Simulation of flow through Supersonic Cruise Nozzle: A validation study

Balasubramanyam Sasanapuri
Manish Kumar & Sutikno Wirogo
ANSYS Inc.

Presented By
Reza Ghias

Thermal & Fluids Analysis Workshop
TFAWS 2011
August 15-19, 2011
NASA Langley Research Center
Newport News, VA

Outline

- Problem Description
- Results Requested
- Reference
- Model and Flow Conditions
- Nozzle Parameters
- Test Matrix
- CFD Study
- Results and Conclusion

Problem Description

- Problem Statement
  - Simulate flow in a supersonic cruise nozzle
- Objectives
  - Compare ANSYS CFD predictions with the wind tunnel results presented in NASA TP-1953
  - Compare density-based and pressure-based solvers
  - Compare the effects of grid adaption on the solution

Expected Results

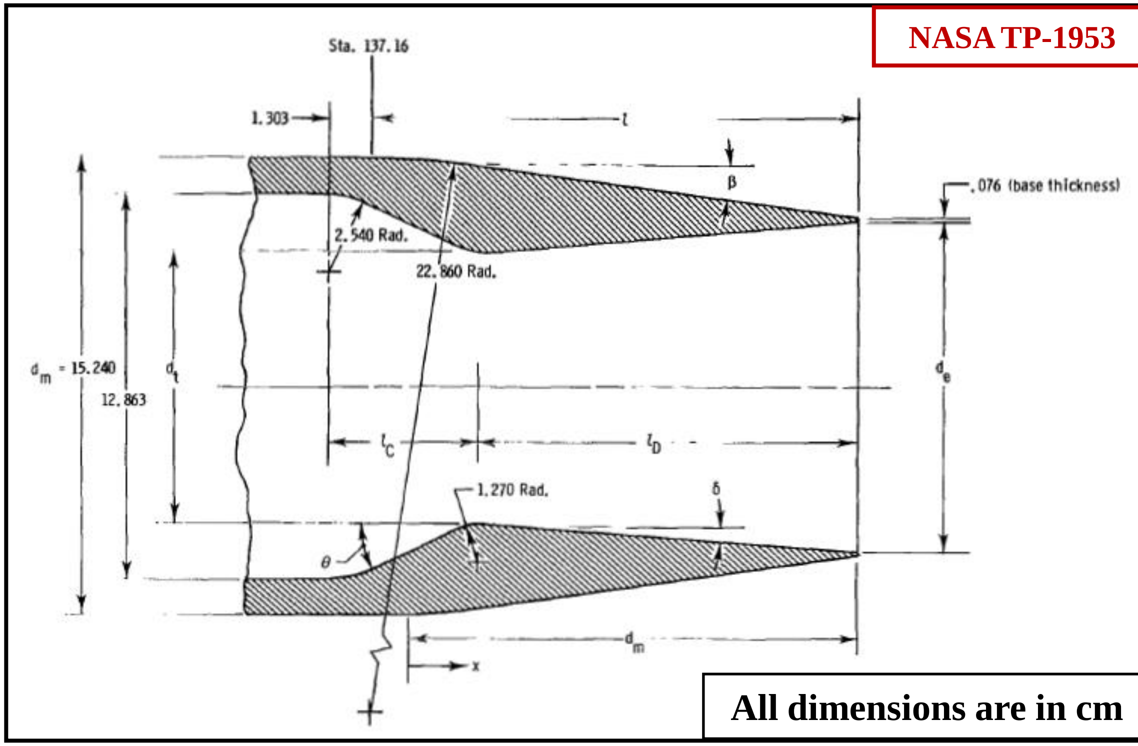
- Contours of Mach Number and Pressure
- Comparisons of
  - Discharge coefficient
  - Thrust parameter

Reference

- Experimental data from reference NASA TP-1953
  - Simulation of a supersonic aircraft's operation over a wide altitude-velocity flight envelope
  - Angle of attack: 0°, Free Stream Mach: 0.60 to 1.30
  - Five different axisymmetric convergent-divergent nozzles tested
    - Different internal and external geometries representing the variable-geometry nozzle operating over a range of engine operating conditions
- **Configuration 2 (supersonic cruise nozzle) was selected for the present study**

Model and Flow Conditions

- Supersonic Cruise Nozzle
- Data from NASA TP-1953



Configuration	Flight segment	Power setting	$(p_{t,j}/p_\infty)_{des}$	Design dimensions										
				A_e/A_t	A_t/A_m	A_e/A_m	d_t/d_m	l_C/d_m	l_D/d_m	θ, deg	δ, deg	l/d_m	d_e/d_m	β, deg
1	Subsonic cruise	Dry	4.25	1.250	0.250	0.312	0.500	0.286	0.800	42.35	2.12	1.000	0.559	15.05
2	Supersonic cruise	Dry	21.23	3.000	.250	.750	.500	.286	.779	42.35	13.18	.979	.866	3.82

