CFD simulation on the J–2X engine exhaust in the center-body diffuser and spray chamber at the B–2 facility

Xiao-Yen Wang,* Thomas Wey, and Robert Buehrle
NASA Glenn Research Center
Cleveland, Ohio 44135
* xiao-yen.j.wang@nasa.gov

Abstract

A computational fluid dynamic (CFD) code is used to simulate the J–2X engine exhaust in the center-body diffuser and spray chamber at the Spacecraft Propulsion Facility (B–2). The CFD code is named as the space-time conservation element and solution element (CESE) Euler solver [1] and is very robust at shock capturing. The CESE results are compared with independent analysis results obtained by using the National Combustion Code (NCC) [2] and show excellent agreement.

1. Introduction

The B–2 in the Plum Brook Station (PBS) was originally designed to test full-scale upper-stage rockets up to 100,000 lbf thrust in a simulated space environment. Since most rocket engines that have been tested in the B–2 were in the 30,000 lbf thrust class, the B–2 must be adapted to accommodate the engines with much larger thrust such as the J–2X engine that produces 294,000 lbf thrust, which results in a more severe thermal environment and a larger scale of energy.

A sketch of B-2 facility is shown in Fig. 1. The J-2X engine exhaust was directed into the center-body diffuser to slow down before hitting the top surface of the water tank at the bottom of the spray chamber. When the rocket engine is operating, the water is injected inside the spray chamber to cool down the hot exhaust gas. The mixture of water vapor and hot gas will vent through the ejector when the spray chamber pressure is high enough. The steam is sprayed through the steam blocker to prevent the back flow in the event of J-2X engine shutdown. A CFD code is used to simulate how the J–2X engine exhaust expands through the center-body diffuser and into the spray chamber, then vents to outside the chamber through the ejector. The water spray inside the spray chamber is not modeled here. The two-dimensional/axisymetric CESE Euler code is used here. In the following, the J–2X engine performance is described first, which is followed by the CFD results and validations.

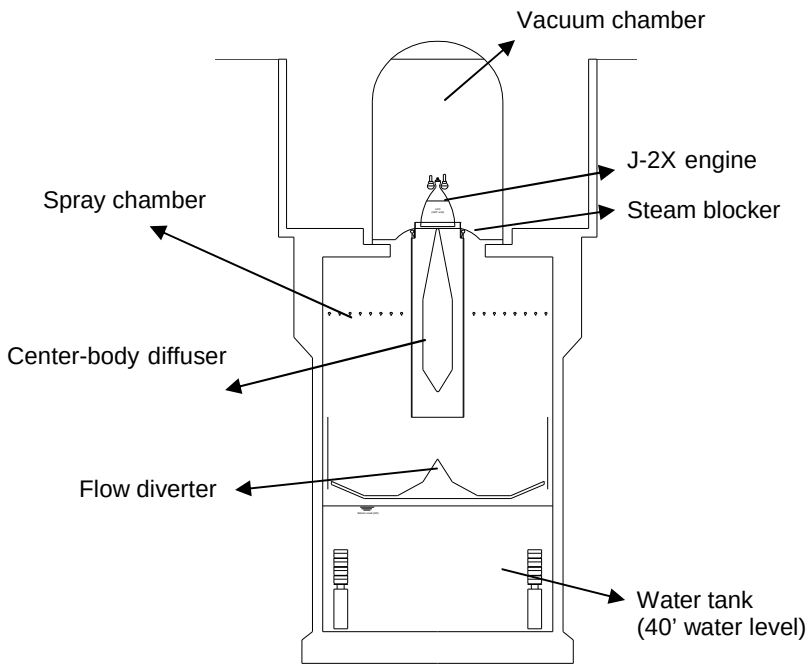


Fig. 1 A sketch of the B-2 facility for J-2X engine testing.

2. J–2X engine performance

The J–2X engine uses liquid oxygen (O_2) and hydrogen (H_2) as a propellant with an oxidizer to fuel (o/f) ratio of 5.5 at the chamber pressure of 1,338 psia to produce 294,000 lbf of thrust. The chemical equilibrium compositions and applications (CEA) code is used to compute the performance of the J–2X engine. In table 1, the CEA results of the pressure (*p*), temperature (*T*), density (ρ), mole weight, ratio of specific heat (γ), sonic velocity, Mach number (*Ma*), and mole fractions are listed for different locations inside the J-2X engine.

