J. Szantyr – Lecture No. 12 – Computations of Viscous Flows – The Finite Volume Method

The Finite Volume Method is based on transformation of the partial differential equations into their algebraic equivalents through their integration within the limits of each finite volume on the basis of the assumed approximation of the variables describing flow parameters inside the volume (e.g. linear, quadratic etc.)

A simple example: general steady transport equation by diffusion and convection.

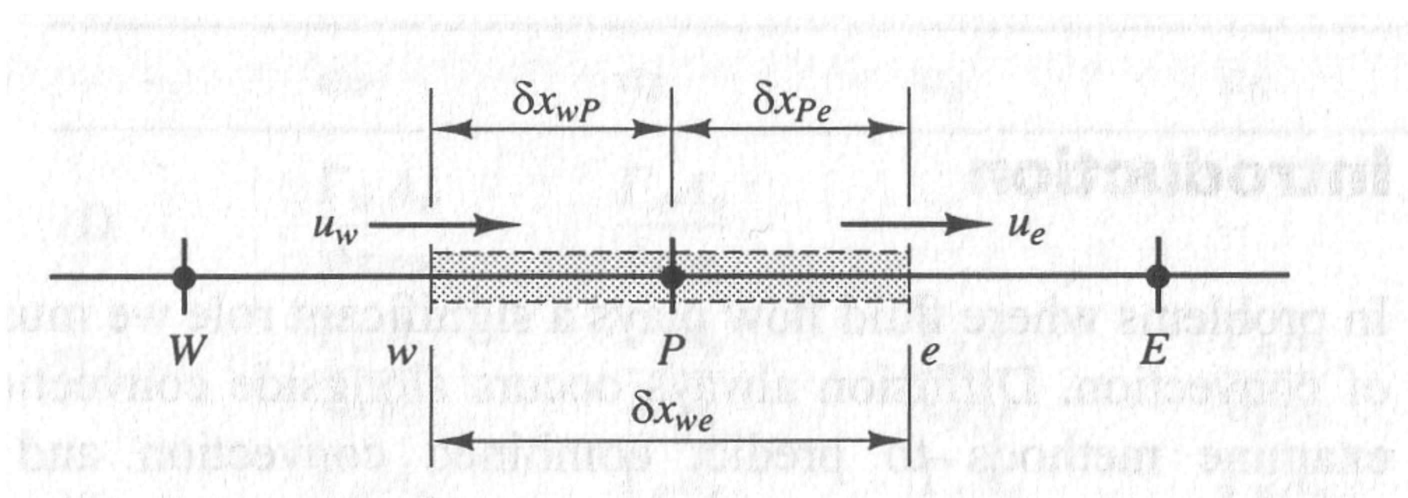
$$div\left(\rho\,\bar{u}\,\varphi\right) = div\left(\Gamma\, grad\,\varphi\right) + S_{\varphi}$$

In the one-dimensional case we have:

Transport equation: $$\frac{d}{dx}\left(\rho u \varphi\right)=\frac{d}{dx}\left(\Gamma\frac{d\varphi}{dx}\right)$$

Mass conservation equation: $$\frac{d\left(\rho u\right)}{dx}=0$$

We assume that the velocity u is known.



Integration of the equations within the finite volume leads to:

$$\left(\rho uA\varphi\right)_e - \left(\rho uA\varphi\right)_w = \left(\Gamma A\frac{\partial \varphi}{\partial x}\right)_e - \left(\Gamma A\frac{\partial \varphi}{\partial x}\right)_w$$

$$\left(\rho uA\right)_e - \left(\rho uA\right)_w = 0$$

If we introduce the following notation:

convection coefficient $F = \rho u$ $F_e = \rho u_e$

diffusion coefficient $D = \dfrac{\Gamma}{\delta x}$ $D_e = \dfrac{\Gamma_e}{\delta x_{PE}}$

then the equations may be written in the form:

$$F_e\phi_e - F_w\phi_w = D_e\left(\phi_E - \phi_P\right) - D_w\left(\phi_P - \phi_W\right)$$

$$F_e - F_w = 0$$

Coefficients of the above equation may be determined e.g. on the basis of the linear central difference scheme:

$$\phi_e = \left(\phi_P + \phi_E\right)/2 \qquad \phi_w = \left(\phi_W + \phi_P\right)/2$$