Nozzle Parameters

- Static discharge coefficient, C_d

$$\boxed{C_d = \dot{m}/\dot{m}_i}$$

Isentropic Mass Flow Rate

$$\boxed{\dot{m}_i = PA_t\sqrt{\frac{\gamma}{RT}\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{\gamma-1}}}}$$

$\dot{m}$ Mass flow Rate from CFD/Experiment at nozzle exit,

P Total Pressure at Nozzle Inlet

T Total temperature at Nozzle Inlet

p_∞ Ambient pressure

A_t Throat Area

- Nozzle thrust performance, Cfg

$$\boxed{Cfg = F_j / F_i}$$

Isentropic Thrust

$$\boxed{F_i = \dot{m}_i\sqrt{\frac{2\gamma}{\gamma-1}RT\left[1-\left(\frac{p_\infty}{P}\right)^{\frac{\gamma-1}{\gamma}}\right]}}$$

F_j Thrust from CFD/Experiment

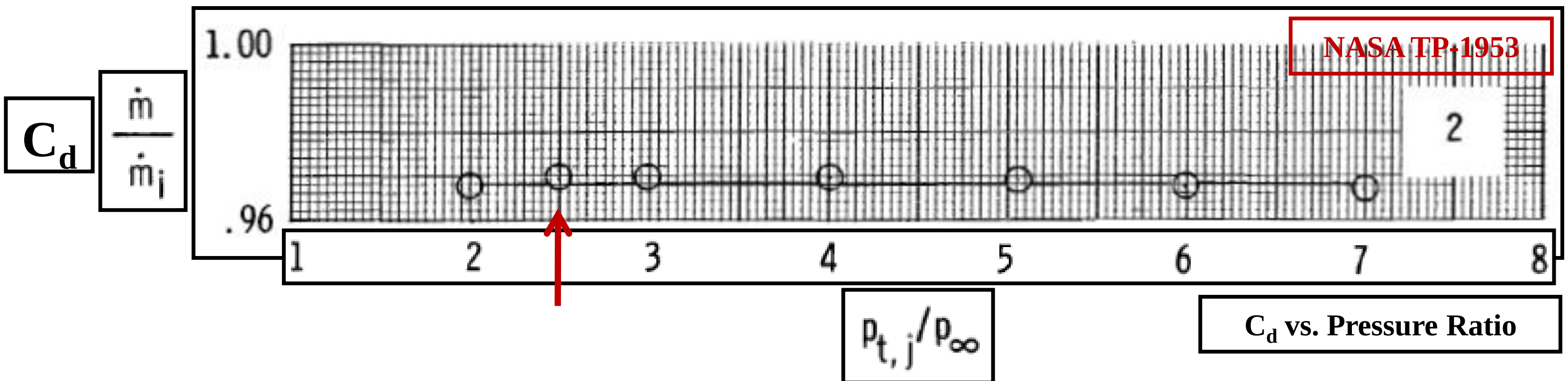
$$F_j = \dot{m}V_e + (p_e - p_\infty)A_e$$