Table 1. J–2X engine performance (CEA results)

	Combustor end	Throat	Exit
p, BAR	85.62	51.066	0.0592
T, K	3406.75	3210.22	1005.01
ρ, kg/m^3	3.8389	2.4501	9.28E-03
Mole weight, (1/n)	12.7	12.806	13.103
Cp, kJ/kg-K	7.52	6.81	2.895
γ	1.15	1.15	1.28
Sonic velocity, m/s	1601.3	1549.8	903.7
Ma	0.26	1	4.92
Mole fractions:			
H_2	0.301	0.301	0.307
H_2O	0.64	0.655	0.693

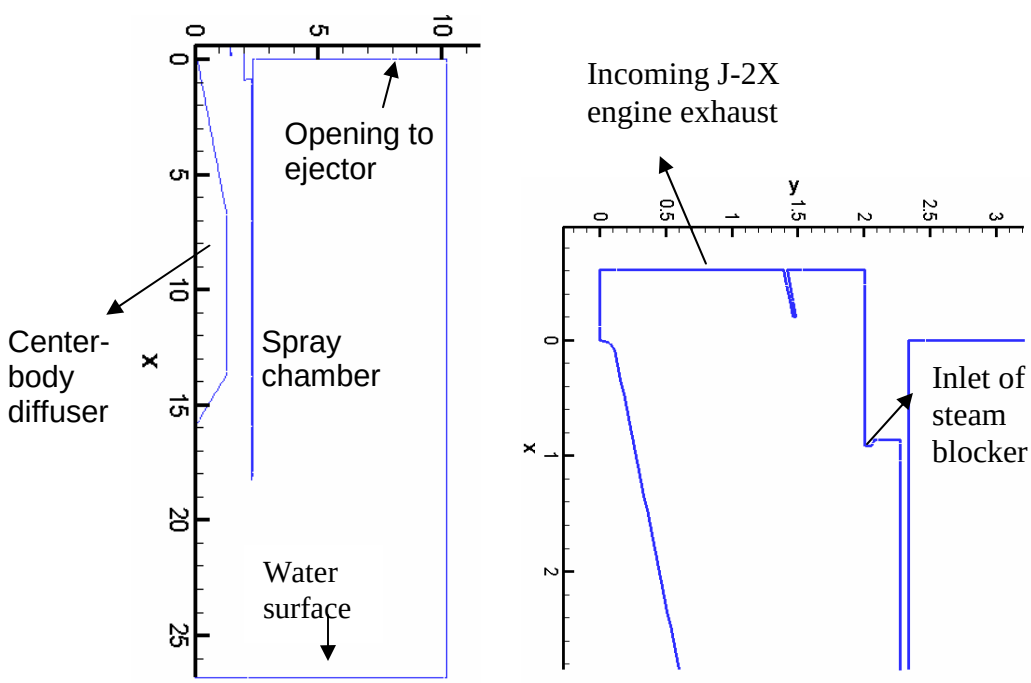
3. CFD simulation on the J–2X engine exhaust

It was assumed that the hot gas is an ideal gas. The hot gas properties at the engine nozzle exit obtained from CEA is used in the CFD simulation. The CESE two-dimensional/ axisymmetric Euler code is used and a finite-element mesh is generated using MSC Patran. In the analysis, the flow variables are nondimensionalized by using those at the engine nozzle exit as follows:

$$p^* = p/\rho_e u_e^2, \quad \rho^* = \rho/\rho_e$$
$$x^* = x/L, \quad T^* = T/T_e$$
$$u^* = u/u_e, \quad t^* = t/(L/u_e)$$

where $\rho_e = 9.94e-3\,\text{kg/m}^3$, $u_e = 4{,}446.2\,\text{m/s}$, and $T_e = 1{,}005\,\text{K}$ are the density, velocity, and temperature, and

$$L = 1\,\text{m} \text{ and } \rho_e u_e^2 = 1.965\text{e}+5\,\text{N/m}^2 = 28.5\,\text{psia}.$$



(a) the computational domain (b) close view of the domain inlet

Fig.2 The computational domain used in the analysis.

