Pressure correction equation for CFD turbulence models

In computational fluid dynamics (CFD), turbulence models are used to simulate turbulent flows, which are characterized by chaotic fluctuations in velocity, pressure, and other flow properties. One common approach in turbulence modeling is the use of Reynolds-averaged Navier-Stokes (RANS) equations, which involve averaging the flow variables over time to account for the turbulent fluctuations.

In RANS simulations, the pressure field is typically decomposed into a mean component and a fluctuating component. However, solving the RANS equations directly doesn't always provide accurate results, especially in regions of strong turbulence where the pressure field may not be well resolved.

To address this issue, pressure correction methods are often employed to correct the computed pressure field and improve the accuracy of the simulation. One such method is the pressure correction equation, which is used to adjust the pressure field iteratively until it satisfies certain criteria.

The pressure correction equation typically takes the form of a Poisson equation, which relates the divergence of the corrected velocity field to the corrected pressure field. It can be written as:

$$\nabla^2 p' = -\nabla \cdot (\rho \overline{u'u'})$$

where:

- p' is the pressure correction,
- ρ is the fluid density,
- $\overline{u'}$ represents the resolved fluctuating velo components.

This equation essentially states that the Laplacian of the pressure correction is equal to the negative divergence of the Reynolds stress tensor, which is a measure of the momentum flux due to turbulent fluctuations.

The pressure correction equation is typically solved iteratively within each time step of the simulation until the corrected pressure field satisfies certain convergence criteria. Once the pressure field is corrected, it is used to update the velocity field, and the process is repeated until a solution is obtained.

Overall, the pressure correction equation is a key component of turbulence modeling in CFD, helping to improve the accuracy of simulations by better accounting for the effects of turbulence on the flow field.