Proceedings of ASME Turbo Expo 2020
Turbomachinery Technical Conference and Exposition
GT2020
June 22-26, 2020, London, England

GT2020-14994

AN APPROACH TO MODEL A LOSSLESS JUNCTION FOR FLUID NETWORK CALCULATIONS IN TURBOMACHINERY

Andrii Khandrymailov[1], Leonid Moroz[1], Viktor Yevlakhov[1], Shanel Staple[2], Gregory Vogel[2]

[1]SoftInWay Inc., Burlington, Massachusetts
[2]Power Systems Mfg., LLC (PSM) - Ansaldo Energia Group, Jupiter, FL

ABSTRACT

Turbine secondary flow system calculations are usually performed by utilizing the thermal-fluid network approach. The network consists of branches and nodes. Fluid branches usually describe the flow resistance of the correspondent fluid path section, while fluid nodes are used to connect fluid branches between each other. Fluid flow in branches and nodes is described by the set of conservation equations such as mass and momentum equations. For fluid nodes, usually total or static pressure is used as a variable in these equations. However, such an approach may yield a system of equations that cannot be solved (for theoretical cases where zero resistance branches are present) or may produce significant differences in results compared to the more precise CFD solution (for real cases with resistances). This paper provides an approach on how to create a model of a fluid lossless junction in order to solve the mentioned problems. Such a junction enables the connection of any number of inflow and outflow fluid branches without bringing any flow resistance, which allows fluid branches to handle flow resistance influence by themselves.

The proposed model is based on the idea, that each node should contain not just one pressure (static or total) as a variable, but two variables - both static and total pressures. Such a node can be used to model junctions of different types with flow mixing, as well as separation and sudden cross-sectional area changes. With this approach it is also possible to model chambers by modifying just one equation. Also, this new method can be applied for both compressible and incompressible calculation types and can handle chocked flows as well.

Keywords: secondary air flow, nodes, flow network, multi-branch connection.

1. INTRODUCTION

Thermal-fluid network approach allows modelling of thermal-fluid systems, such as turbine secondary flow system, by representing fluid paths and solid structures as a connected set of 0D-1D components, as shown in Fig. 1. Compared to a 3D CFD solution, this approach can generate quick predictions for gas turbine internal flow systems on different design phases. Also, it allows modeling on different levels of accuracy and complexity.

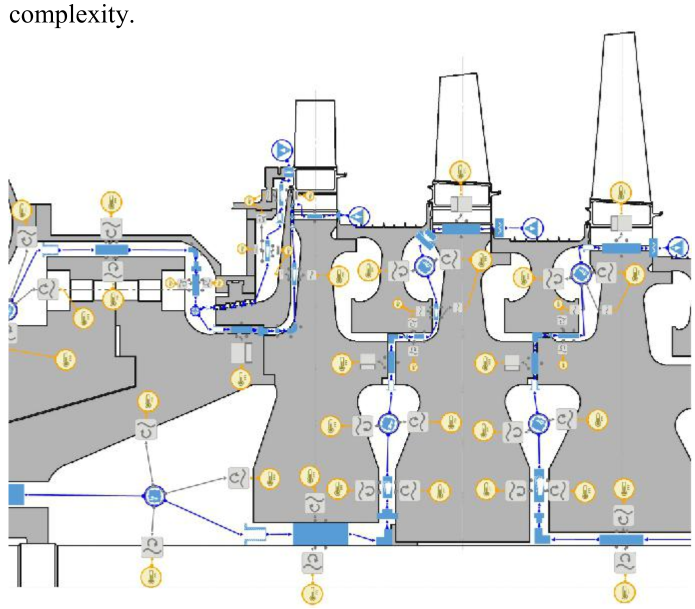

FIGURE 1: EXAMPLE OF THERMAL-FLUID NETWORK

This paper deals with the fluid part of the network, which consists of fluid components. The purpose of such components is to represent some part of the system and calculate fluid flow properties (pressures, densities, velocities, etc.) according to the given data (geometry information, empirical data, etc.). Each component may consist of one or several elements, which can be branches and nodes. Branch could be treated as a representative of flow rate through some section, while node – as a connection point or volume where flows with different properties may be mixed and separated.

Branches always have two connections, which could be treated as inlet and outlet. Nodes can be boundary or internal. A boundary node is used to end a branch (or several branches depending on computational algorithm implementation). An internal node serves to connect branches, so it can have multiple connections. The key point for an internal node is that its model should not introduce additional losses to the fluid system (losses are responsibilities of branches). So, an internal node should transfer fluid flow from inflow branches to outflow branches without any additional losses.

It is common practice to utilize conservation of mass, energy and other conservation equations for nodes while just one equation – conservation of momentum – for branches [1-3]. Accordingly, branches contain just one variable for the above equations – mass flow rate (or velocity), while nodes – all others (including pressure, enthalpy or temperature, etc.) except mass flow rate.

Usually, while dealing with conservation of mass and momentum equations, only total or static pressure is used as a variable for an internal node [1-5]. It is mentioned by Sultanian [4], that "There is often a debate among gas turbine engineers about which pressure, static or total, should be used at the junctions in a flow network to compute mass flow rate through the connecting elements". According to Botha [5], where different modeling methods and commercial codes in the thermal-fluid network area were discussed, "Solving flow through junctions can prove to be problematic. … In Flownet this has been overcome by solving for total pressure at junctions". In the next sections it will be shown, that the approach with static or total pressure as one variable may yield a system of equations that either cannot be solved or the results are significantly different compared to more precise CFD solution. Similar empirical correlations for branches (used to describe losses) combined with differing underlying approaches for internal nodes may produce different results. In order to generate a more accurate solution, a new method proposes the use of both static and total pressures as variables for internal nodes. This approach is beneficial for cases where an internal node must connect more than two branches and where dynamic pressure from upstream to downstream branches is partially preserved (like in Wyes, Tees, and other types of manifold junctions) without treating them in a special manner.

