SIMPLIFIED METHOD ON MATHEMATICAL MODEL OF TRANSONIC AXIAL COMPRESSORS

Árpád VERESS Ph. D. student
BUDAPEST UNIVERSITY OF TECHNOLOGY AND ECONOMICS
Department of Aircraft and Ships

Keywords: *CFD, turbomachinary, C, C++, finite volume method and transonic compressor characteristics.*

Abstract

Performance goals for gas turbine engines continue to increase to meet the needs of both military and commercial aviation.

In the compressor system, each stage must produce a significant total pressure rise with high efficiency.

According to research there are many widespread computational method so far to solve the Navier-Stokes, Euler equation or other interaction in 3D. It is obvious this kind of procedures something like that need very significant computational time and background.

The paper tries to show a simplified fluid dynamics method, which would be able to describe airflow over sound speed correctly using for PC.

1 The Governing Equations

The general technique for obtaining the equations governing fluid motion is to consider a small control volume through which the fluid moves, and to require that mass and energy are conserved, and that the rate of change of the three components of linear momentum are equal to the corresponding components of the applied force. This produces five equations, which when combined with an equation of state, provide sufficient information for the determination of six variables: p, ρ, T, u, v, w typically.

1.1 Modelling of Flow

In obtaining the basic equations of fluid motion, the following philosophy is always followed:

1. Choose the appropriate fundamental physical principles from the laws of physics, such as
   - Mass is conserved,
   - *F=ma* (Newton's 2nd law),
   - Energy is conserved.
2. Apply these physical principles to a suitable model of flow.
3. From this application, extract the mathematical equations, which embody such physical principles.

1.1.1 Compressible, unsteady, two dimensional conservation form of equation for inviscid flow.

First, for the sake of simplicity let us consider two-dimensional airflow. The governing equation for the flow where the dissipative, transport phenomena of viscosity, mass diffusion, body forces (gravitational, electric and magnetic), which act directly on the volumetric mass of the fluid element and thermal conductivity are neglected:

Continuity Equation:

$$\frac{\partial \rho}{\partial t}+\nabla\left(\rho \bar{v}\right)=\frac{\partial \rho}{\partial t}+\left[\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}\right]=0\,;$$

Momentum Equation (Euler equations):

x-component: $$\frac{\partial(\rho u)}{\partial t}+\nabla\left(\rho u\bar{v}\right)=-\frac{\partial p}{\partial x};$$

y-component: $$\frac{\partial(\rho v)}{\partial t}+\nabla\left(\rho v\bar{v}\right)=-\frac{\partial p}{\partial y};$$

Energy Equation:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\left[\rho\left(e+\frac{V^2}{2}\right)\bar{v}\right]=$$

$$=\rho\dot{q}-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y};$$

where: $\bar{v} = u\bar{i} + v\bar{j}$, $V^2 = u^2 + v^2$ and e is the internal energy. [8]

Forms of the governing equations particularly suited for CFD: According to previous remarks, the conservation form of all the governing equations: continuity, momentum and energy are given by

$$\boxed{\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0;}$$

where

$$U = \left\{ \begin{matrix} \rho \\ \rho u \\ \rho v \\ 0 \\ \rho\left(e + \frac{V^2}{2}\right) \end{matrix} \right\};$$

$$F = \left\{ \begin{matrix} \rho u \\ \rho u^2 + p \\ \rho vu \\ 0 \\ \rho\left(e + \frac{V^2}{2}\right)u + pu - k\frac{\partial T}{\partial x} \end{matrix} \right\};$$

$$G = \left\{ \begin{matrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ 0 \\ pv - k\frac{\partial T}{\partial y} \end{matrix} \right\};$$

1.1.2 Equation for viscous flow.

Secondly, it would be to arrange more complex calculation for transonic airflow is provided next. The conservation form of governing equation for an unsteady, three dimensional, compressible, viscous flows are:

<u>Continuity Equation:</u>

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \bar{v}) = 0;$$

<u>Momentum Equation:</u>

x-component:

$$\frac{\partial(\rho u)}{\partial t} + \nabla(\rho u \bar{v}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x;$$

y-component:

$$\frac{\partial(\rho v)}{\partial t} + \nabla(\rho v \bar{v}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y;$$

z-component:

$$\frac{\partial(\rho w)}{\partial t} + \nabla(\rho w \bar{v}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z;$$

<u>Energy Equation:</u>

$$\frac{\partial}{\partial t}\left[\rho\left(e + \frac{V^2}{2}\right)\right] + \nabla\left[\rho\left(e + \frac{V^2}{2}\right)\bar{v}\right] = \rho\dot{q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) +$$
$$+ \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} -$$
$$- \frac{\partial(wp)}{\partial z} + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} +$$
$$+ \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} +$$
$$+ \frac{\partial(w\tau_{zz})}{\partial z} + \rho\bar{f}\bar{v};$$

Forms of the governing equations particularly suited for CFD: The conservation form of all the governing equations: continuity, momentum and energy are given by

$$\boxed{\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = J;}$$

where

$$U = \left\{ \begin{matrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho\left(e + \frac{V^2}{2}\right) \end{matrix} \right\};$$

$$F=\begin{Bmatrix} \rho u \\ \rho u^2+p-\tau_{xx} \\ \rho vu-\tau_{xy} \\ \rho wu-\tau_{xz} \\ \rho\left(e+\dfrac{V^2}{2}\right)u+pu-k\dfrac{\partial T}{\partial x}-u\tau_{xx}-v\tau_{xy}-w\tau_{xz} \end{Bmatrix};$$

$$G=\begin{Bmatrix} \rho v \\ \rho uv-\tau_{yx} \\ \rho v^2+p-\tau_{yy} \\ \rho wv-\tau_{yz} \\ \rho\left(e+\dfrac{V^2}{2}\right)v+pv-k\dfrac{\partial T}{\partial y}-u\tau_{yx}-v\tau_{yy}-w\tau_{yz} \end{Bmatrix};$$

$$H=\begin{Bmatrix} \rho w \\ \rho uw-\tau_{yx} \\ \rho vw-\tau_{zy} \\ \rho w^2+p-\tau_{zz} \\ \rho\left(e+\dfrac{V^2}{2}\right)w+pw-k\dfrac{\partial T}{\partial z}-u\tau_{zx}-v\tau_{zy}-w\tau_{zz} \end{Bmatrix};$$

$$J=\begin{Bmatrix} 0 \\ \rho f_x \\ \rho f_y \\ \rho f_z \\ \rho\left(uf_x+vf_y+wf_z\right) \end{Bmatrix};$$

2 Rough Sketch to Plane

2.1 Finite volume method.

Discretization of the partial equations is called finite differences method, and discretization of integral form of the equations is called finite volume method. The finite volume method was originally developed as a special finite difference formulation. It is central to four of the five main commercially available CFD codes: PHOENICS, FLUENT, FLOW3D and STAR-CD. The numerical algorithm consist of the following steps:

1. Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.
2. Discretization involves the substitution of a variety of finite-difference-type approximation for the terms in the integrated equation representing flow process such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations.
3. Solution of the algebraic equations by an iterative method.

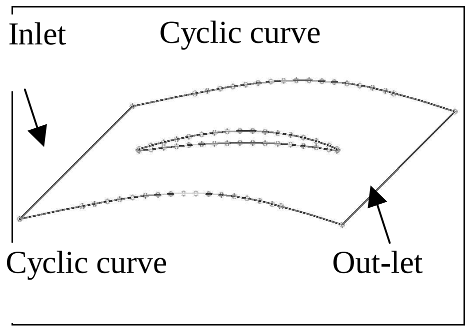
There are three mathematical concepts can be useful for our calculation: **convergence, consistency and stability.**

2.2 Grids and its transformations.

Usually the physical and computational space is different. The physical space may be nonuniform, curvilinear according to shape of body. The governing partial differential equations are solved by a finite difference method carried out in the rectangular computational space. So, the governing equations must be transformed from physical space to computational space and backwards. In this project, it is solved in C, C++ computer program also.