The computational domain that has 18,119 mesh points and 35,332 triangular elements is shown in Fig. 2. The water surface is approximated by using a solid wall. For the core flow, it was assumed that the total pressure p_t = 1,338 psia and the total temperature T_t = 3,552 K (5,934 °F) with a mass flow rate of 650 lbm/s. For the steam blocker, p_t = 165 psia, T_t = 459 K (366 °F) with a mass flow rate of 147 lbm/s. The initial conditions at t = 0 (B-2 evacuated conditions) are defined as

$$p^* = 0.0056\ (p = 0.16\,\text{psia}), u^* = 0, v^* = 0, \rho^* = 0.8047$$

At the inlet of the computational domain (exit of the engine nozzle)

$$p^* = 0.0368, u^* = 0.9957, v^* = 0,\ \ \rho^* = 1.1092$$

At the inlet of steam blocker

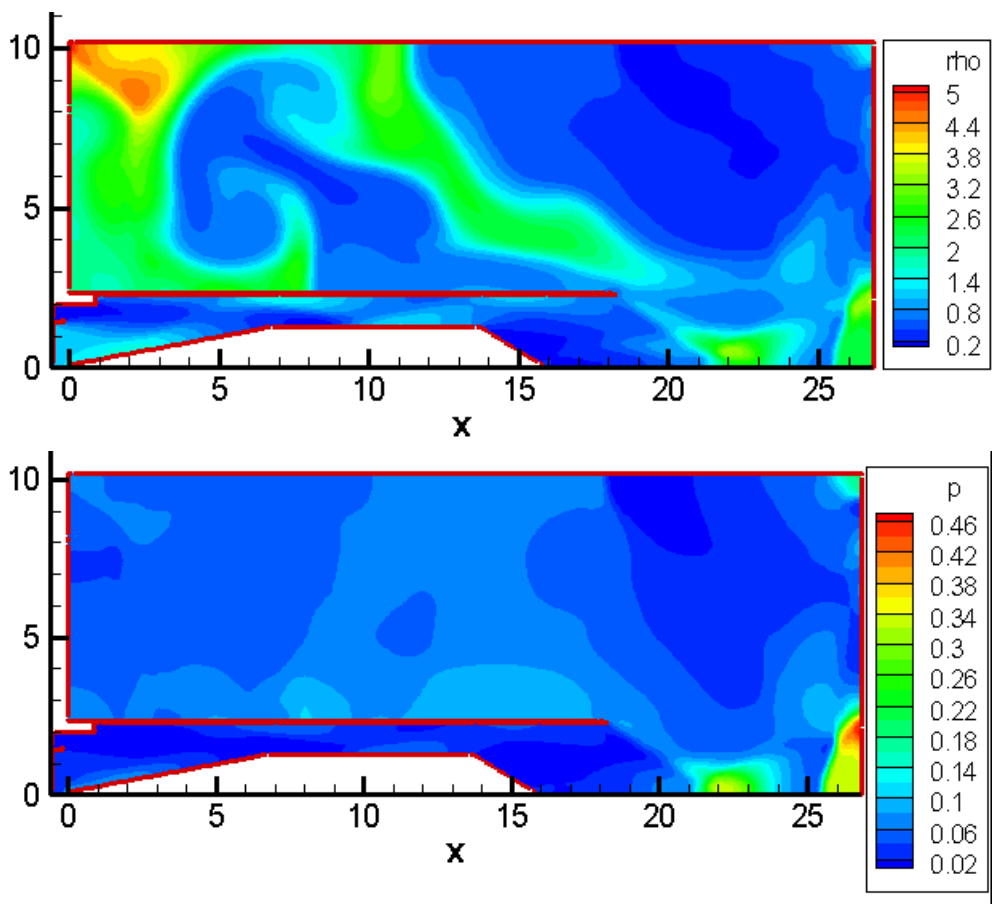
$$p^* = 0.00723, u^* = 0.7028, v^* = 0, \rho^* = 0.7313$$

At the opening to the ejector

$$p^*_{back} = 0.014\ (p = 0.4\,\text{psia})$$

The computed CESE results of the non-dimensional density, pressure, temperature, Mach number, and velocity vector at t = 0.0787 s are plotted in Fig. 3. It can be seen that complex shock waves exist inside both the diffuser and the spray chamber. In the center-body diffuser, a series of oblique shock waves start at the exit of the engine nozzle (inlet of the computational domain) and keep reflecting along the wall. The flow field inside the diffuser reaches steady state within 0.0787 s. The flow at the exit of the diffuser is still supersonic. The shock waves in the spray chamber still bounce back and forth along the chamber wall and water surface.