2. A PROBLEM TO USE TOTAL PRESSURE AS A VARIABLE

There is a flow mixing problem in Fig. 2, which cannot be solved by using total pressure as a variable for an internal node.

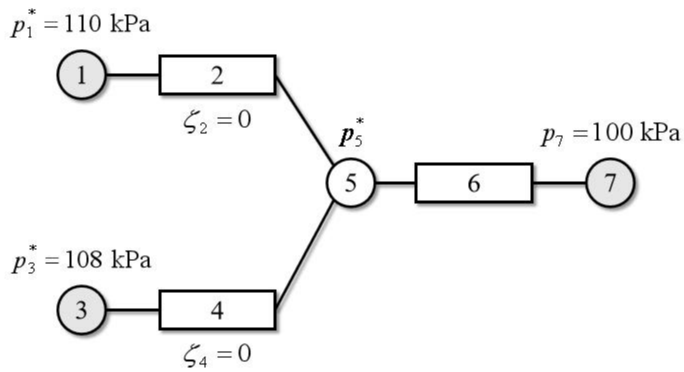

FIGURE 2: FLOW MIXING PROBLEM

In this problem there are two inlet boundary nodes 1 and 3, for which different total pressures p_1^* and p_3^* are specified. There are three branches 2, 4, and 6, which are connected at the internal node 5 with the unknown total pressure variable $\boldsymbol{p}_5^*$. Branches 2 and 4 have zero resistance coefficients ζ_2 and ζ_4. There is also one outlet boundary node 7 with specified static pressure p_7.

Momentum equation for a branch with resistance can be written in the next form

$$\Delta p^* = \zeta \cdot \rho \cdot u^2/2 \,, \tag{1}$$

which states, that the total pressure drop between the inlet and outlet of a branch equals a resistance coefficient multiplied by dynamic pressure at a specific cross-section. If resistance coefficient is zero, then total pressure drop equals zero:

$$\Delta p^* = 0 \,. \tag{2}$$

Applying Eq. (2) to the branches 2 and 4 (see Fig. 2) gives:

$$p_1^* - \boldsymbol{p}_5^* = 0 \,; \tag{3}$$

$$p_3^* - \boldsymbol{p}_5^* = 0 \,. \tag{4}$$

If specified values of total pressures for inlet boundary nodes are substituted into the above equations, results will contradict each other:

$$\boldsymbol{p}_5^* = 110 \text{ kPa} \,; \tag{5}$$

$$\boldsymbol{p}_5^* = 108 \text{ kPa} \,. \tag{6}$$

So, while using algorithms based on the approach of total pressure as a variable for an internal node, solution for similar cases must not converge or will yield wrong results.

3. A PROBLEM TO USE STATIC PRESSURE AS A VARIABLE

There is a flow separation problem in Fig. 3, which cannot be solved by using static pressure as a variable for an internal node.

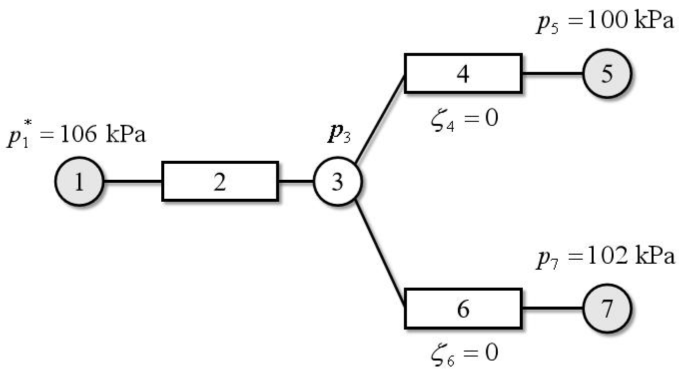

FIGURE 3: FLOW SEPARATION PROBLEM

In this problem there is one inlet boundary node 1 with specified total pressure p_1^*. There are three branches 2, 4, and 6, which are connected at the internal node 3 with the unknown static pressure variable $\boldsymbol{p}_3$. Branches 4 and 6 have zero resistance coefficients ζ_4 and ζ_6. There is also two outlet boundary nodes 5 and 7, for which different static pressures p_5 and p_7 are specified.

For steady state flow in the absence of mass and resistance forces according to the CFD finite volume method the difference of forces due to the static pressures on the inlet and outlet of a branch must equal zero:

$$p_{in} \cdot A - p_{out} \cdot A = 0\ , \tag{7}$$

where A is the cross-sectional area of a branch. Applying this equation to the branches 4 and 6 (see Fig. 3) gives:

$$\boldsymbol{p}_3 \cdot A_4 - p_5 \cdot A_4 = 0\ ; \tag{8}$$

$$\boldsymbol{p}_3 \cdot A_6 - p_7 \cdot A_6 = 0\ ; \tag{9}$$

If specified values of static pressures for outlet boundary nodes are substituted into the above equations, results will also contradict each other:

$$\boldsymbol{p}_3 = 100 \text{ kPa}\ ; \tag{10}$$

$$\boldsymbol{p}_3 = 102 \text{ kPa}\ . \tag{11}$$

So, while using algorithms based on the approach of static pressure as a variable for an internal node, solution for similar cases also must not converge or will yield wrong results.

4. AN APPROACH TO USE BOTH STATIC AND TOTAL PRESSURES

In order to solve the problems, which were presented above, a new method is proposed. The first conception of the method is that each internal node should contain not just one pressure (static or total) as a variable, but two variables – both static and total pressures. The second conception is that branches should treat total pressure from an upstream node as total pressure on their upstream faces, and static pressure from a downstream node – as static pressure on their downstream faces, as shown in Fig. 4. It means, that momentum equations for branches should be based on upstream total pressure and downstream static pressure.