2.3 Explicit or implicit scheme.

In the explicit approach, by definition, each difference equation contains only one unknown and therefore can be solved explicitly for this unknown in straightforward manner. This method is simple, but in some cases not too stable and the computational time is long. The better approach is the implicit scheme. By definition, an implicit approach (for example Crank-Nicolson form) is one where the unknowns must be obtained by means of a simultaneous solution of the difference equations applied at all the grid points arrayed at given time level. Because of this need to solve large system of simultaneous algebraic equations, implicit methods are usually involved with the manipulations of large matrices. This procedure more stable, but more complicated and time spending with truncation error



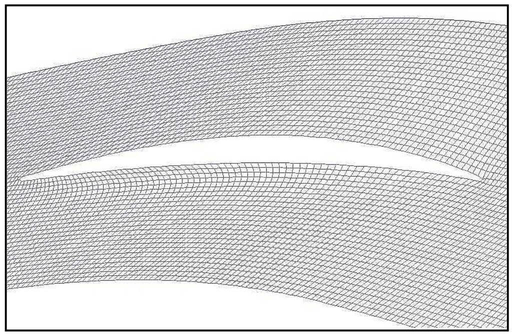


Fig. 1.
NACA RM E52C27 Transonic Axial flow compressor, rotor blade geometry (with flow channel) slice at medium section in FLUENT program

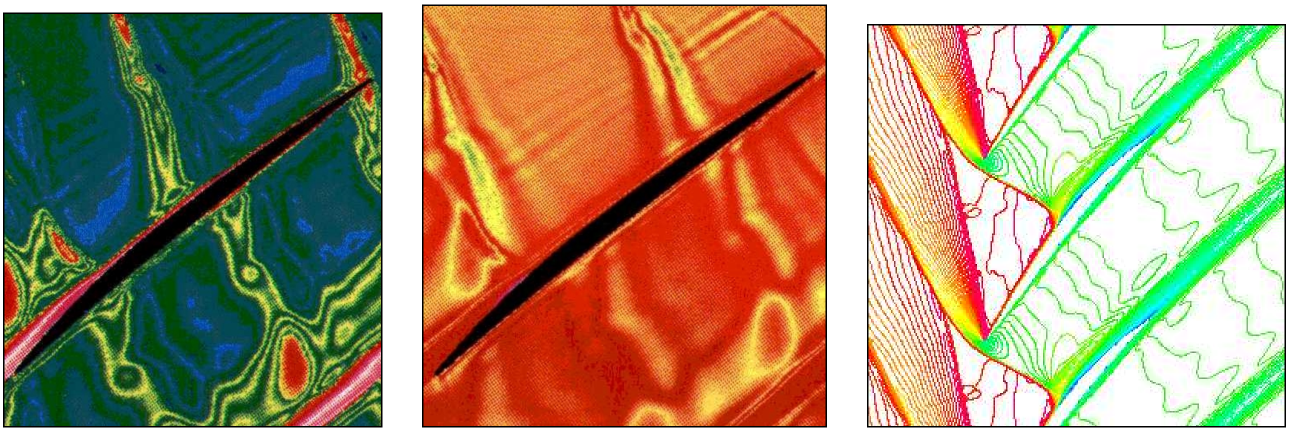


Fig. 2.
Pressure level in the transonic axial compressor blade row (according to CFD Branch (from CFD Gallery))

(because of large matrices) then explicit approach.[10]