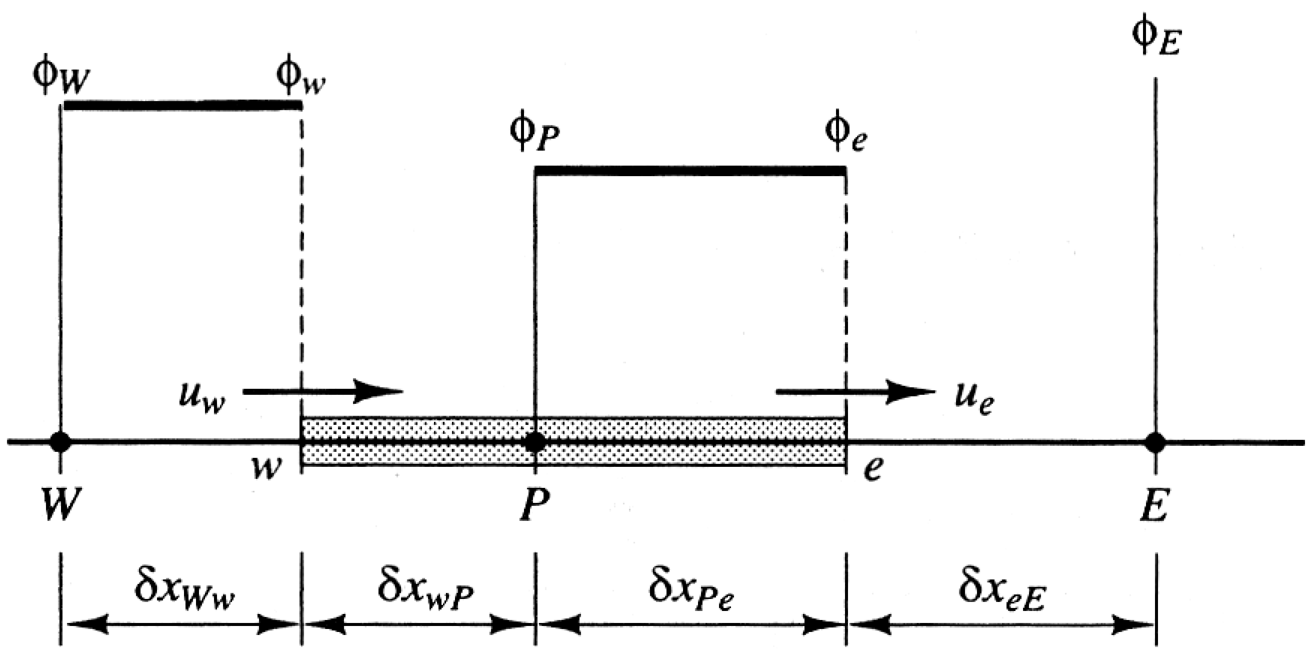
Substitution of the above into the differential transport equation leads to the equivalent algebraic equation, i.e. the interpolation formula determining the value at point P on the basis of the values in the neighbouring points W and E:

$$\left[\left(D_w - \frac{F_w}{2}\right) + \left(D_e + \frac{F_e}{2}\right)\right]\varphi_P = \left(D_w + \frac{F_w}{2}\right)\varphi_W + \left(D_e - \frac{F_e}{2}\right)\varphi_E$$

The central difference interpolation scheme works well when the intensities of convection and diffusion in the transport process are of similar order. In the case of flows dominated by convection, the so called upwind scheme produces better results.

The simplest upwind scheme is based on the assumption that the transported quantity is transferred downstream through convection, unchanged, by the distance of half the finite volume length, i.e.:

$$\phi_w = \phi_W \qquad \phi_e = \phi_P$$



Application of the upwind scheme leads to the following form of the algebraic transport equation (i.e. interpolation formula):