For an internal node such a pressures treatment means, that all inflow branches on their downstream face have the same static pressure, but allowed having different total pressures (due to different inlet boundary conditions). On the other hand, all outflow branches on their upstream face have the same total pressure, but allowed having different static pressures (due to different outlet boundary conditions). The important issue here is the necessity to add an additional equation for each internal node because of the additional variable (two pressures instead of one). This equation must describe the correlation between static and total pressure variables at an internal node in order for momentum equations of inflow and outflow branches to be coupled.

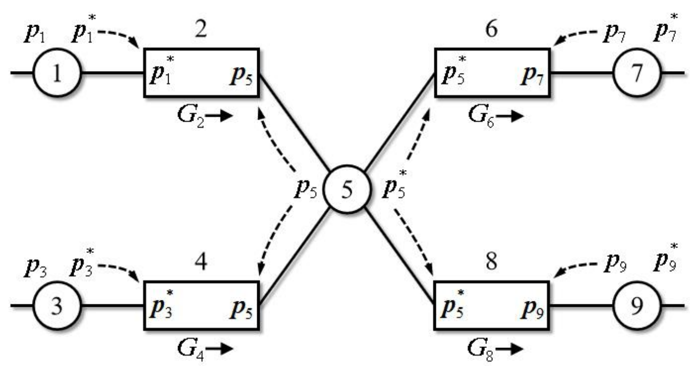

FIGURE 4: PRESSURES TREATMENT

As was stated above, all inflow branches of an internal node are allowed having different total pressures on their downstream faces. These total pressures yield resulting total pressure in the internal node. In order to derive a relationship between these pressures and the resulting pressure at the internal node (node 5), the principle of resultant force can be used, as shown on Fig 5.

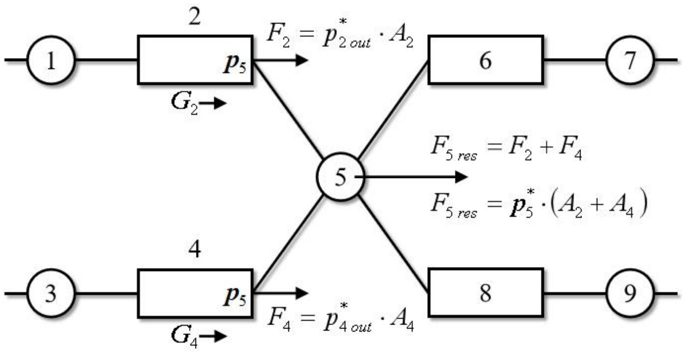

FIGURE 5: FORCES OF TOTAL PRESSURES

Figure 5 shows force of total pressure on the downstream face of each inflow branch, which can be found by multiplying total pressure by the area of corresponding downstream face:

$$F_j = p^*_{j\,out} \cdot A_j \,, \tag{12}$$

where j is the index of an inflow branch. An assumption that these forces act at the same direction is introduced here, in order not to deal with losses due to flows interaction. Based on this assumption, the arithmetic sum of the forces gives resultant force:

$$\sum F_j = F_{res} \,, \tag{13}$$

which also equals resulting total pressure in the internal node multiplied by total inflow cross-sectional area (see also Fig. 5):

$$F_{res} = \boldsymbol{p}^* \cdot \sum A_j \,. \tag{14}$$

Combining Eq. (12) – Eq. (14) gives:

$$\boldsymbol{p}^* \cdot \sum A_j = \sum p^*_{j\,out} \cdot A_j \,. \tag{15}$$

Total pressures on the downstream face of inflow branches $p^*_{j\,out}$ in Eq. (15) are unknown. But they can be derived based on the static pressure variable ($\boldsymbol{p}$) in the internal node and mass flow rate variable ($\boldsymbol{G}_j$) of the corresponding inflow branch:

$$p^*_{j\,out} = f\left(\boldsymbol{p}, \boldsymbol{G}_j\right). \tag{16}$$

Equation (15) together with Eq. (16) yield needed additional equation for an internal node, which describe the correlation between static and total pressure variables at this node. For incompressible flow Eq. (16) can be written as

$$p^*_{j\,out} = \boldsymbol{p} + 0.5 \cdot \frac{\boldsymbol{G}_j^2}{\rho_j \cdot A_j^2} \,. \tag{17}$$

where ρ_j is the density on the downstream face of an inflow branch, which could be treated as a frozen value and updated later according to the chosen solution algorithm.

For the compressible calculation type Eq. (16) can be presented as a set of thermodynamic calculations based on the given equation of state. But the idea is the same as in Eq. (16) – total pressure on the downstream face of an inflow branch must be found based on static pressure and mass flow rate variables. Some properties in these calculations could also be treated as frozen values (for example enthalpy or temperature) and updated later.

An internal node in the presented method can serve as a lossless junction for any number of branches in order to solve flow mixing and separation problems without bringing any flow resistance to its model.

5. SOLUTION TO FLOW MIXING PROBLEM

In order to present solution to the flow mixing problem that was discussed before, some additional information about task definition has to be given. The resistance coefficient ζ_6 of branch 6 is set to zero. The fluid flow is considered incompressible with constant density $\rho = 1000$ kg/m^3 along the network.

The scheme corresponding to the new approach is presented in Fig. 6. For simplicity, equations will be written only for correct direction of fluid flow, but real computational algorithm should compose equations taking into account flow direction, which can be defined for each branch by the current sign of the mass flow rate (or velocity) variable.

