CFD technology are used. The size of the calculated model grid directly affects the accuracy of the final results, the speed of calculation and precision. The CFD grid takes into account the following factors: high quality, low computational costs, sufficient resolution to achieve the required precision [2].

The COMSOL multiphysics program uses *Mesh* grid generation to create CFD simulations. The model grid generator provides *Mesh* with the ability to build a computational domain in four ways [4]:

- by its bounding surfaces;
- rotation of the existing surface around the selected axis;
- movement of the surface along the selected line or vector;
- along the edges.

As mentioned above, in the COMSOL multiphysics program, the preprocessor for generating a model grid is a *Mesh*, which allows you to create various types of model grids: automatic (unstructured) hexahedral and tetrahedral grids; structured hexahedral grid.

Materials and methods. When solving any problem, the goal is always to find a solution in some computational domain. As a rule, the size and shape of the computational domain are naturally determined by the problem under study. For example, if the flow of liquid in a certain segment of a hydraulic system is being investigated, then the walls of the system limit the design area only to the inner cavity of the system, where the flow velocity field is being studied. Other tasks, on the contrary, involve searching for solutions in infinite or semi-infinite domains [5]. In this case, a calculated region of finite size is selected, but boundary conditions are set at the boundaries of this region, simulating the solution for (semi)infinite geometry. Geometric modeling of the computational domain is carried out using a set of primitives such as rectangle, ellipse, point, polyline, Bezier curves of 2 and 3 orders in the 2D computational domain. In the 3D domain, there are three-dimensional analogues of these primitives. By combining the use of various primitives, it is possible to construct complex objects with numerous faces and surfaces [6].

The *Mesh* is generated and configured in the model tree after building the geometry of the device and configuring the physics. There are two types of mesh settings: *User Control Mesh* and *Physics Controlled Mesh*.

User Control Mesh is used to manually adjust the mesh. This setting allows you to adjust the size of each part individually. It is usually used in complex geometries and in areas of elevated gradients. The time it takes to set up is much higher than that of a physics-controlled grid.

By default, the program uses *Physics Controlled Mesh*. The grid is generated and adjusted automatically. With this setting, the shape and size of the grid elements is adjusted by the program and depends on the selected physics and input parameters.

When modeling the experimental apparatus, the *Physics Controlled Mesh* setting was applied. After generating the grid, the Information graph appears in the model tree, where a more fine-tuning of the grid is performed.

The next step in setting up the model grid is to select the areas of the experimental apparatus. There are two ways to select the areas to generate the boundary layer: *All geometry* or *Domen*. With *All geometry*, the geometry of the device is automatically selected. With *Domen*, the device areas are manually selected. *Domen* allows you to fine-tune the model grid of the boundary layer of a specific selected area.

Boundary Layer Mesh (border layer grid) is a grid with a dense distribution of elements in the normal direction along certain boundaries. This type of grid is commonly used to solve fluid flow problems in order to resolve thin boundary layers along boundaries without sliding. In 3D modeling, the boundary layer grid is a multilayer prismatic grid or a hexagonal grid, depending on whether the corresponding boundaries of the boundary layer contain a triangular or quadrangular grid [6].

Research results and discussion. The most important component of the *Mesh* model grid is the selection and adjustment of the size of the *Size* model grid. The accuracy of the final results depends on the setting of this item. By default, the program uses the size of the model grid – *Normal*, but to obtain precise results, the grid sizes are usually used: *Fine* and *Finer*. During the simulation of the heat exchanger, the grid size was applied: *Extra coarse*, which is shown in Figure 1 [7].

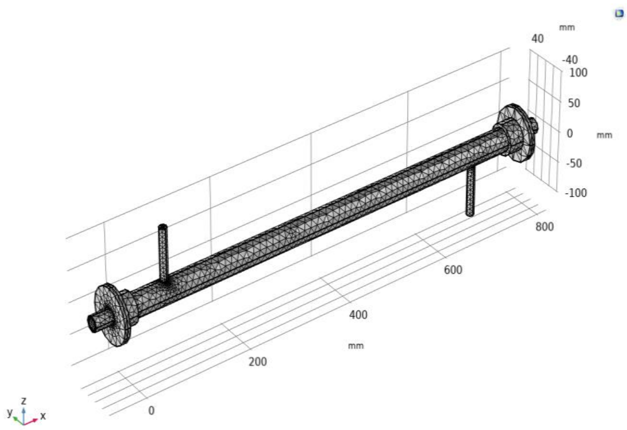

Fig. 1. Model grid Mesh of the heat exchanger

As can be seen from Figure 1, the dimensions of the model grid elements in different areas of the device differ in size and shape. The largest elements are located in the body of the device.

For devices with complex geometries that contain various nozzles, small parts, etc., it is necessary to configure the model grid separately for each *Domen*.

The thinner the model grid, the more resources and time the program requires for calculation. By default, COMSOL multiphysics uses two types of model grid construction, a triangular and a tetrahedral grid. Each type of mesh can be configured manually, following the sequence of *Mesh – Mesh parameters* [7].