p_e Area-averaged pressure at Exit

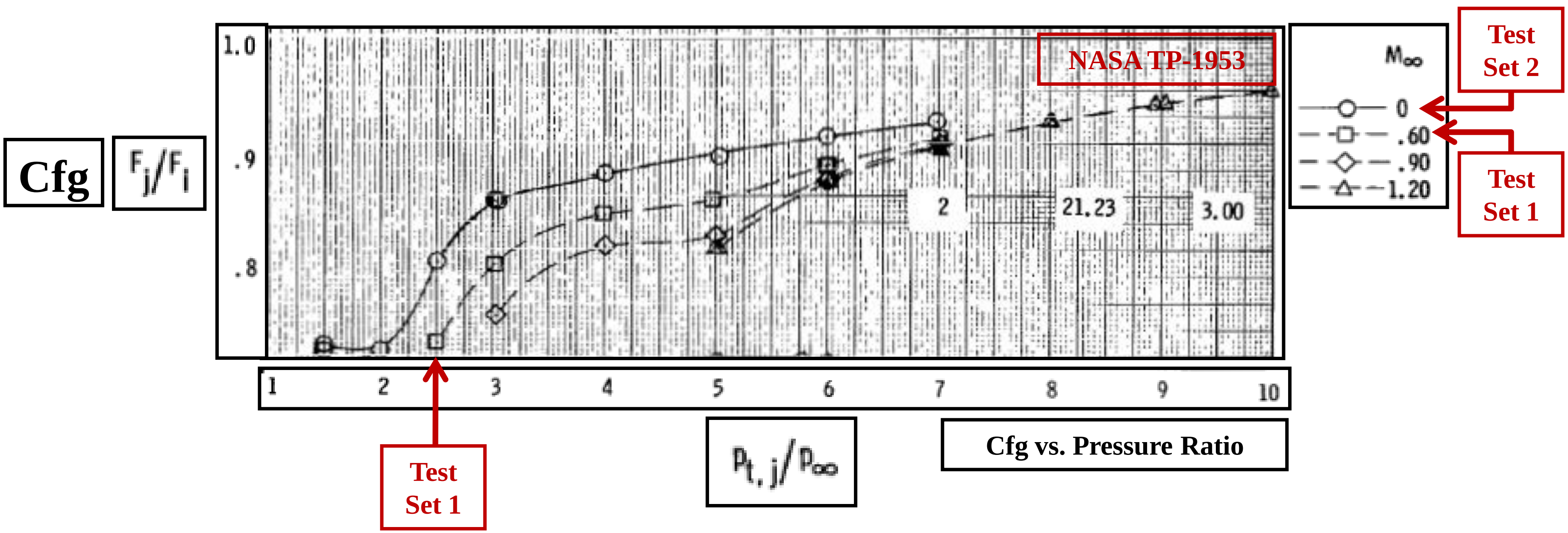
A_e Exit Area

V_e Mass-Averaged velocity at Exit

Nozzle Parameters – Experimental Data



$p_{t,j}$: Pressure at nozzle inlet (P)

