CONSTRUCTION MESH IN ANSYS MESHING MODELS FOR CFD FINITE ELEMENTS METHOD

V.I. Trokhanyak

In numerical calculation of heat transfer and fluid dynamics problems using finite element method (FEM). The method consists in the approximate solution of variational problem. To formulate this problem using the concept of functional. The operator I [f (x)] is called functional, who asked on a set of functions, if for each function f (x) is assigned a specific numerical value I [f (x)] [2]. In other words, the functionality is like a "function of a function". Often functionals have the form of integrals. The variation problem is to find the function f (x), which would meet the minimum value of the functional I [f (x)]. View this functional is different for different tasks, which is chosen by a special selection.

Today FEM is widely used in solving problems of heat conduction in solids and in the calculations for strength. However, it can also be applied when calculating flows of liquids and gases. [1] There are also methods that combine elements of finite volume method and finite element method [3, 5]. The combination of these techniques allows a wider range of calculation grids (tetragonal meshes, pyramids, prisms, polyhedra) that is necessary for solving problems with complex geometry. This approach is used CFD packages Ansys CFX, Ansys Fluent, Star-CD, Star-CCM +, Comsol and others.

There are many methods to assess the quality of the grid. The main criteria of quality elements (cells) for Fluent is Orthogonal Quality and Skewness, which are listed in the table. 1. We recommend using mesh models for which the minimum Orthogonal Quality >0.1 or maximum value Skewness «bias» <0.95 [4]. "Skewed" - a measure of change element compared to its original form, which ranges from 0 (excellent) to 1 (invalid). Elements of poor quality can lead to inaccurate results, or in some cases a decision may not come together!

The purpose of research - the creation of 2D FEM mesh so that the region to analyze the flow tubes in appropriate locations and surfaces (temperature, thermal and hydrodynamic flow, pressure), the error in this case should not exceed 5%, with the desired accuracy of ~ 1%.

In ANSYS Meshing Platform for 2D geometry are four methods that can be applied to the surface of bodies or shells:

- Auto (Automatic Method) (Quadrilateral Dominant);
- All triangular elements (All Triangles);
- Uniform distribution of rectangular and triangular elements;
- Uniform distribution of rectangular elements.

Results. When building a grid for heat exchangers used local control grid. Created continuous layers in the boundary layer at a different number of layers to get the full thickness, that is, the thickness of the first layer (2 • 10-4 m), and for each subsequent layer thickness chosen constant.

Traditional corridor location of the beam pipe.

Fig. 2 that used hybrid network: in the area around the surface of the tubes used rectangular and triangular mesh netting in all different area. Square mesh provides greater accuracy viscosity gradients near the surface of the tubes.

In constructing hybrid mesh was found that using the method of construction of boundary layer total thickness (TT), the thickness of the first layer $2 \cdot 10^{-4}$ m at an amount of 3 to 12 layers of mesh quality two parameters remained unchanged and within Orthogonal Quality 0,68, Skewness 0,49. Simulating the hydrodynamics of flow tubes, the choice of number of layers is important to analyze vouchers zones. Therefore, it is recommended to set the maximum number of layers to 12 layers (Fig. 1). Such construction requires to use a large number of items, compared with a square grid.

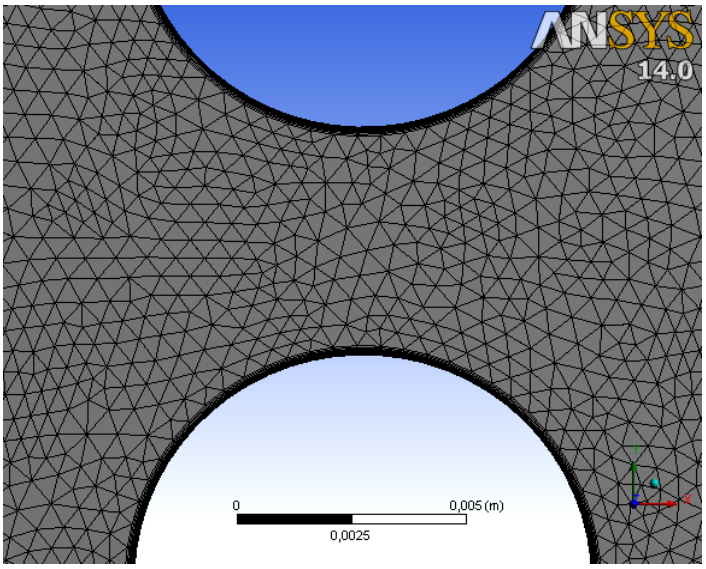

Fig. 1. Hybrid mesh, boundary layer built by TT of 12 layers in the original thickness of $2 \cdot 10^{-4}$

Findings

1. A construction algorithm and analyzes 2D FEM mesh in ANSYS Meshing for traditional corridor arrangement beam pipes.
2. Features of grid construction tasks for hydro gas dynamics and thermal mass transfer at cross flow around bundles of pipes.
3. As a result, the developed approach and selected the optimal quality meshes for CFD models, allowing to obtain reliable and accurate results of calculation of heat exchangers for various purposes.

References

1. Власова Е.А., Зарубин В.С., Кувыркин Г.Н. Приближённые методы математической физики, Е.А. Власова. М.: Изд-во МГТУ им. Н.Э. Баумана, 2001. − 700 с.
2. Г.Н. Дульнев. и др. Применение ЭВМ для решения задач теплообмена − М: Высш. шк., 1990. − 207 с.
3. Патанкар С. Численные методы решения задач теплообмена и динамики жидкости / С. Патанкар; пер. с англ. – М.: Энергоатомиздат, 1984. – 152 с.
4. ANSYS Meshing User's Guide. / ANSYS Inc.– ANSYS Meshing Release 15.– 2013.
5. J.H. Ferziger, M. Perić. Computational Methods for Fluid Dynamics 3., rev. ed. - Berlin; Heidelberg; New York; Barcelona; Hong Kong; London; Milan; Paris; Tokyo: Springer, 2002.