Further, the flow field inside the center-body diffuser is compared between CESE results and those computed independently using the NCC. The details of the NCC simulation will be given in a separate paper and will not be described here. It can be seen that the wave patterns captured in the NCC and CESE codes are very similar as shown in Fig. 4.



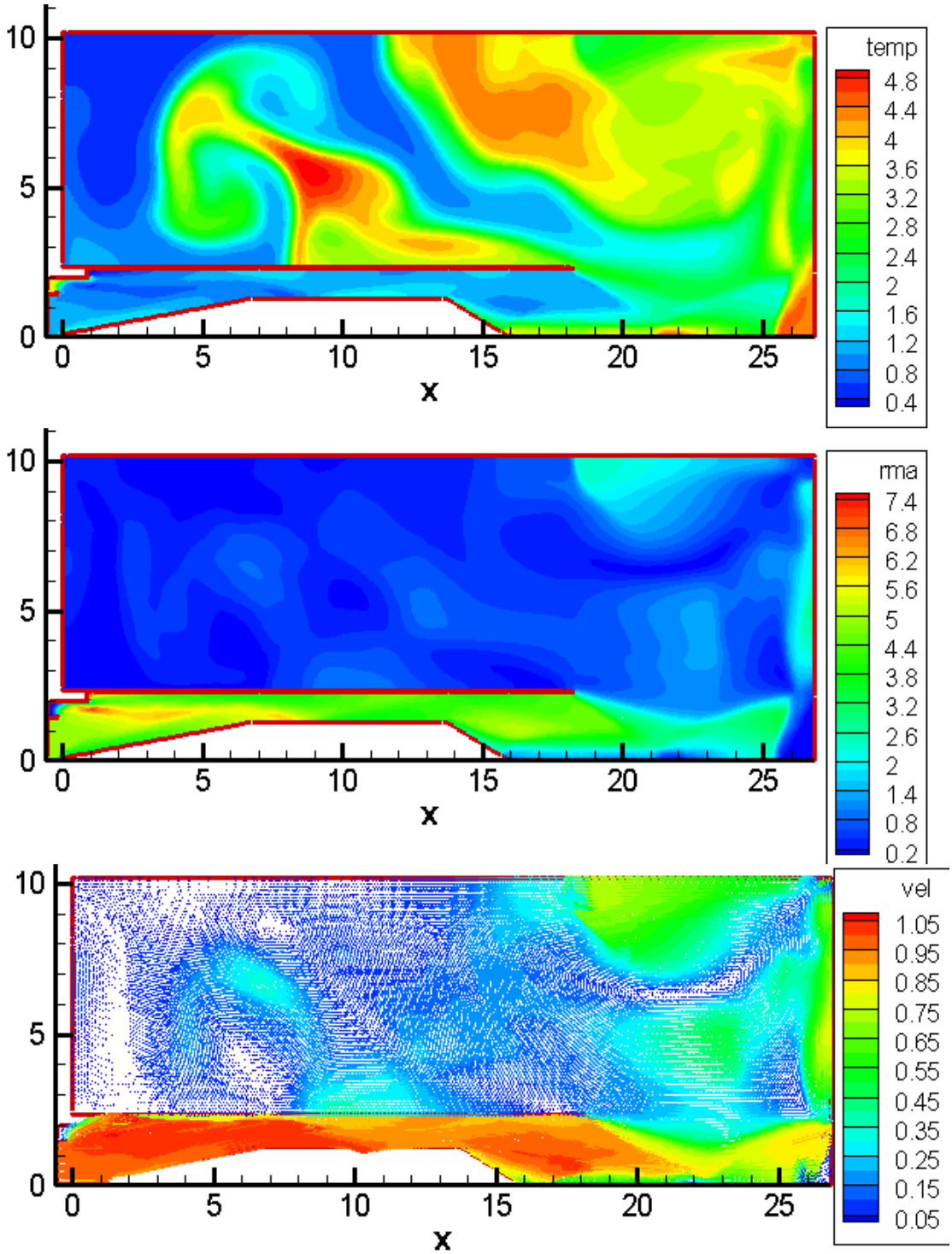


Fig. 3 CESE results at t = 0.0787 s.