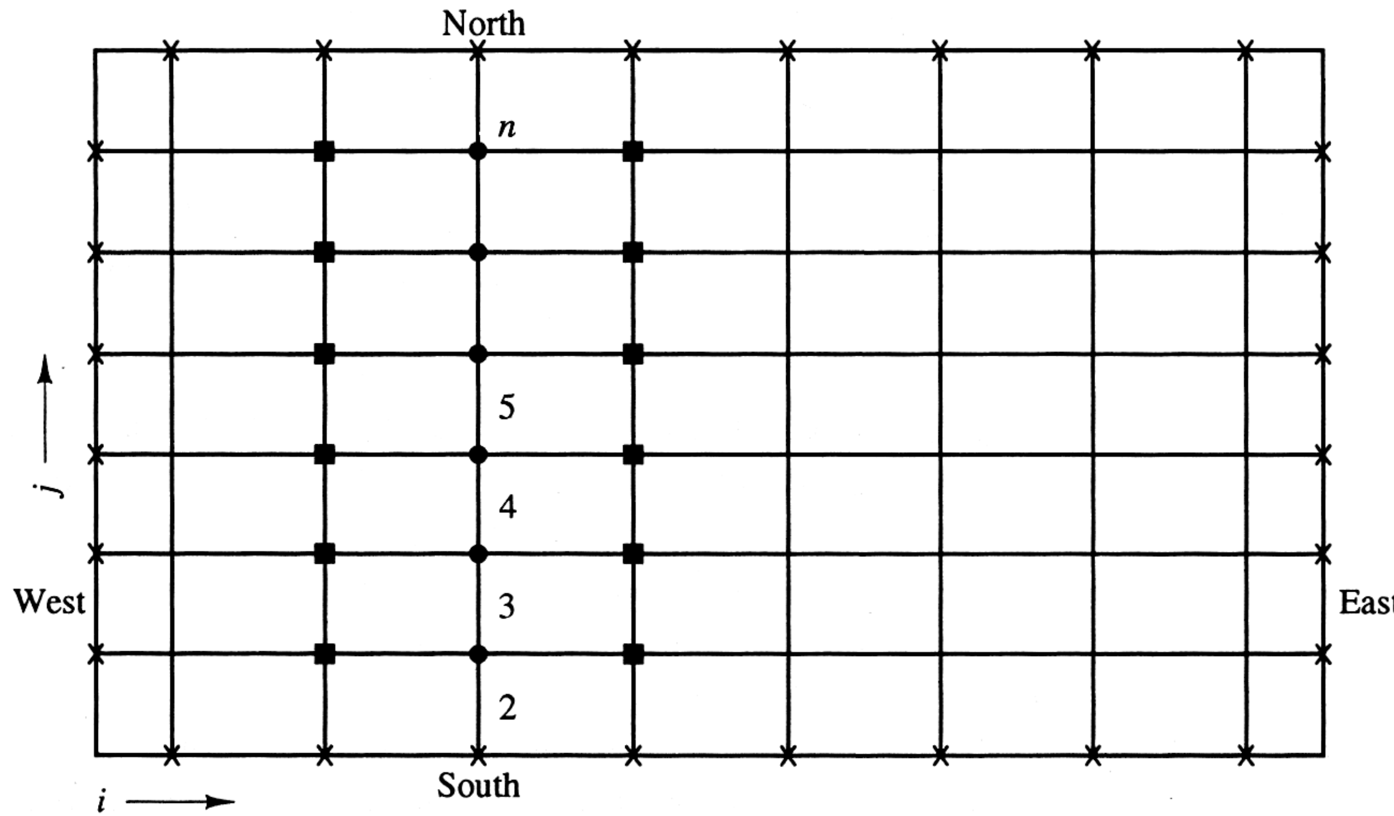
$$\left[\left(D_w + F_w\right) + D_e + \left(F_e - F_w\right)\right]\phi_P = \left(D_w + F_w\right)\phi_W + D_e\phi_E$$

The upwind interpolation scheme produces stable solutions of the transport equation dominated by convection, but in 2D and 3D applications it generates artificial (non-physical) „numerical" diffusion, particularly in cases when the direction of convection coincides with diagonals of the finite volumes. Elimination of this numerical diffusion requires a large numbers of small finite volumes, thus increasing the size of the computational problem.

When the algebraic transport equation is applied to all finite volumes in the above one-dimensional case, it leads to the system of linear equations, solution of which supplies the values of the transported quantity in the central points of all volumes.

In two- or three-dimensional cases it is necessary to apply an iterative procedure, which is explained in the following figures. Obtaining the converged solution requires multiple repetitions of the iterative proces in the entire flow domain. In the two-dimensional case the interpolation formula determines the required value on the basis of four neighbouring points (W, E, N, S), and in the three-dimensional case – on the basis of six neighbouring points (W, E, N, S, B, T).