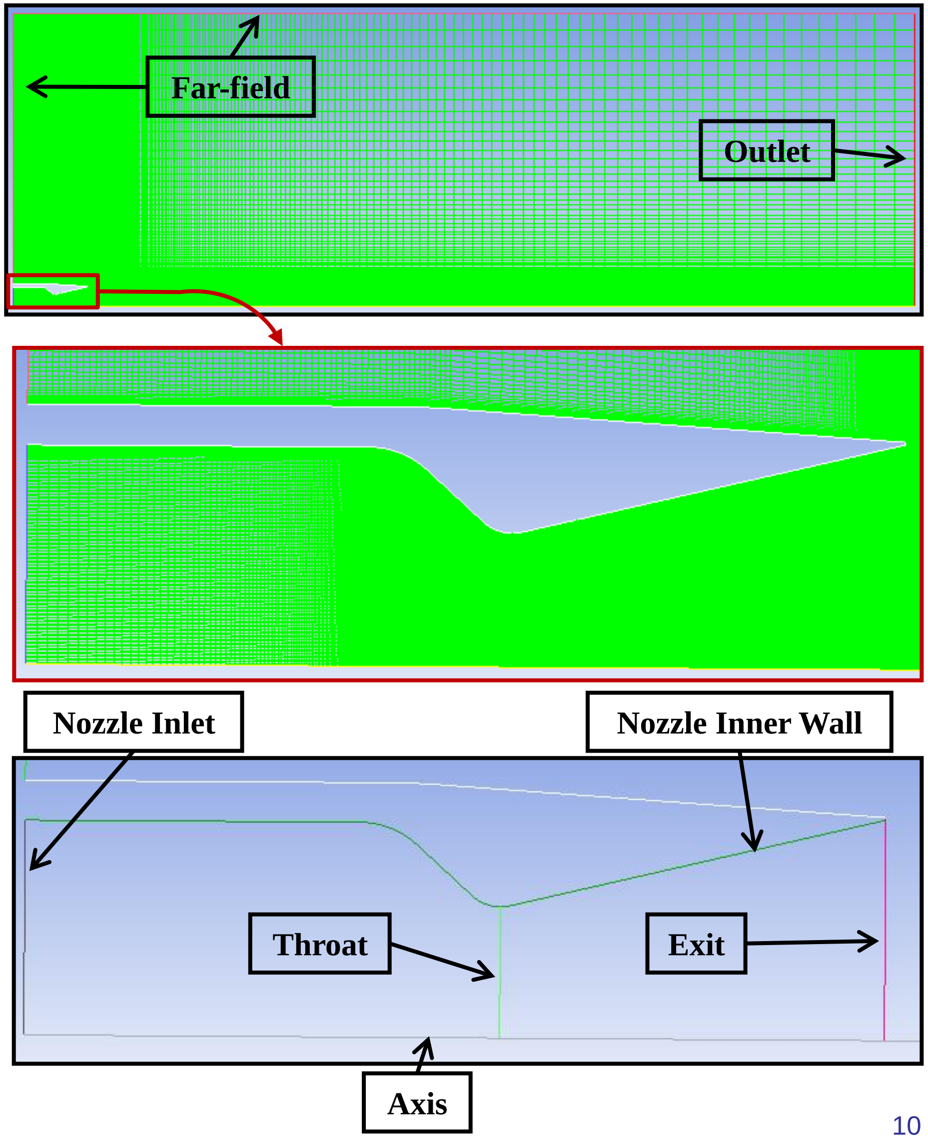


Test Matrix

- Test Set 1
  - M = 0.6, Nozzle Pressure Ratio (NPR) = 2.5
  - Comparison of various solver schemes
    - Pressure Based Coupled Solver (PBCS)
      - 2^{nd} Order discretization
      - PRESTO, QUICK discretization
    - Density Based Navier-Stokes Solver (DBNS)
  - Effect of Mesh Adaption
- Test Set 2
  - M = 0
  - NPR = 2.5, 4.0, 5.0, 6.0, 7.0
  - Best solver settings from Test Set 1

CFD Study : Mesh and Boundary Conditions

- 2-D axisymmetric flow domain
  - Nozzle Exit Diameter = 0.132 m
  - Domain length = 3.1 m
  - Domain height = 1.0 m
- Total no. of cells ~ 359 K
- Interior boundaries at nozzle exit and throat for post-processing
- Boundary Conditions:
  - Outlet at ambient condition
  - Test Set 1
    - Far-field Mach number = 0.6
    - Nozzle Inlet, P = 2.5 atm; T = 300 K
  - Test Set 2
    - Far-field Mach Number = 0.0
    - Nozzle Inlet, T = 300 K, P = Various



CFD Study : Solver Settings

- Various solver parameters were tested
  - Pressure Based Coupled Solver (PBCS)
    - 2nd Order for all equations
    - PRESTO for Pressure, QUICK for other equations
  - Density Based Solver (DBNS)
    - 2nd Order for all equations
- Mesh Adaption
  - Performed using Blast Wave Identification Parameter (BWIP) scheme
  - Results compared for all schemes pre- and post-adaption
- k-ω SST turbulence model ($y^+ \sim 1$)

CFD Study : Choice of Solvers

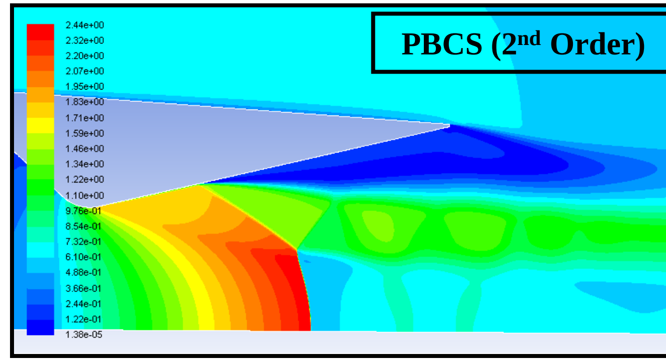
- Density Based Coupled Solver (DBNS)
  - High speed external flow(supersonic and hypersonic regime)
    - Sharp shock structures
  - Less efficient for resolving large low-speed circulating wake
  - Less efficient for internal flow and heat transfer cases
- Pressure Based Coupled Solver (PBCS)
  - Subsonic, transonic, and mild supersonic external flows
    - Smearing of shocks clearly visible
  - Efficient in resolving large circulating wake and internal flow
  - It is not the segregated pressure based solver
  - Very fast and less memory requirement

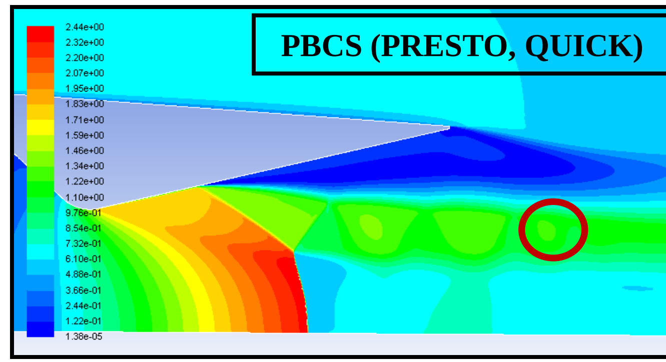
Test Set 1 : Flow Characteristics

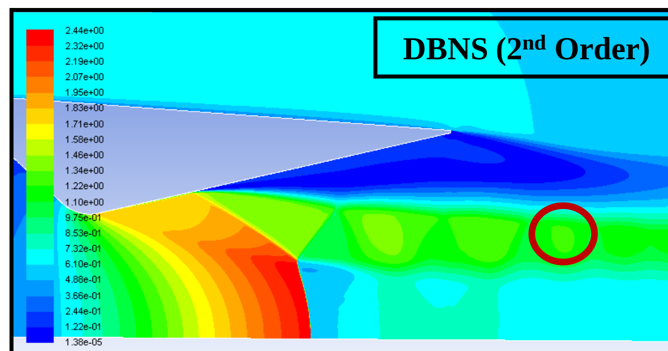
- Over-expanded nozzle (Pressure at the nozzle exit ~ 92000 Pa) : Jet contracting at the exit