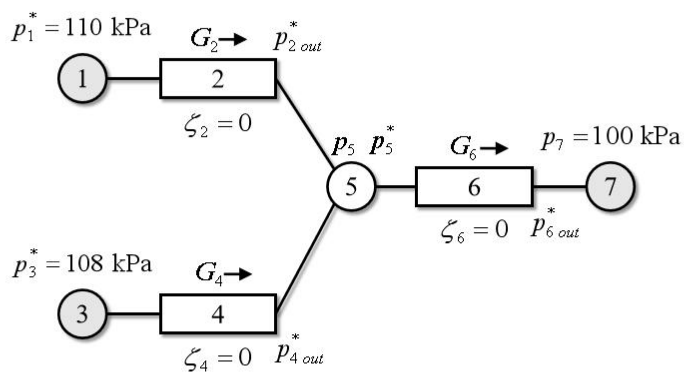

FIGURE 6: FLOW MIXING PROBLEM USING NEW APPROACH

Applying Eq. (2) to the branches 2, 4, and 6 gives:

$$p^*_1 - p^*_{2\,out} = 0 \,; \tag{18}$$

$$p^*_3 - p^*_{4\,out} = 0 \,; \tag{19}$$

$$\boldsymbol{p}^*_5 - p^*_{6\,out} = 0 \,. \tag{20}$$

Flow total pressures on the downstream face of the branches can be derived using Eq. (17):

$$p^*_{2\,out} = \boldsymbol{p}_5 + 0.5 \cdot \frac{\boldsymbol{G}_2^2}{\rho \cdot A_2^2}\,; \quad (21)$$

$$p^*_{4\,out} = \boldsymbol{p}_5 + 0.5 \cdot \frac{\boldsymbol{G}_4^2}{\rho \cdot A_4^2}\,; \quad (22)$$

$$p^*_{6\,out} = p_7 + 0.5 \cdot \frac{\boldsymbol{G}_6^2}{\rho \cdot A_6^2}\,. \quad (23)$$

Above equations allow rewriting Eq. (18)-(20) as follows:

$$p^*_1 - \boldsymbol{p}_5 - 0.5 \cdot \frac{\boldsymbol{G}_2^2}{\rho \cdot A_2^2} = 0\,; \quad (24)$$

$$p^*_3 - \boldsymbol{p}_5 - 0.5 \cdot \frac{\boldsymbol{G}_4^2}{\rho \cdot A_4^2} = 0\,; \quad (25)$$

$$\boldsymbol{p}^*_5 - p_7 - 0.5 \cdot \frac{\boldsymbol{G}_6^2}{\rho \cdot A_6^2} = 0\,. \quad (26)$$

For internal node 5 conservation of mass equation gives:

$$\boldsymbol{G}_2 + \boldsymbol{G}_4 - \boldsymbol{G}_6 = 0\,. \quad (27)$$

Finally, additional equation for internal node 5 can be written based on Eq. (15) as follows:

$$\boldsymbol{p}^*_5 \cdot (A_2 + A_4) = p^*_{2\,out} \cdot A_2 + p^*_{4\,out} \cdot A_4\,. \quad (28)$$

Taking into account Eq. (21) and (22) the last equation takes the next form:

$$\boldsymbol{p}^*_5 \cdot (A_2 + A_4) - \left(\boldsymbol{p}_5 + 0.5 \cdot \frac{\boldsymbol{G}_2^2}{\rho \cdot A_2^2}\right) \cdot A_2 - \left(\boldsymbol{p}_5 + 0.5 \cdot \frac{\boldsymbol{G}_4^2}{\rho \cdot A_4^2}\right) \cdot A_4 = 0 \quad (29)$$

Equations (24)-(27) and (29) form the system of five equations with five variables (p_5, p^*_5, G_2, G_4, G_6), which can be solved, for example, using Newton-Raphson method. As an initial conditions mass flow rate values might be set to zero; total and static pressure values at internal node 5 might be set to the value of static pressure at the outlet boundary node.

The results of calculations for three combinations of cross-sectional area values are shown below.

A) $A_2 = 0.01\ \text{m}^2$, $A_4 = 0.01\ \text{m}^2$, $A_6 = 0.01\ \text{m}^2$:
$p^*_5 = 109\ \text{kPa}$, $p_5 = 106.6\ \text{kPa}$,
$G_2 = 25.93\ \text{kg/s}$, $G_4 = 16.5\ \text{kg/s}$, $G_6 = 42.43\ \text{kg/s}$.

In this case cross-sectional areas of all branches are equal. The mass flow rate value of branch 2 is higher than the one of branch 4, because of the higher inlet total pressure. Total pressure at internal node 5 is the average of inlet total pressures, because the cross-sectional areas of branches 2 and 4 are the same.

B) $A_2 = 0.01\ \text{m}^2$, $A_4 = 0.005\ \text{m}^2$, $A_6 = 0.01\ \text{m}^2$:
$p^*_5 = 109.33\ \text{kPa}$, $p_5 = 105.13\ \text{kPa}$,
$G_2 = 31.22\ \text{kg/s}$, $G_4 = 11.98\ \text{kg/s}$, $G_6 = 43.2\ \text{kg/s}$.

In this case the cross-sectional area of branch 2 is two times larger than the one of branch 4. Because of this, the mass flow rate value of branch 2 is even higher than the one from previous case. Also the value of total pressure at internal node 5 is closer to the inlet total pressure of branch 2.

C) $A_2 = 0.005\ \text{m}^2$, $A_4 = 0.01\ \text{m}^2$, $A_6 = 0.01\ \text{m}^2$:
$p^*_5 = 108.67\ \text{kPa}$, $p_5 = 104.76\ \text{kPa}$,
$G_2 = 16.18\ \text{kg/s}$, $G_4 = 25.45\ \text{kg/s}$, $G_6 = 41.63\ \text{kg/s}$.