Iterative procedure for solution of the system of algebraic equations for the two-dimensional case.

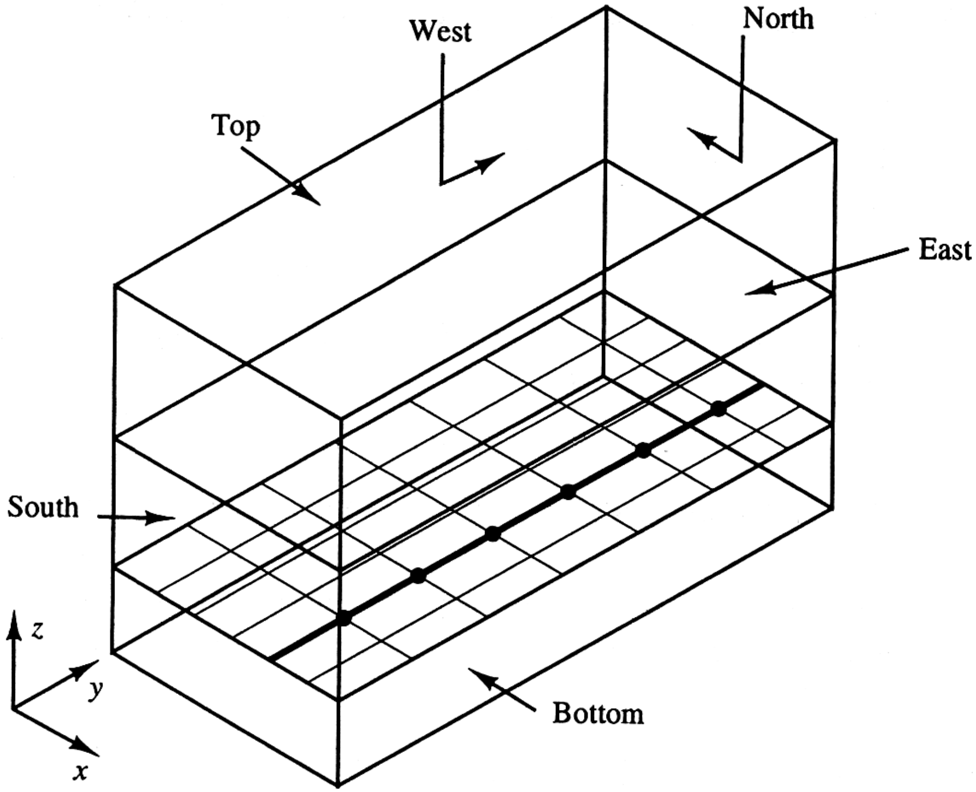


● Points at which values are calculated

■ Points at which values are considered to be temporarily known

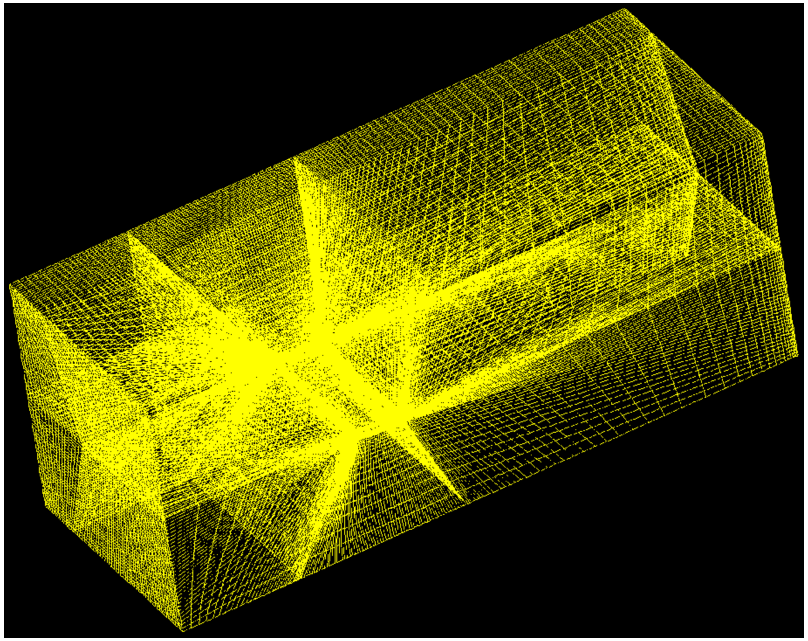
x Known boundary values

Iterative procedure for solution of the system of algebraic equations for the three-dimensional case.

