ParCFD'2019
31st International Conference on Parallel Computational Fluid Dynamics
May 14-17 2019, Antalya TURKEY

NOZZLE OPTIMIZATION USING ADJOINT-BASED OPTIMIZATION TOOL SU^2 – PARCFD'2019

BERKAN YERLİKAYA[1,2], ENGİN LEBLEBİCİ[2], LEVENT ÜNLÜSOY[2] AND İSMAİL HAKKI TUNCER[1]

[1] Middle East Technical University (METU)
Department of Aerospace Engineering
06800 Ankara, TURKEY
e-mail: berkan.yerlikaya@metu.edu.tr, e-mail: ismail.h.tuncer@ae.metu.edu.tr

[2] Roketsan Inc.
Analysis Technologies Unit
06780 Ankara, TURKEY
e-mail: engin.leblebici@roketsan.com.tr, e-mail: levent.unlusoy@roketsan.com.tr

Key words: Supersonic nozzle, Adjoint-based optimization, SU^2.

Summary: *Preliminary analysis for a generic nozzle geometry is done using an open-source computational fluid dynamics tool, SU^2. The generic nozzle geometry is planned to be used as baseline, in adjoint-based optimization procedure on the geometry to maximize thrust.*

1 INTRODUCTION

Nozzle is the exhaust system of a propulsion system. It completes the flow path and controls the expansion of high pressure and temperature gas mixture so that the flow exits in the axial direction. During the expansion, high internal energy of the flow transforms into kinetic energy[1].

A nozzle should smooth out the distortions generated within the flow and minimize stagnation pressure loss. Nozzle performance is based mainly on maximizing the generated thrust with minimum pressure loss. In order to supply maximum thrust for a desired design condition, a nozzle geometry optimization procedure emerges as a necessity. For this optimization, many design configurations should be explored that cannot be feasibly done solely with experiments. Therefore, computational fluid dynamics (CFD) appears as a solution to try many configurations practically.

Stanford University Unstructured (SU^2) software suite is chosen as a CFD tool to work with since it has features to construct adjoint-based optimization on Reynolds-averaged Navier Stokes (RANS) solver[2].

2 METHOD

In this work, a generic nozzle geometry is analyzed with the open-source CFD solver SU[2]. Results are compared with the experimental and numerical study by Olivera Kostic, Zoran Stefanovic and Ivan Kostic[3]. In the full paper, adjoint-based optimization on this nozzle geometry is planned to be done. Therefore, for the baseline geometry, this model is chosen as a start and validation case. The work done by Olivera Kostic, Zoran Stefanovic and Ivan Kostic is both experimental and numerical investigation of a supersonic converging diverging nozzle. The experimental work is performed in T-36 supersonic wind tunnel in Military Technical Institute VTI Žarkovo. The geometry of the experiment is represented in Figure 1.

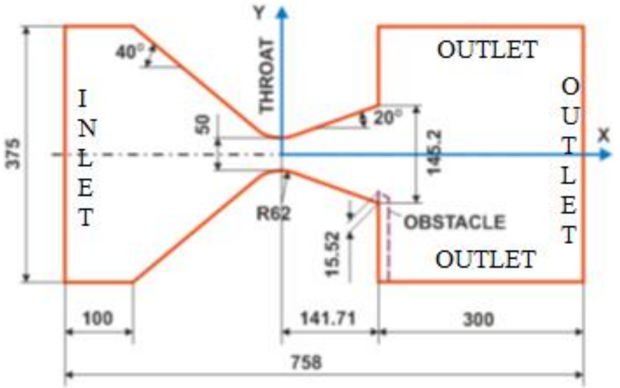

Figure 1: Baseline nozzle geometry[3]

In the experiments static pressure distribution on upper divergent wall is obtained and flow field is visualized with color Schlieren photography[3]. The working fluid is air and the boundary conditions for the CFD setup that mimics the experimental study from Kostic et Al[3], is given in the Table 1.

Boundary Conditions	Specified Boundary Properties
Inlet	Pressure = 101831.3Pa Temperature = 286.75 K
Outlet	Pressure = 500 Pa

Table 1: Boundary conditions[3]

RANS equations are solved with Jameson-Schimidt Turkel (JST) scheme as convective numerical method, using Green Gauss method for calculation of spatial gradients and first order numerical integration. Working fluid is modeled as an ideal gas with Sutherlands' Law for viscosity. Shear Stress Transport (SST) model is used for the turbulence closure. No-slip adiabatic wall boundary condition is applied on the nozzle wall. The inlet boundary condition is defined with the given temperature and pressure values. The outlet boundary condition is defined with the given back pressure value. The solution domain is discretized using unstructured grids. The grid consists of 406252 cells and the y+ value of the grid is below 1.

The generic nozzle geometry is planned to be used as baseline, in adjoint based optimization procedure. Adjoint-based techniques are appropriate tools for aerodynamic shape optimization because these techniques can provide sensitivity of an objective function for large number of parameters without repeating flow evaluations[4].

SU^2 optimization process mainly consists of surface deformation code which deforms the baseline geometry within the limitations of free form deformation (FFD) boxes, direct solvers which solves the Navier-Stokes/Euler equations for the problem to be investigated, adjoint solver which calculates sensitivities, gradient computation tool and the optimizer. The schematic of relations between the optimization components are given in Figure 2[5].