This is the opposite case to the previous one, where cross-sectional area of branch 2 is two times smaller than the one of branch 4. Correspondingly, mass flow rate value for branch 2 is lower than the one for branch 4, and also total pressure at internal node 5 is closer to the inlet total pressure of branch 4.

6. SOLUTION TO FLOW SEPARATION PROBLEM

The scheme of flow separation problem corresponded to the new approach is presented in Fig. 7. The resistance coefficient ζ_2 of branch 2 is set to zero. The fluid flow is considered incompressible with constant density $\rho = 1000\ \text{kg/m}^3$ along the network.

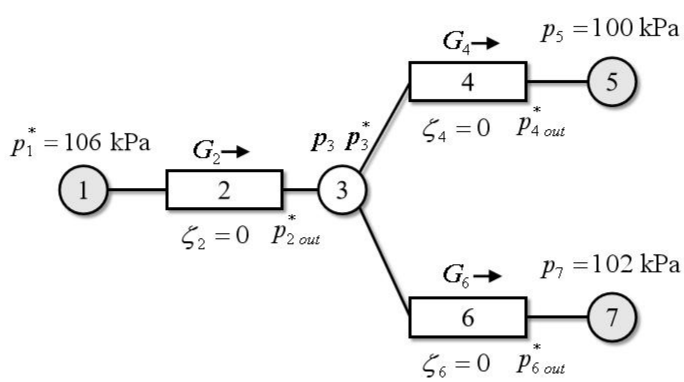

FIGURE 7: FLOW SEPARATION PROBLEM USING NEW APPROACH

Similarly to the problem of flow mixing, five equations can be written in order to form system of equations with five variables (p_3 , p_3^* , G_2 , G_4 , G_6). These equations in their final form are presented below.

$$p_1^* - \boldsymbol{p}_3 - 0.5 \cdot \frac{\boldsymbol{G}_2^2}{\rho \cdot A_2^2} = 0 ; \quad (30)$$

$$\boldsymbol{p}_3^* - p_5 - 0.5 \cdot \frac{\boldsymbol{G}_4^2}{\rho \cdot A_4^2} = 0 ; \quad (31)$$

$$\boldsymbol{p}_3^* - p_7 - 0.5 \cdot \frac{\boldsymbol{G}_6^2}{\rho \cdot A_6^2} = 0 ; \quad (32)$$

$$\boldsymbol{G}_2 - \boldsymbol{G}_4 - \boldsymbol{G}_6 = 0 ; \quad (33)$$

$$\boldsymbol{p}_3^* - \boldsymbol{p}_3 - 0.5 \cdot \frac{\boldsymbol{G}_2^2}{\rho \cdot A_2^2} = 0 . \quad (34)$$

The results of calculations for three combinations of cross-sectional area values are shown below.

A) $A_2 = 0.01\ \text{m}^2$, $A_4 = 0.01\ \text{m}^2$, $A_6 = 0.01\ \text{m}^2$:
$p_3^* = 106\ \text{kPa}$, $p_5 = 86.2\ \text{kPa}$,
$G_2 = 62.92\ \text{kg/s}$, $G_4 = 34.64\ \text{kg/s}$, $G_6 = 28.28\ \text{kg/s}$.

In this case cross-sectional areas of all branches are equal. The mass flow rate value of branch 4 is higher than the one of branch 6, because of the lower outlet static pressure. Total pressure at internal node 3 is equal to the inlet total pressure.

B) $A_2 = 0.01\ \text{m}^2$, $A_4 = 0.01\ \text{m}^2$, $A_6 = 0.005\ \text{m}^2$:
$p_3^* = 106\ \text{kPa}$, $p_5 = 94.1\ \text{kPa}$,
$G_2 = 48.78\ \text{kg/s}$, $G_4 = 34.64\ \text{kg/s}$, $G_6 = 14.14\ \text{kg/s}$.

In this case, the cross-sectional area of branch 6 is two times smaller than the one from previous case. Because of this, its mass flow rate value becomes two times lower. The mass flow rate value of branch 4 remains the same because total pressure at internal node 3 and outlet static pressure for this branch are the same as in the previous case.

C) $A_2 = 0.01\ \text{m}^2$, $A_4 = 0.005\ \text{m}^2$, $A_6 = 0.01\ \text{m}^2$:
$p_3^* = 106\ \text{kPa}$, $p_5 = 95.6\ \text{kPa}$,
$G_2 = 45.6\ \text{kg/s}$, $G_4 = 17.32\ \text{kg/s}$, $G_6 = 28.28\ \text{kg/s}$.

Compared with case A, this one has branch 4 with two times smaller cross-sectional area. Accordingly, the mass flow rate variable of branch 4 is two times smaller, but mass flow rate of branch 6 remains the same.

7. FLOW MIXING PROBLEM WITH RESISTANCES

In this section, it will be shown that different treatment of pressures as variables for internal nodes may produce noticeable differences in results, even by using the same loss models for branches.

The scheme and data for the problem under consideration is shown in Fig. 8. There is a T-junction with two inlets and one outlet. Inlets have different total pressures and the same total temperature as boundary conditions. Static pressure is used as the outlet boundary condition. The model of fluid is air ideal gas. Both inlet and outlet channels have the same square cross-section with 0.1 m side size.