- Mixing of subsonic and supersonic flow at the exit
  - Shock diamonds are formed
  - Oscillatory flow at the nozzle exit

- DBNS (2nd Order) and PBCS (PRESTO, QUICK) capture shock diamond effect better than PBCS (2nd Order)



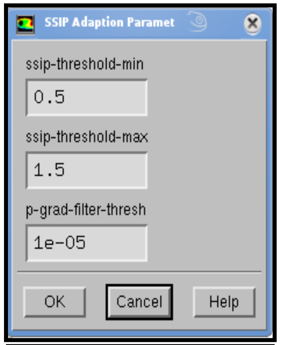




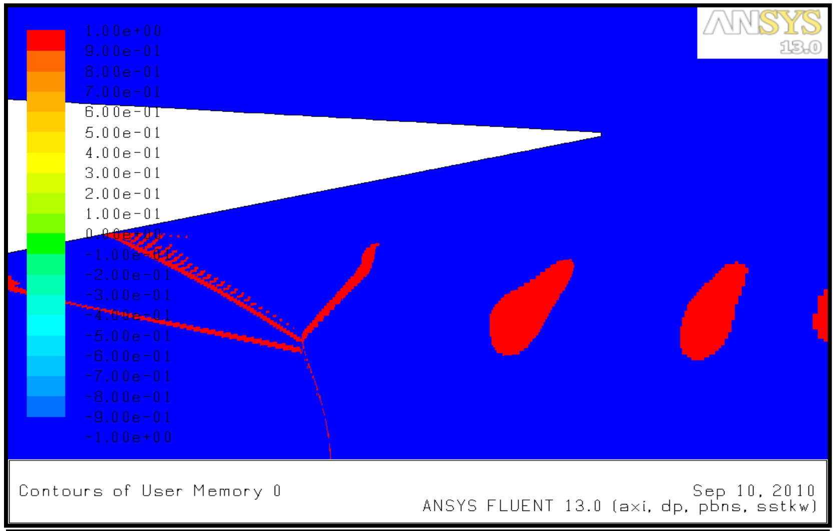
Contours of Mach Number

Test Set 1 : Mesh Adaption

- Mesh Adaption
  - For oscillating solutions adaption performed when the solution reached mid harmonic
  - Using BWIP (Blast Wave Identification Parameter) scheme
  - Adaption near the shocks only
  - Adaption did not increase the number of cells at the nozzle wall