A model grid with triangular and square elements was used for the heat exchange model, the main characteristics of the model grid are shown in Table 1.

Table 1

Characteristics of the model grid of the simulated device

Description	Meaning
Status	Full grid
Grid Vertex	58963
Tetrahedra	196229
Pyramids	11498
Prisms	41754
Triangles	41458
Edge elements	6993
Vertex Elements	952
Number of elements	249481
Minimum element quality	0.005304
Average quality of the element	0.5423
The ratio of the volumes of the elements	4.1519E-8
The volume of the model grid	1.087E6 мм³
Type of grid	Large (Very Rough)

The next factor affecting the model grid is the quality of the grid. The quality of the model grid affects the accuracy of the final results, the speed of calculation and the efficiency of numerical modeling. There are the following types of grid quality: low, medium and high quality. Several factors affect the decrease in the quality of the mesh: strongly curved surfaces, thin lines, small details, small edges and areas. You can track the quality of the final model grid using the quality histogram in the *Status* section.

The state of the model grid can be monitored on the tab *Mesh – Status*. This tab contains the main parameters of the final model grid: area, number of elements, type of elements, and quality of elements. The grid status window shows a histogram of the model grid.

In the areas of increased gradients (velocity, pressure and temperature) of the connection of the fittings, the support grid, as well as in the corners of the device body, the program automatically thickens the grid, Figure 2. The thickening of the grid is formed in places of boundary conditions, areas of separation (boundary layers) of one element from another [2,3].

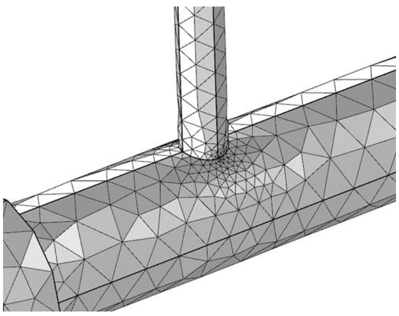

Fig. 2. Thickening of the model at the junction of the tube with the body

In the boundary areas, COMSOL makes a separate adjustment of the model grid, the characteristics of the grid are shown in Table 2.

Table 2

Characteristics of the model grid of the simulated device

Description	Meaning
Calibrated for	Fluid dynamics
Maximum size of the elements	43.7
Minimum size of the elements	9.27
The coefficient of curvature	0.3
Growth rate of elements	1.4
Grid Size	Large (Rough)

Figure 3 shows a diagram of the model grid of the inlet sections of the nozzles of the device, the grid setting area is highlighted in blue. In this area, the size of the grid elements is always smaller than in other parts of the model, not counting angular thickenings.

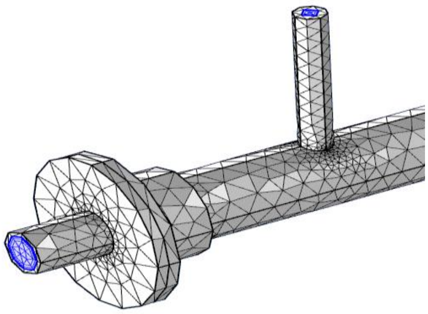

Fig. 3. A model grid of the inlet sections of the nozzles of the device

The state of the model grid can be monitored on the tab *Mesh – Status*. This tab contains the main parameters of the final model grid: area, number of elements, type of elements, and quality of elements. The grid status window shows a histogram of the model grid.

Conclusion. Thus, in this paper, the actions in the COMSOL multiphysics software package for creating the geometry of the computational domain and the grid model of the “pipe-in-pipe” heat exchanger are described.

After constructing the geometry of the calculation, it is required to generate a finite element grid that divides the computational domain into many small finite elements describing the solution of the problem at selected points, called nodes. This operation is often defined in the literature as a discretization operation, when there is a transition from finding a solution in a continuously changing domain of independent variables to solving a problem only in a selected set of points – node. During visualization, the computational grid "covers" the calculated area unevenly, forming a thickening of corners near the boundaries or corners where the solution changes faster than it does away from the boundaries. Generating an efficient grid is a great art, but the algorithms used in COMSOL are quite universal and give a good result for the first approximation.

The construction and adjustment of the model grid plays an important role in multiphysical modeling. The accuracy of the final results depends on the correctness of the model grid setting. During modeling, it is necessary to pay attention to the size of the model grid, the quality and type of the grid.