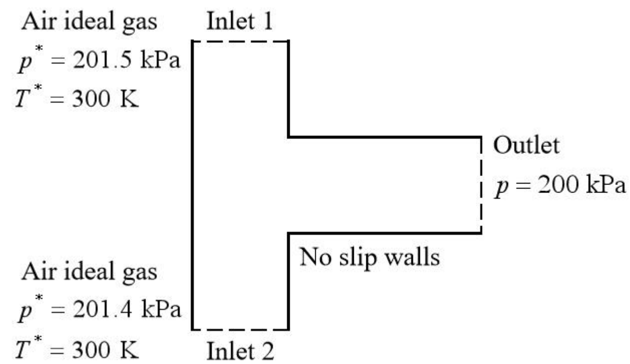

FIGURE 8: FLOW MIXING PROBLEM WITH RESISTANCES

In order to validate 1D fluid network calculations, the CFD solution of this problem has been obtained using ANSYS CFX program. The mesh for CFD model has a mesh size 1.5 million hexahedral elements, with boundary layer to achieve y^+ less than 2 for all solid walls. SST turbulence model was used with 5% of turbulence intensity on inlets as boundary conditions. The length of inlet channels is equal to 0.2 m, and the length of outlet channel is 0.9 m. Figure 9 shows contours of total pressure and other results of CFD calculation, where u – area averaged velocity; G – mass flow rate; p – area averaged static pressure.

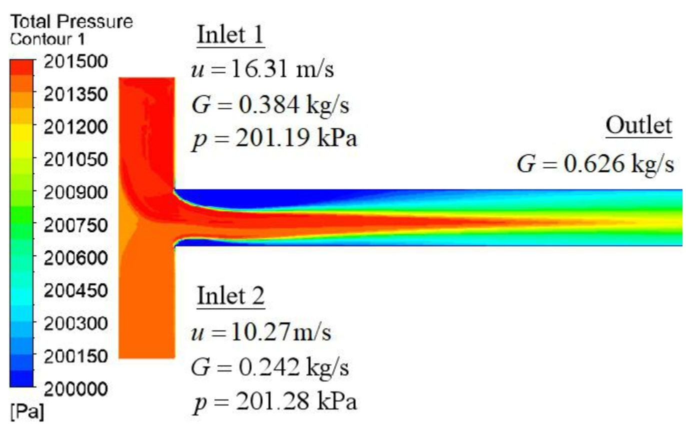

FIGURE 9: CONTOURS OF TOTAL PRESSURE

Fluid-network calculations for this task were performed using AxSTREAM NET™ program, where different approaches to use pressures as variables have been implemented. The network scheme and results of calculations for different approaches are shown in Fig. 10-12. There are two inlet boundary nodes (1 and 3) to define inlet total pressures and total temperature and one outlet boundary node (7) to define outlet static pressure. Also there are three branches (2, 4 and 6) to represent inlet and outlet channels and one internal node (5) to connect these branches. The calculated values at inlet and outlet of the branches in Fig. 10-12 are shown for the next properties: p0 – total pressure; u – velocity; G – mass flow rate; p – static pressure. Additionally, Table 1 shows the comparison between results of fluid network calculations and CFD solution in terms of calculated mass flow rates.

For all approaches of fluid network calculations, the momentum equations for branches were the same:

$$\Delta p^* = \zeta \cdot 0.5 \cdot \frac{G^2}{\rho \cdot A^2}, \tag{35}$$

where Δp^* – total pressure difference between inlet and outlet of the branch; ζ – resistance coefficient; G – mass flow rate through the branch; ρ – average density in the branch; A – cross-sectional area of the branch.

According to Idelchik [6], for this type of junction with given cross-sectional areas, the resistance coefficients for both inlet branches (2 and 4) can be calculated as follows:

$$\zeta = C \cdot \left(2 + 3 \cdot \left(\left(\frac{G}{G_\Sigma}\right)^2 - \frac{G}{G_\Sigma}\right)\right) \Big/ \left(\frac{G}{G_\Sigma}\right)^2, \tag{36}$$

where G – mass flow rate through the given inlet branch (2 or 4); G_Σ – total mass flow rate after mixing in junction, that is the mass flow rate through outlet branch (6); C – coefficient, that depends on the mass flow rate ratio as follows:

$$G/G_\Sigma \le 0.4 \text{, then } C = 0.9 \cdot (1 - G/G_\Sigma); \tag{37}$$

$$G/G_\Sigma > 0.4 \text{, then } C = 0.55. \tag{38}$$

The resistance coefficient for the outlet branch (6) was set equal to zero.

Where necessary, total pressure at inlet or outlet of a branch was calculated based on mass flow rate through the branch and static pressure in a corresponded node, as shown in Eq. (17) as an example for incompressible flow.

Figure 10 shows results of calculation using the new approach. As can be noticed, these results are in a good agreement with CFD solution. Table 1 shows, that the largest difference between the mass flow rates of this calculation and CFD solution is less than 15%. The outlet mass flow rate correlates very well with the CFD solution, with an error of only 0.8%. According to the proposed method, inlet branches (2 and 4) have the same static pressure at their outlets, but the total pressures differ because of differing inlet boundary conditions. Downstream total pressure of internal node 5 is the average of upstream total pressures, because the cross-sectional areas of branches 2 and 4 are the same. The mass flow rate value of branch 2 is higher than the one of branch 4, because of the higher inlet total pressure, that corresponds with results of CFD solution.

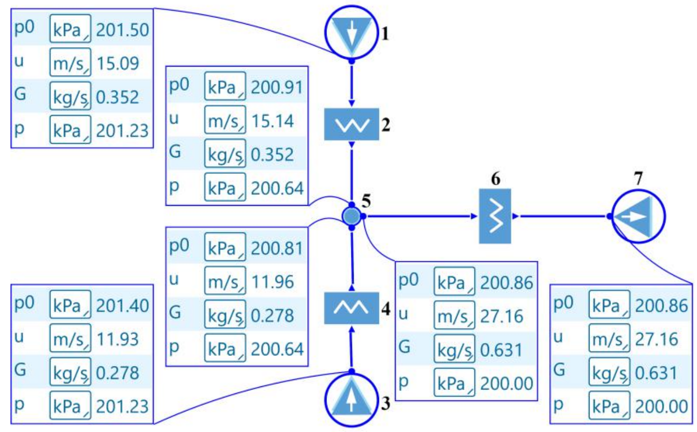

FIGURE 10: VARIANT WITH NEW APPROACH

Figure 11 shows results of calculation using approach with total pressure as one variable for an internal node. As expected from this approach, upstream and downstream total pressures of the internal node (5) have the same value. Table 1 shows that despite the close correlation of outlet mass flow rate with the CFD solution (the difference is about 3%), the inlet mass flow rates are far from CFD results (more than 50%). Also, the mass flow rate value of branch 2 is less than branch 4, which does not correspond with results of the CFD model.