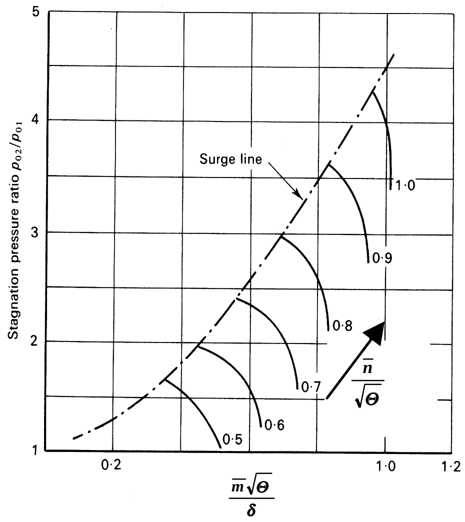
After previous calculation, the next step the solution of full Navier-Stokes equation with boundary layer interaction will be considered for an unsteady, compressible, viscous transonic airflow. This is the final goal of this project. After this we can build the pressure field for all stage in the transonic axial compressor to get the compressor characteristics.

2.4 Problems

However, there are any inaccuracies:

- Because of discretization error, which is the difference between the exact analytical and solution of the partial differential equation and the solution of the corresponding difference equation (in CFD).
- Because of round-off error, the numerical error introduced after repetitive number of calculation in which the computer is constantly rounding the numbers to some significant figure (in CFD).
- Because of neglecting of second order terms in the partial differential equation (in CFD).
- Because of neglecting of interaction of compressor stages (in mathematical model).

The other aim of the theses to work out a computational method in C++ program, which is able to calculate transonic airflow in the axial compressors. For the check of this procedure, it would be some computational software for example FLUENT. This software is applicable



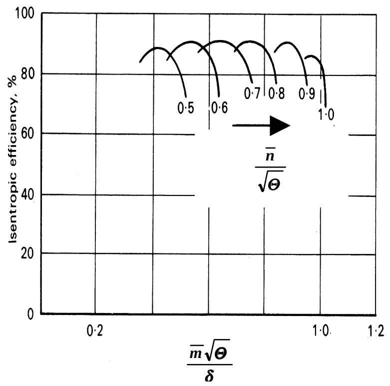


Fig. 3.**Axial compressor characteristics and its isentropic efficiency**

all over the world. At the moment I work on a computational procedure in C program, which is able to convert any forms and expression from the physical domain (x, y) into the mathematical or computational domain (ξ, η) and back.

2.5 Nomenclature

- u: velocity of x direction,
- v: velocity of y direction,
- w: velocity of z direction,
- p: pressure,
- ρ: density,
- τ: stress,
- f: body force,
- k: thermal conductivity,
- T: temperature,
- $\dot{q}$: volumetric heat per unit mass,
- x: distance in i direction in cartesian space,
- y: distance in j direction in cartesian space,
- z: distance in k direction in cartesian space,
- $\boldsymbol{\theta}$: stagnation temperature ratio,
- $\boldsymbol{\delta}$: stagnation pressure ratio,
- $\bar{\boldsymbol{n}}$: relative number of revolution,
- $\bar{\boldsymbol{m}}$: relative mass flow rate.

References

[1] Springer Verlag: HÜTTE Die Grundlagen der Ingenieurwissenschaften (1993),

[2] J. Kevorkian, J. D. Cole: Perturbation Methods in Applied Mathematics (1981),

[3] C. A. J Fletcher: Computational Techniques for Fluid Dynamics (1988),

[4] Dr. József Grúber, Tibor Szentmártony: Gas Dynamics (1952),

[5] Maurice J. Zucow, Joe D. Hoffman: Gas Dynamics (1976),

[6] J. D. Cole, L. P. Cook: Transonic Aerodynamics (1986),

[7] H. K. Versteeg, W.Malalasekera: An Introduction to Computational Fluid Dynamics: The Finite Element Method,

[8] John D. Anderson, JR.: Computational Fluid Dynamics (1995),

[9] Culbert B. Laney: Computational Gasdynamics (1998).