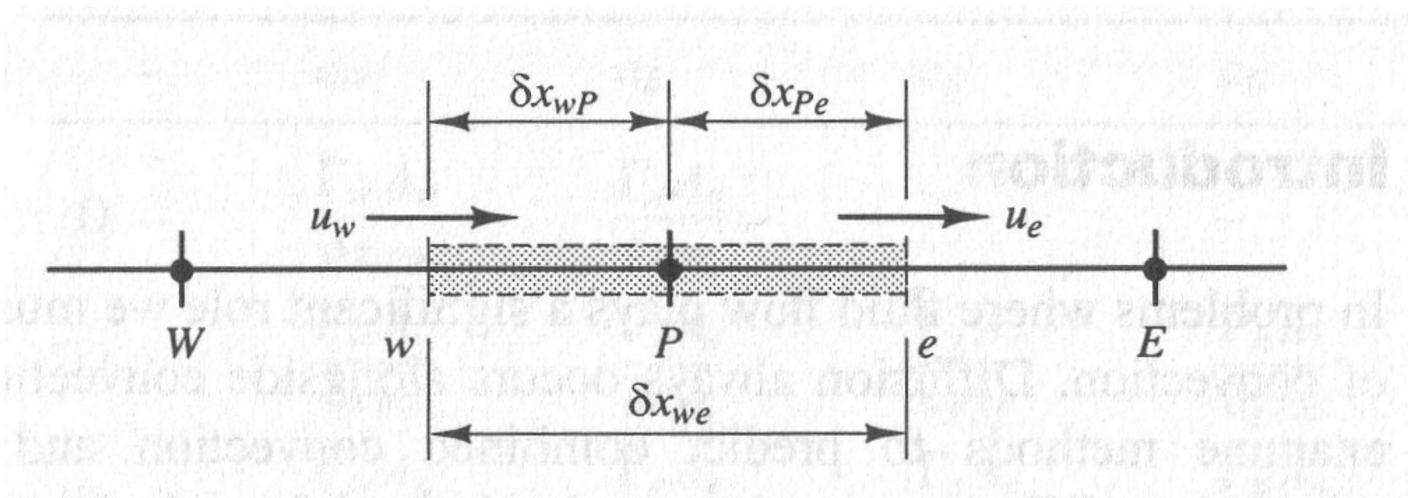


← Iterative scheme

Example of the real finite volume grid →



Unsteady flows based on example of one-dimensional heat diffusion described by the temperature field T in a rod (ue=uw=0)



Basic equation:

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right)$$

After integration within the finite volumes (CV):

$$\int_{t}^{t+\Delta t} \int_{CV} \rho c \frac{\partial T}{\partial t}\, dV dt = \int_{t}^{t+\Delta t} \int_{CV} \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right) dV dt$$

The equation may be now written in the following form:

$$\int_{w}^{e}\left[\int_{t}^{t+\Delta t}\rho c\frac{\partial T}{\partial t}dt\right]dV=\int_{t}^{t+\Delta t}\left[\left(kA\frac{\partial T}{\partial x}\right)_{e}-\left(kA\frac{\partial T}{\partial x}\right)_{w}\right]$$

If the temperature in node P is taken as representative for the entire volume, then it may be written:

$$\rho c\left(T_{P}-T_{P}^{0}\right)\Delta V=\int_{t}^{t+\Delta t}\left[\left(k_{e}A\frac{T_{E}-T_{P}}{\delta x_{PE}}\right)-\left(k_{w}A\frac{T_{P}-T_{W}}{\delta x_{WP}}\right)\right]$$

where the upper index 0 by T denotes the initial value of the time step, and T without index 0– the final value (to be determined)

In order to discretize the right hand side it is necessary to adopt an assumption defining the character of temperature variation with time, e.g. in the form of a weight function:

$$\int_t^{t+\Delta t} T_P dt = \left[\vartheta T_P + (1-\vartheta) T_P^0\right]$$