References

1. Pulin A., Laptev, M., Kortikov N., Barskov V., Roschenko G., Alisov K., Talabira I., Gong B., Rassokhin V., Popovich A., et al. Numerical Investigation of Heat Transfer Intensification Using Lattice Structures in Heat Exchangers // Energies. – 2024. – No. 17. – P. 3333.
2. Mukhametzyanov A.G., Soskov V.N., Alekseev K.A., Dolgova N.V. Sozdaniye trekhmernoy raschetnoy oblasti i generatsiya setki dlya CFD – modelirovaniya gidrodinamiki potoka v staticheskikh smesitelyakh Kenics KM. Chast' 1. [Creation of a three-dimensional computational domain and grid generation for CFD modeling of flow hydrodynamics in Kenics KM static mixers. Part 1.] // Bulletin of the Technological University. – 2017. – Vol. 20. – No. 7. – P. 93-95. [in Russian].
3. Mukhametzyanov A.G., Soskov V.N., Alekseev K.A., Dolgova N.V. Sozdaniye trekhmernoy raschetnoy oblasti i generatsiya setki dlya CFD – modelirovaniya gidrodinamiki potoka v staticheskikh smesitelyakh Kenics KM. Chast' 2. [Creation of a three–dimensional computational domain and grid generation for CFD modeling of flow hydrodynamics in Kenics KM static mixers. Part 2.] // Bulletin of the Technological University. – 2017. – Vol. 20. – No. 7. – P. 101-102. [in Russian].
4. Egorov, V.I. Primeneniye EVM dlya resheniya zadach teploprovodnosti. Uchebnoye posobiye [The use of computers for solving problems of thermal conductivity. A study guide]. – St. Petersburg State University of ITMO, 2006. – 77 p. [in Russian].
5. Tsvetova, E.V. Chislennoye modelirovaniye [Numerical modeling]: textbook. – Ulyanovsk: UlSTU, 2022. – 49 p.
6. Abiev, R.S. Vychislitel'naya gidrodinamika i teplomassoobmen. Vvedeniye v metod konechnykh raznostey [Computational hydrodynamics and heat and mass transfer. Introduction to the finite difference method]: textbook. – St. Petersburg: Publishing House of the St. Petersburg State University Institute of Chemistry, 2002. – 576 p. [in Russian].
7. COMSOL Multiphysics User’s Guid. – COMSOL 6.1. – 2012, – 1292 p.

Received: 12 October 2024
Accepted: 7 March 2025

Д.М. Кенжебеков[1], А.Е. Хусанов[1],
И. Иристаев[1], А. Жолшыбек[1], Д.Ж. Джанабаев[2]

[1]М. Әуезов атындағы Оңтүстік Қазақстан университеті, Шымкент қ., Қазақстан
[2]Шымкент университеті, Шымкент қ., Қазақстан

ЖЫЛУАЛМАСТЫРҒЫШ АППАРАТТЫҢ COMSOL MULTIPHYSICS-ТЕ CFD МОДЕЛЬДЕУ ҮШІН МОДЕЛЬДІК ТОРЫН ҚҰРУ ЖӘНЕ КОНФИГУРАЦИЯЛАУ

Аңдатпа. «Құбыр ішіндегі құбыр» жылу алмастырғышындағы ағынның гидродинамикасын CFD модельдеу үшін үш өлшемді есептеу аймағын құру және тор моделін құру кезеңдері егжей-тегжейлі қарастырылды. COMSOL Multiphysics-те тор құру кезіндегі әрекеттер тізбегі көрсетілген. Беттің көлемін бөлу арқылы торды қалыңдату әдісінің және алынған торлардың әрқайсысында қажетті интервал мөлшерімен құрылыстың сипаттамасы берілген. Жылу алмастырғыш моделі үшін үшбұрышты және шаршы элементтері бар модельдік тор пайдаланылды. Модельдік торды құру және реттеу мультифизикалық модельдеуде маңызды рөл атқарады. Соңғы нәтижелердің дәлдігі модель торының дұрыс орнатылуына байланысты.

Модельдеу кезінде модель торының өлшеміне, тордың сапасына және түріне назар аудару қажет.

Тірек сөздер: сандық модельдеу, гидродинамика, жылу алмасу, жылу алмастырғыш, мультифизикалық модельдеу, генерация, тор.

Д.М. Кенжебеков[1], А.Е. Хусанов[1]
И. Иристаев[1], А. Жолшыбек[1], Д.Ж. Джанабаев[2]

[1]Южно-Казахстанский университет им. М. Ауэзова, г. Шымкент, Казахстан
[2]Шымкентский университет, г. Шымкент, Казахстан

ГЕНЕРАЦИЯ И НАСТРОЙКА МОДЕЛЬНОЙ СЕТКИ ТЕПЛООБМЕННОГО АППАРАТА ДЛЯ CFD-МОДЕЛИРОВАНИЯ В COMSOL MULTIPHYSICS

Аннотация. Подробно рассмотрены этапы создания трехмерной расчетной области и генерация сеточной модели для CFD-моделирования гидродинамики потока в теплообменном аппарате «труба в трубе». Изложена последовательность действий при генерации сетки в COMSOL Multiphysics. Дано описание метода сгущения сетки путем рассечения объема поверхностью и построения в каждом из получившихся связанных объёмов сеток с необходимой величиной интервала. Для модели теплообменика было использована модельная сетка с треугольными и квадратными элементами. Построение и настройка модельной сетки играет не маловажную роль в мультифизическом моделировании. От корректности настройки модельной сетки зависит точность конечных результатов. В ходе моделирования необходимо обращать внимание на размер модельной сетки, качеству и типу сетки.

Ключевые слова: численное моделирование, гидродинамика, теплообмен, теплообменник, мультифизическое моделирование, генерация, сетка.