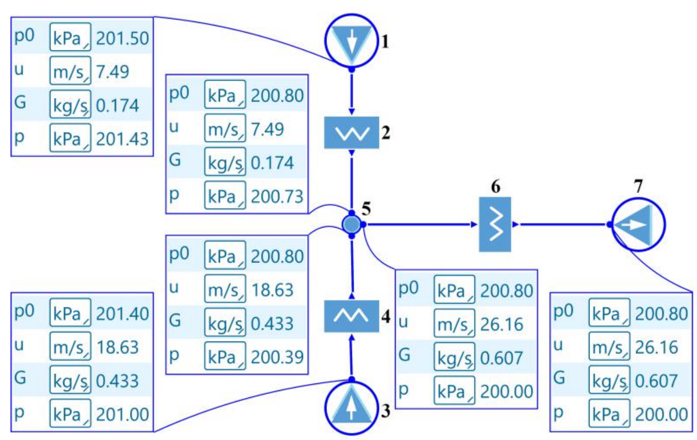

FIGURE 11: VARIANT WITH TOTAL PRESSURE AT NODE

Figure 12 shows the results of calculation using approach with static pressure as one variable for an internal node. As this approach suggests, upstream and downstream static pressures of the internal node (5) have the same value. Table 1 shows overestimation of mass flow rates through inlet branches (more than 50% for inlet 2) compared to CFD results, which yields more than 30% difference for total outlet mass flow rate.

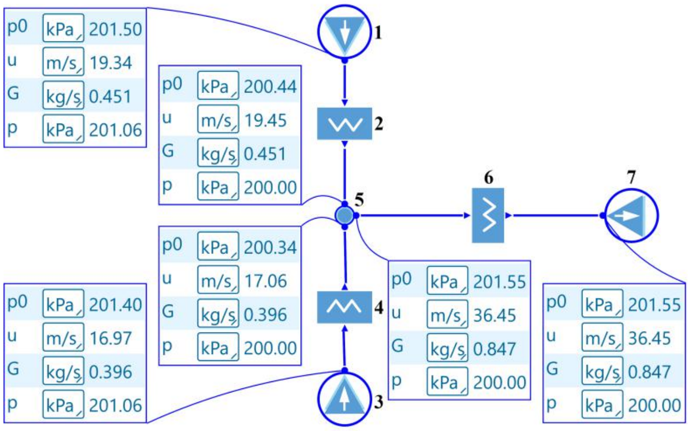

FIGURE 12: VARIANT WITH STATIC PRESSURE AT NODE

As can be seen in Table 1, different approaches to use pressures as variables for internal nodes may produce significant differences in results (more than 50% of mass flow rate) compared to the more precise CFD solution... even when the same loss models for branches are in use. The new approach gives best correlation to the CFD solution.

TABLE 1: COMPARISON OF MASS FLOW RATES

Location	CFD	New approach	Total pressure approach	Static pressure approach
	kg/s	kg/s (difference %)		
Inlet 1	0.384	0.352 (-8.3%)	0.174 (-54.7%)	0.451 (+17.4%)
Inlet 2	0.242	0.278 (+14.9%)	0.433 (+78.9%)	0.396 (+63.6%)
Outlet	0.626	0.631 (+0.8%)	0.607 (-3.0%)	0.847 (+35.3%)

8. CHAMBER MODEL

A chamber (or a plenum) can be considered as an internal node (junction) with no associated dynamic pressure [4], where total pressure equals static pressure. Taking this into account, the additional equation, which must describe the correlation between static and total pressure variables at internal node as a chamber, instead of Eq. (15) together with Eq. (16) takes the next simple form:

$$p^{*} = p\,. \qquad (39)$$

Figure 13 shows an example of calculation with a chamber. There are two inlet boundary nodes with specified different total pressures. There are four branches connected by the chamber element. Also, there are two outlet boundary nodes with specified different static pressures. All branches are supposed to have zero resistance coefficients and the same cross-sectional area $A = 0.01\ \mathrm{m}^2$. The fluid is considered incompressible with constant density $\rho = 1000$ kg/s along the network. By analogy with flow mixing and flow separation problems, the system of equations can be constructed taking into consideration Eq. (39). The resulting solution for this problem should be the same as presented in Fig. 13, where p0 – total pressure; G – mass flow rate; p – static pressure.