BWIP



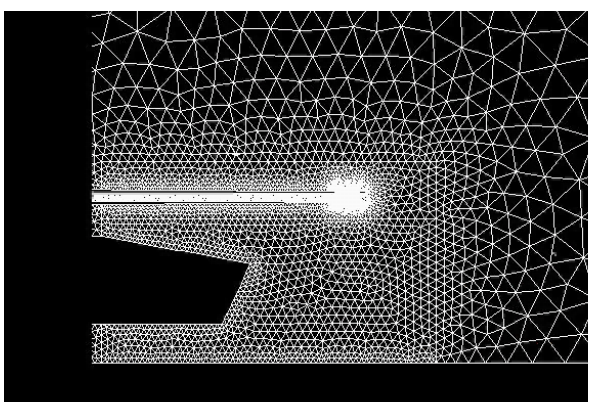
Regions to be adapted

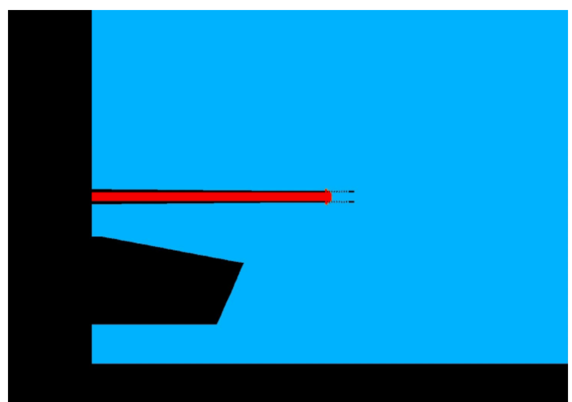
Test Set 1 : Mesh Adaption Continued…

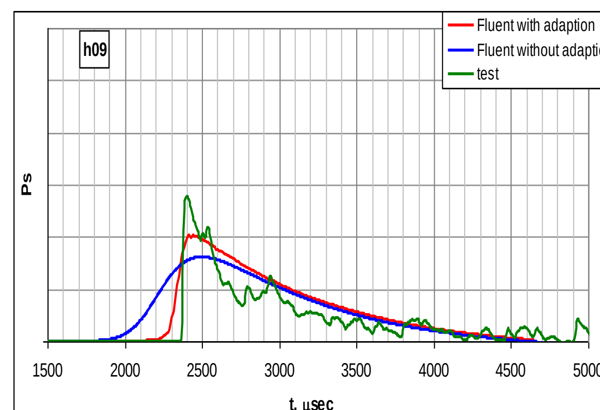
- **Blast Wave Identification Parameter (BWIP)**
  - Collaboration with Benet Weapon Lab
  - Specially formulated for stationary and moving shocks

$$\boxed{f_{BWIP} = -a\frac{1}{|\nabla p|}\nabla\cdot(\rho\vec{u}) + \frac{\vec{M}\cdot\nabla p}{|\nabla p|}}$$

  - Refine the cells where $L_1 \le f_{BWIP} \le L_2$

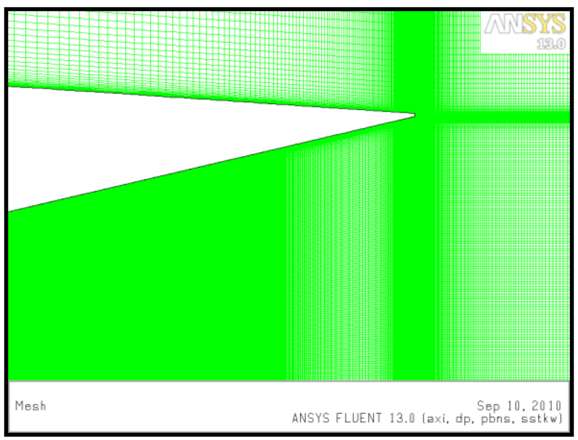




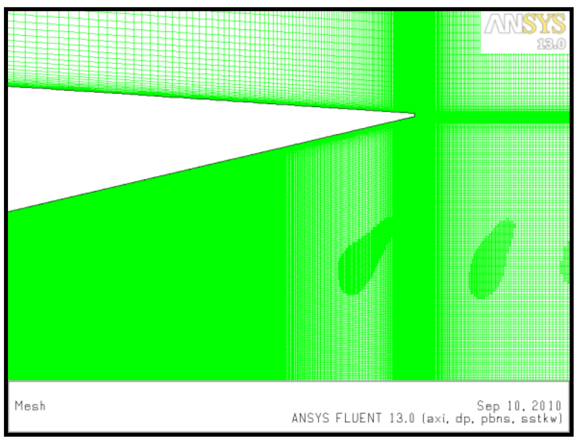


Test Set 1 : Mesh Adaption Continued…

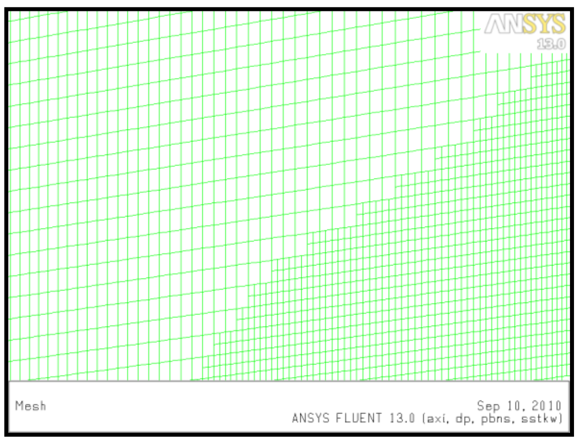
- Mesh Adaption
  - Adaption 1: Cell count changed from 359,100 to 388,788
    - Used with all schemes
  - Adaption 2: Cell count changed from 388,788 to 481,122
    - Used with PBCS with PRESTO & QUICK