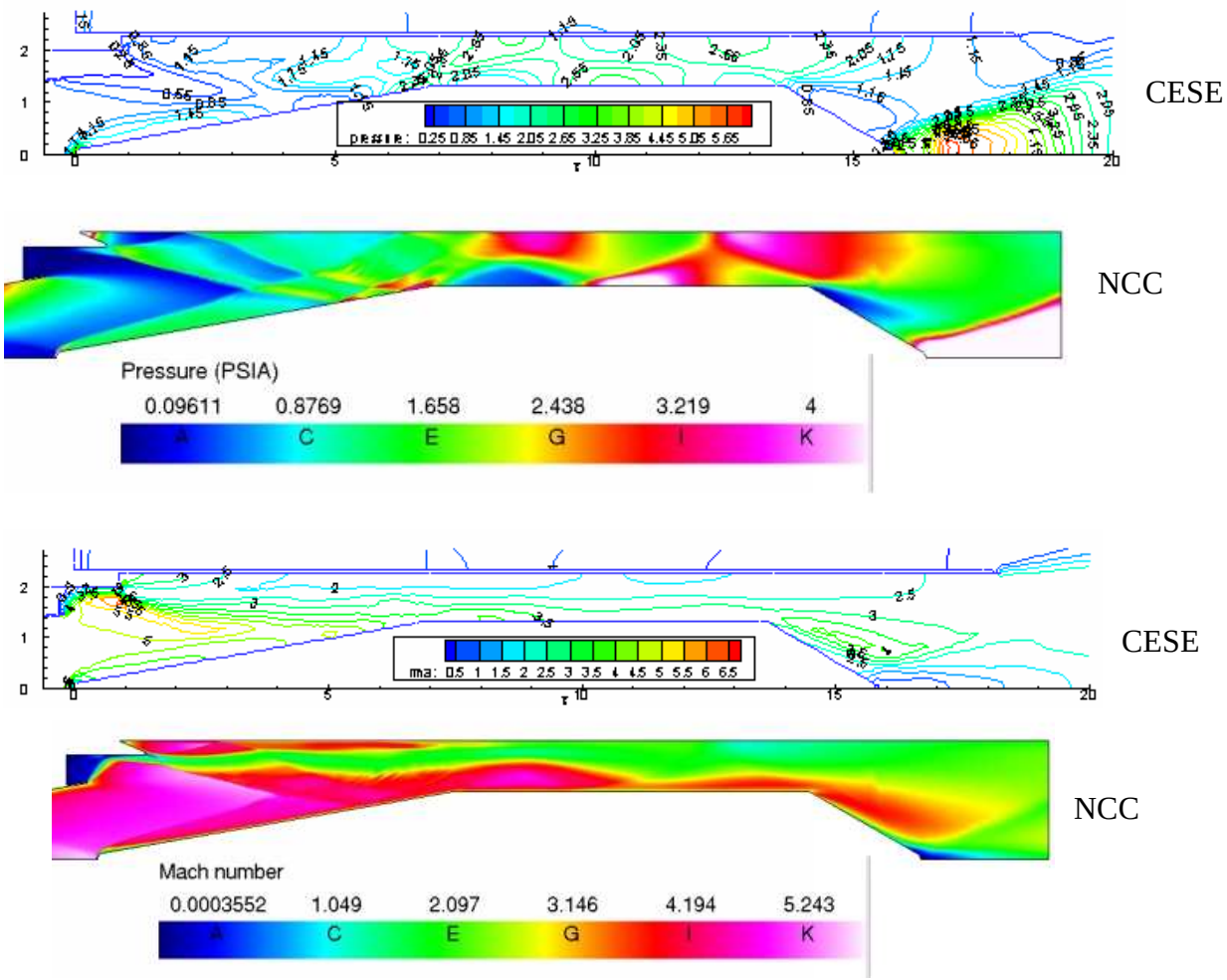


Fig. 4 The comparison between CESE and NCC results inside the center-body diffuser.

3. Conclusions

The J–2X engine exhaust in the center-body diffuser and spray chamber at the Spacecraft Propulsion Facility (B2) is simulated using the CESE method. The shock wave pattern was captured by the CESE method and agrees well with the corresponding results obtained by using the NCC. Further analysis is needed to validate the design of the B–2 for testing rocket engines with up to 300,000 lbf thrust.

Acknowledgments

The first author would like to thank Kevin Dickens and Daryl Edwards for their valuable input to this work.

References

1. X.Y. Wang, “Space-time CESE Euler Solver 6.0, User’s Guide”, July, 2001.
2. R. M. Stubbs and N. S. Liu, "Preview of National Combustion Code," AIAA 97-3114, 1997.