

A Report on the Panel Method with Sources

Nathan R. Lehnhof*
BYU Provo, UT

Computational Fluid Dynamics (CFD) enables the analysis and optimization of fluid flows in industries such as aviation, HVAC, and propulsion. The panel method, a foundational CFD technique, represents a body in a fluid using distributed singularities on its surface. Solving the resulting system yields a scalar potential, from which fluid velocities around the body can be determined. This report presents the theory and derivation of the source-based panel method, providing a framework for future CFD applications.

I. Introduction

From the beginning of aviation, modeling and simulation techniques have been used to generate informed designs, and to clarify and correct old data and processes. After disappointing flight tests in 1900 and 1901, the Wright brothers began to question the Lilienthal data that had previously guided their designs. Instead of continuing to rely on this aerodynamic data, they decided to build their own wind tunnel to measure the lift and drag of their models themselves (Figure 1). Through October 1901, the Wright brothers built and tested over one hundred models. This data was critical as they designed their new aircraft, contributing directly to their successful flight in 1903 [1]. Wind tunnel testing, while still quite popular and effective, is quite expensive. Subsonic, professional tests consistently number in the tens of thousands of dollars, with high-end, supersonic tests costing upwards of a million.



Fig. 1 The Wright Brothers 1901 wind tunnel

As computing power increased, designers have leaned more and more towards Computational Fluid Dynamics as a cheaper, faster solution. The first CFD method was proposed in the 1940s by Lewis Fry Richardson. Richardson, a meteorologist, proposed a numerical method for solving partial differential equations to describe fluid flows, but it wasn't until the 1970s and greater computing power that the first CFD software was introduced [3]. With the advent of super computers, engineers can now run simulations previously thought impossible, allowing them to optimize designs. CFD has led to cheaper and quicker design as engineers can run many simulations at once, instead of having to wait for the results of costly wind tunnel tests. Even now, engineers are working to improve the existing CFD theories and techniques to further enhance and accelerate simulations.

The classical hydrodynamic theory is one of the foundational theories of CFD [2]. It states that fluid flow around a body can be modeled by a distribution of sources, sinks, vortices, and dipoles. Panel methods take advantage of this

*Research Assistant, Mechanical Engineering Dept.

theory by defining the surface of the body with a distribution of source panels. These source panels counter the normal component of the free stream velocity, ensuring the no-flow-through condition. Here we will derive the panel solve equation for source panels.

II. Methods

The panel method for source panels discretizes the surface of the body into panels, distributing a source to each panel to satisfy the boundary conditions. For the analysis, we assume that the entire panel has a constant strength source. Since we assume inviscid, incompressible, and irrotational flow, the panel method returns the potential flow equation.

Assume an inviscid, incompressible, irrotational flow (steady for simplicity).

Incompressible (continuity):

$$\nabla \cdot \mathbf{u} = 0. \quad (1)$$

Irrotational:

$$\nabla \times \mathbf{u} = 0 \quad \Rightarrow \quad \mathbf{u} = \nabla \phi, \quad (2)$$

for some scalar potential $\phi(\mathbf{x})$.

Substitute into continuity:

$$\nabla \cdot (\nabla \phi) = \nabla^2 \phi = 0. \quad (3)$$

So the potential satisfies Laplace's equation

$$\nabla^2 \phi(\mathbf{x}) = 0, \quad (4)$$

For a body with no-penetration, the surface velocity is zero ($\mathbf{V}_s = 0$):

$$\nabla \cdot \mathbf{n} = 0 \text{ on } S$$

or, if the body is moving:

$$\nabla \phi \cdot \mathbf{n} = \mathbf{V}_s \cdot \mathbf{n}, \quad \text{where } \mathbf{n} \text{ is the surface normal and } \mathbf{V}_s \text{ is the surface velocity.} \quad (5)$$

In a 3D source panel method, each panel is assigned a constant source strength σ_j . The potential at a point \mathbf{x}_i is obtained by integrating the fundamental solution over the panel surface S_j . Green's function is the fundamental solution for Laplace's equation. Since we gave each panel a constant strength, we can factor out σ_j and find the potential at a point \mathbf{P} of a unit strength source at \mathbf{Q} (Equation 6).

$$G(\mathbf{P}, \mathbf{Q}) = \frac{1}{4\pi r}, \text{ where } r = |\mathbf{P} - \mathbf{Q}| \quad \text{Let } \mathbf{Q} \text{ and } \mathbf{P} \text{ be points on the surface } S. \quad (6)$$

A. Boundary Condition

To satisfy the boundary condition, we need the velocity on the surface. Since velocity is the gradient of potential (Equation 7), to find the velocity at \mathbf{P} , we take the gradient of $G(\mathbf{P}, \mathbf{Q})$ w.r.t \mathbf{P} .

$$\mathbf{u} = \nabla \phi. \quad (7)$$

With these equations, we set up a matrix of the influences of every source on every other source using Equation 6 giving us an $N \times N$ matrix where N is the number of panels. We factored out the source strength σ_j of each panel, giving us a vector of length N . And finally, we have our no-flow-through boundary condition, giving us a vector of length N , so we can set up a linear systems of equation.

$$\sum_{j=1}^N A_{ij} \sigma_j = \mathbf{V}_\infty \cdot \mathbf{n}_i \quad (8)$$

$$\begin{bmatrix} A_{11} & \cdots & A_{1N} \\ \vdots & \ddots & \vdots \\ A_{N1} & \cdots & A_{NN} \end{bmatrix} \begin{bmatrix} \sigma_1 \\ \vdots \\ \sigma_N \end{bmatrix} = \begin{bmatrix} \mathbf{V}_\infty \cdot \mathbf{n}_1 \\ \vdots \\ \mathbf{V}_\infty \cdot \mathbf{n}_N \end{bmatrix}$$

Once we solve the linear systems of equations for σ , we compute the velocity and coefficient of pressure:

$$\mathbf{V}(\mathbf{P}) = \mathbf{V}_\infty + \sum_{j=1}^N \sigma_j \nabla \phi_j(\mathbf{P}) \quad (9)$$

Compute the velocity vectors

$$C_p(\mathbf{P}) = 1 - \frac{|\mathbf{V}(\mathbf{P})|^2}{|\mathbf{V}_\infty|^2} \quad (10)$$

Compute the pressure coefficient

III. Conclusion

The panel method is a powerful tool for modeling fluid flow around a body using surface singularities. By solving the resulting system, we can find the fluid velocity and pressure distribution without running a full simulation. Understanding the derivation and implementation of the source-based panel method provides a strong foundation for future CFD projects in aviation, propulsion, HVAC, and other fields.

IV. References and Resources

References

- [1] NASA Glenn Research Center, “Wind Tunnel Testing of the Wright Brothers’ Airplane,” , 2025. URL <https://www.grc.nasa.gov/WWW/K-12/airplane/wrights/tunnel.html>, accessed: 2025-09-11.
- [2] ScienceDirect Topics, “Vortex Panel Method,” , 2025. URL <https://www.sciencedirect.com/topics/engineering/vortex-panel>, accessed: 2025-09-11.
- [3] Resolved Analytics, “History of Computational Fluid Dynamics (CFD),” , 2025. URL <https://www.resolvedanalytics.com/cfd/history-of-cfd>, accessed: 2025-09-11.