Mesh before adaption



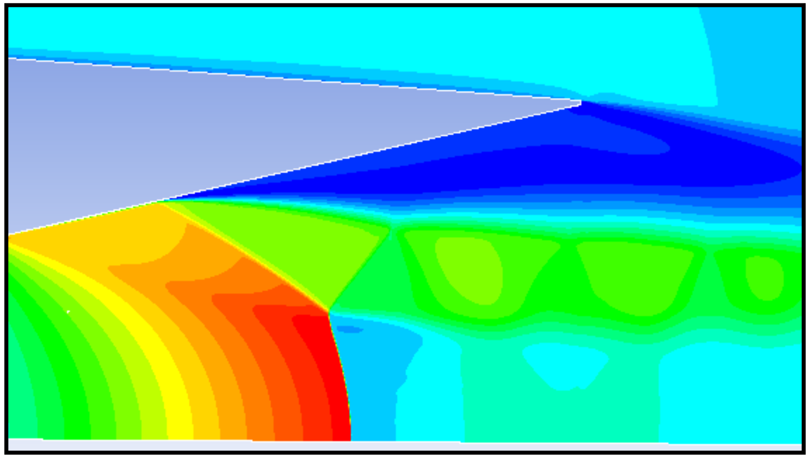
Mesh after adaption



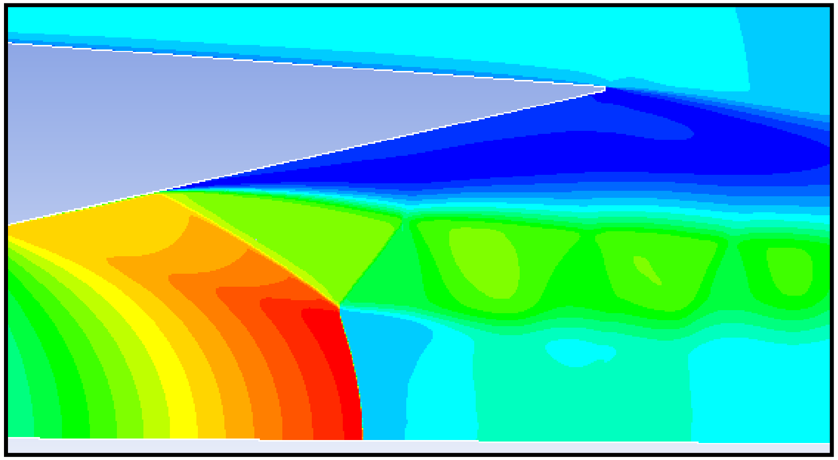
Mesh after adaption : Close-up

Test Set 1 : Mesh Adaption Continued…

- Effect of Mesh Adaption on Velocity Contours
  - Mesh adaption only leads to small changes in the velocity
  - Pressure is also only slightly affected by adaption (not shown here)
  - Similar behavior seen for PBCS solvers with both (2nd Order) and (PRESTO, QUICK) discretizations