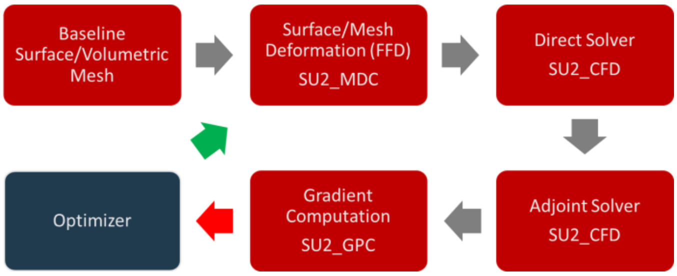

Figure 2: SU^2 optimization process[5]

Free form deformation (FFD) embeds the baseline geometry into parallelepiped lattice of control points. It assumes the geometry is made of clear rubber and this allows the desired location of an object to deform smoothly. Accordingly, surface continuity and volume of the baseline geometry is preserved. Therefore, it is easy to calculate analytic sensitivities derivatives and implement FFD to gradient based optimization[6].

3 PRELIMINARY WORK

As a preliminary work, , CFD solution is obtained with SU^2 on the nozzle geometry given in Figure 1. The grid given in Figure 3 is used for the solution domain.

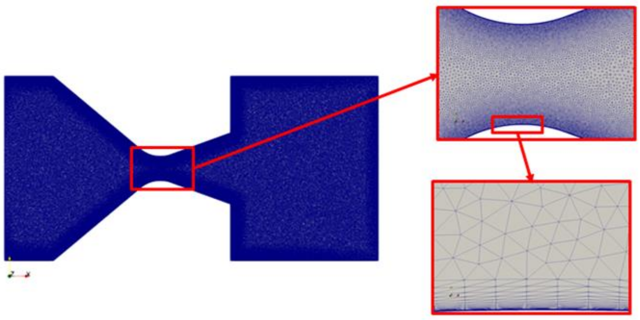

Figure 3: Computational grid

Static pressure measurements from the experiment through the upper divergent wall is compared with the numerical results of SU^2 in the Figure 4.

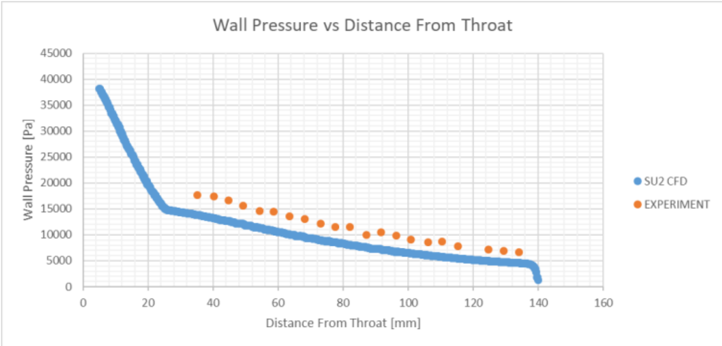

Figure 4: Upper diverging wall pressure distribution comparison[3]

As it can be seen from the graph SU^2 and the experimental results have the same trend through the upper diverging wall in terms of static pressure distribution. Also, Mach number contours from the SU^2 solutions are compared with CFD solutions by Kostic et Al[3] in Figure 5.

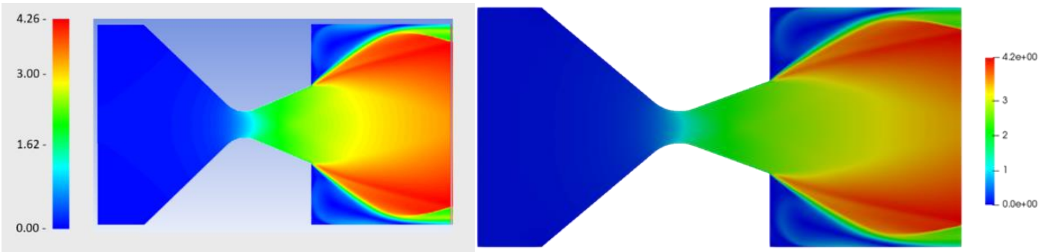

Figure 5: Mach number contour comparison[3]

The Mach number contours and máximum values of both SU^2 and numerical solutions done by Kostic et Al[3] are nearly identical.

REFERENCES

[1] Flack, R. D. (2010). Fundamentals of jet propulsion with applications. New York: Cambridge University Press.

[2] Palacios, F., Economon, T. D., Aranake, A., Copeland, S. R., Lonkar, A. K., Lukaczyk, T. W., . . . Alonso, J. J. (2014). Stanford University Unstructured (SU2): Analysis and Design Technology for Turbulent Flows. 52nd Aerospace Sciences Meeting. doi:10.2514/6.2014-0243

[3] Kostic, O., Stefanovic, Z., & Kostic, I. (2015). CFD modeling of supersonic airflow generated by 2D nozzle with and without an obstacle at the exit section. FME Transaction, 43(2), 107-113. doi:10.5937/fmet1502107k

[4] Taylor, T. W., Alonso, J. J., Palacios, F., Duraisamy, K., Iaccarino, G., & Jameson, A. (2013). A hybrid adjoint approach for systems of arbitrarily complex partial differential equations (Unpublished master's thesis).

[5] Alonso, J. J., Palacios, F., Economon, T., Colonno, M., & Copeland, S. (n.d.). High-Fidelity Aerodynamic Shape Optimization Using SU2. Lecture presented at 2013 MDO Consortium Workshop in Stanford University, Stanford, CA

[6] Hradil, J. (2015). Adaptive Parameterization For Aerodynamic Shape Optimization In Aeronautical