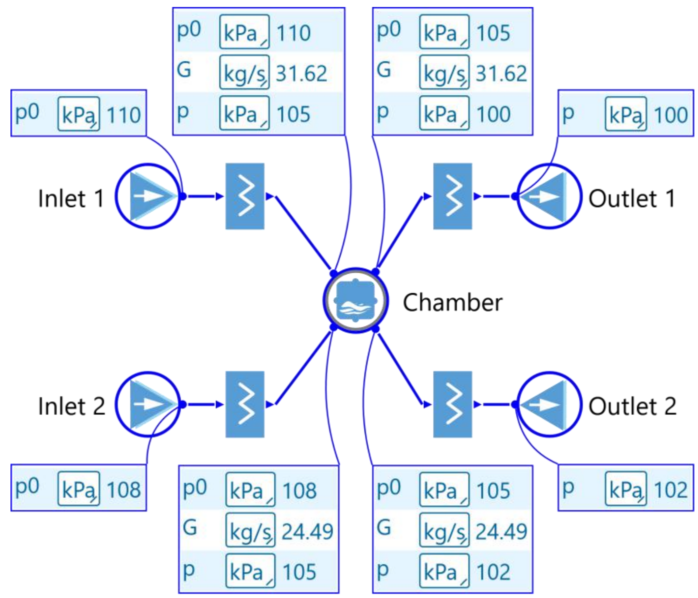

FIGURE 13: EXAMPLE OF CHAMBER MODELING

The key point in this example is that static pressures for the inflow branches of the chamber are the same (p = 105 kPa) and are equal to total pressures for the outflow branches of the chamber (p0 = 105 kPa), as it should be according to the chamber model. As can be noticed, the chamber model introduces additional losses, which are the losses of kinetic energy of all incoming flows.

9. CHOCKED FLOW MODELING

Branch models should be able to deal with chocked flow simulation. For example, to model gas flow through the orifice, a branch model should first check if downstream pressure falls below a critical pressure and then decide what equation to use to calculate mass flow rate through the orifice.

Flow parameters in a chocked branch are calculated based just on upstream conditions. But the fact that static pressure from the downstream node of this branch is not used in its momentum equation, is not yield a decoupling of the total and static pressure variables in the downstream node from the upstream conditions. Because, according to Eq. (15) and Eq. (16) total pressure still depends on mass flow rate (or velocity) variable of inflow branch. So, no modification to the presented model of an internal node is needed to deal with chocked flow, only branch model have to take care of it.

10. UNSTEADY STATE CALCULATIONS

An internal node as a lossless junction may be treated as a non-inertial element, which means it has no associated volume and instantly transfers all perturbations between branches. On the other hand, an internal node as a chamber may be considered as an inertial element with assigned volume. Amount of fluid inside this volume may be changed during the process, which should be taking into account in the mass conservation equation. Following these conventions, Eq. (15), together with Eq. (16) and the next conservation of mass equation, should be used to model lossless junction in unsteady state calculations:

$$\sum \boldsymbol{G}_i = 0\,, \tag{40}$$

where $\boldsymbol{G}_i$ – mass flow rate variables of connected branches. Equation (39) and mass conservation equation of the next form should be used to model chamber in unsteady state calculations:

$$\frac{\partial(\rho \cdot V)}{\partial t} = \sum \boldsymbol{G}_i\,, \tag{41}$$

where the left hand side represents the variation of the mass in the chamber per unit of time (ρ – average fluid density in the chamber; V – volume of the chamber; t – time).

Conservation of energy equation and other conservation equations (if needed) should be constructed for an internal node as a lossless junction or as a chamber based on the same conventions presented above.

11. NUMERICAL ISSUES

The proposed method brings an additional variable and an additional equation to the model of an internal node, which might be one of the most used elements in the fluid network schemes. This increases overall complexity of a system of equations to be solved and potentially increases computational time, compared to the approaches with one pressure variable. Time delay may be more significant while dealing with compressible calculation type, where complex thermodynamic calculations may take place. Still, compared with CFD calculations, the model complexity and time to get solution remain significantly less. There can be potential difficulties with reversed flow calculations and bad initial conditions, but such problems depend on many aspects and relate to many types of network calculations.

12. CONCLUSION

The presented approach to model internal node using both static and total pressures as variables permits obtaining solutions to problems where methods based on one pressure variable yields a system of equations that cannot be solved or accurately converged. This method suits to model flow mixing and separation for any number of branches with any cross-sectional areas, without bringing any flow resistance. It can be used both for compressible and incompressible types of calculations. Compared to one pressure variable approaches, the new method generates more precise solutions for junctions with resistances, where an internal node must connect more than two branches and where dynamic pressure from upstream to downstream branches is partially preserved. So the new approach can benefit in network calculations with Wyes, Tees, and many types of manifold junctions without treating them in a special manner.

In the proposed method, branch models should contain the momentum equation based on total pressure from the upstream node and static pressure from the downstream node. They also should be able to take into account the possibility of flow chocking.

The paper shows how to derive an additional equation, which describes the connection between static and total pressure variables at an internal node. No modification to the model of an internal node is needed to handle chocked flow.

Based on the idea that total pressure is equal to static pressure in a plenum, a simple form of the additional equation is proposed to model chambers.

The presented approach can also be used in unsteady state calculations by using appropriate conservation equations.

REFERENCES

[1] Majumdar, A. K., LeClair, A. C., Moore, R., and Schallhorn, P. A. "Generalized Fluid System Simulation Program, Version 6.0." Technical Report No. NASA/TP-2016-218218, M-1407. NASA Kennedy Space Center, Cocoa Beach, Florida. 2016.

[2] Oliver, Jacobus Coenraad. "Network Modelling of Transient Heat Exchanger Performance." Thesis (Magister Engineering). North-West University, Potchefstroom. 2005.

[3] Hou, Shengping. "A New Unsteady Fluid Network Approach to Simulate the Characteristics of the Air System of a Gas Turbine System." Journal of Energy and Power Engineering 6 (2012): pp. 1751-1757.

[4] Sultanian, Bijay K. *Gas turbines internal flow systems modeling*. Cambridge University Press (2018).

[5] Botha, Frederick. "An Explicit Method for the Analysis of Transient Compressible Flow in Pipe Networks." Thesis (Magister Engineering). School of Mechanical and Materials Engineering at Potchefstroom University for Christian Higher Education, Potchefstroom. 2003.

[6] Idelchik, I. E. *Handbook of Hydraulic Resistance*. Begell House, New York (2008).