This allows writing of the diffusion equation in the form of an algebraic interpolation formula:

$$\left[\rho c \frac{\Delta x}{\Delta t} + \vartheta\left(\frac{k_e}{\delta x_{PE}} + \frac{k_w}{\delta x_{WP}}\right)\right] T_P = \frac{k_e}{\delta x_{PE}}\left[\vartheta T_E + (1-\vartheta) T_E^0\right]$$

$$+\frac{k_w}{\delta x_{WP}}\left[\vartheta T_W + (1-\vartheta) T_W^0\right] + \left[\rho c \frac{\Delta x}{\Delta t} - (1-\vartheta)\frac{k_e}{\delta x_{PE}} - (1-\vartheta)\frac{k_w}{\delta x_{WP}}\right] T_P^0$$

Depending on the value of the weight parameter we can obtain different interpolation schemes, which require different relations between the time step and the step in space (i.e. the size of finite volumes) in order to ensure numerical stability:

$\vartheta = 0$ **Explicit scheme, stability condition:**

$$\Delta t \leq \rho c \frac{\left(\Delta x\right)^2}{2\mathrm{k}}$$

$\vartheta = 0{,}5$ **Crank-Nicholson scheme, stability condition**:

$$\Delta t \leq \rho c \frac{\left(\Delta x\right)^2}{k}$$

$\vartheta = 1$ **Implicit scheme, unconditionally stable**

Momentum conservation equation (Navier-Stokes equation) in the differential form describes both the laminar and turbulent flows. Retaining of such universality of its algebraic equivalents is possible in three ways:

-through direct numerical simulation of turbulence down to the smallest vortex scales, i.e. down to Kolmogorov scale (this is called DNS –Direct Numerical Simulation),

-through division of the turbulent scales into the part simulated numerically (large vortices) and part modelled by the appropriate special equations (small, sub-grid vortices) (this is called LES – Large Eddy Simulation or DES – Detached Eddy Simulation),

-through modelling of all turbulence scales by means of special equations (this is called RANSE or Reynolds Averaged Navier Stokes Equations).

The Reynolds equations have the form:

$$\frac{\partial(\rho U)}{\partial t}+div(\rho U\,\bar{U})=-\frac{\partial P}{\partial x}+div(\mu\,gradU)+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}+S_x$$

$$\frac{\partial(\rho V)}{\partial t}+div(\rho V\,\bar{U})=-\frac{\partial P}{\partial y}+div(\mu\,gradV)+\frac{\partial \tau_{yx}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}+S_y$$

$$\frac{\partial(\rho W)}{\partial t}+div(\rho W\,\bar{U})=-\frac{\partial P}{\partial z}+div(\mu\,gradW)+\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{zy}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+S_z$$

Closing of the Reynolds equations requires application of a turbulence model. The most frequently used is the two-equation model k-ε, where k is the kinetic energy of turbulence and ε – is the rate of dissipation of this energy. This model is based on the following relations:

$$k=\frac{1}{2}\left(u'^2+v'^2+w'^2\right) \qquad \mu_t=\rho C_\mu \frac{k^2}{\varepsilon} \qquad \tau_{ij}=-\rho u_i' u_j'=\mu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)$$

$$\frac{\partial(\rho k)}{\partial t}+div\left(\rho k\bar{U}\right)=div\left[\frac{\mu_t}{\sigma_k}gradk\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon$$

$$\rho\varepsilon \{ \bar{U}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div\left(¿\right)=div\left[\frac{\mu_t}{\sigma_\varepsilon}grad\,\varepsilon\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

$$E_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \qquad C_\mu=0{,}09 \qquad \sigma_\varepsilon=1{,}30$$

$$\sigma_k=1{,}0 \qquad C_{1\varepsilon}=1{,}44 \qquad C_{2\varepsilon}=1{,}92$$

Closing of the Reynolds equation by means of a turbulence model requires additional boundary conditions. In the case of k-ε model the following conditions are necessary:

- at inlet – given distributions of k and ε

- at outlet $\partial k / \partial n = 0$ and $\partial \varepsilon / \partial n = 0$

- at a free boundary k=0 and ε=0

- on a rigid wall the approach depends on the Reynolds number value – for high values the so called law of the wall is applied, avoiding integration of the equation to the very edge of the flow domain, - for low Reynolds numbers another form of the model equations is used, based on the assumption that the flow near the wall is dominated by the viscous stresses.