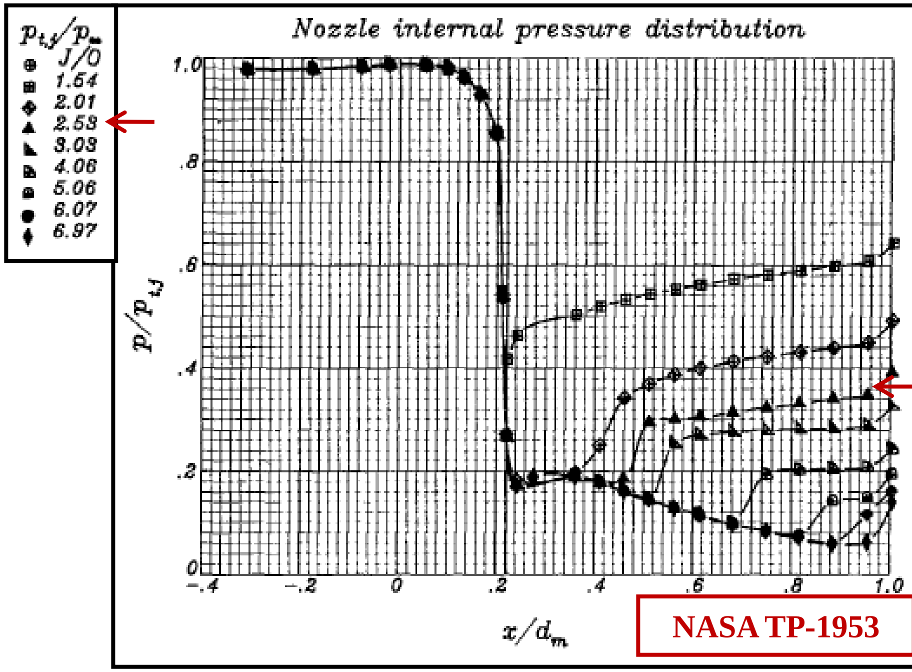


Mach Contours (DBNS) - Before Adaption

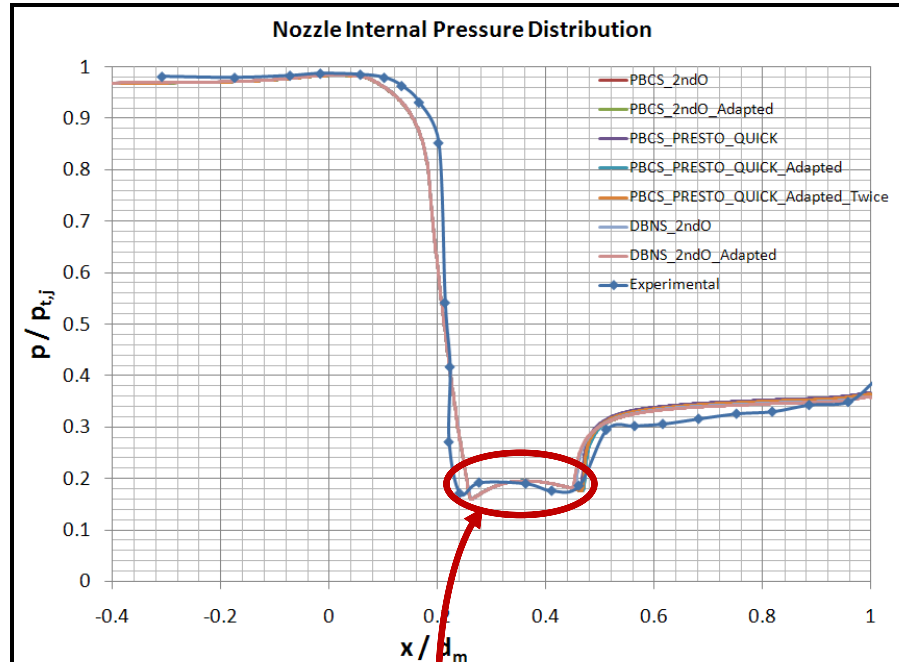


Mach Contours (DBNS) - After Adaption

Test Set 1 : Nozzle Internal Pressure Distribution

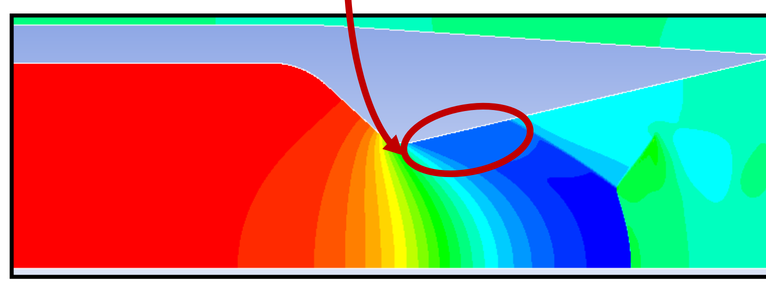


Experimental Data



CFD Results

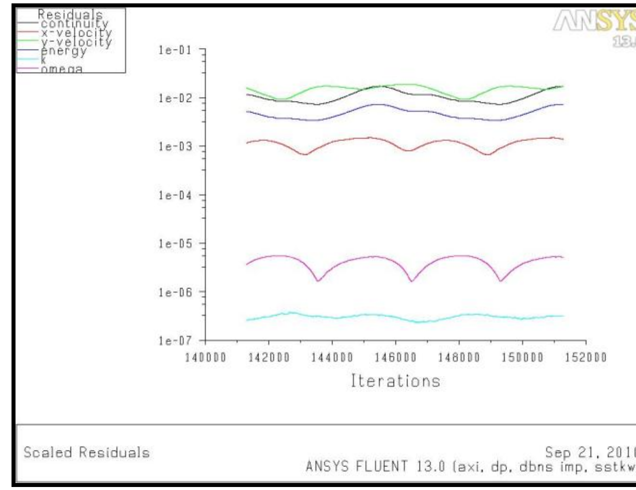
- Pressure distribution at the nozzle internal wall is captured quite well
- Results from all solver schemes are overlapping



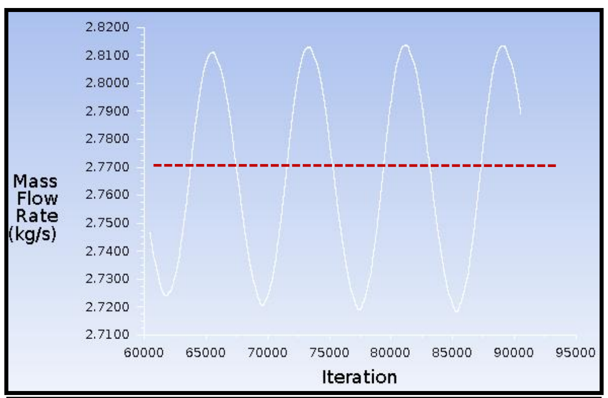
Contours of Pressure

Nozzle Parameters – Calculation from CFD

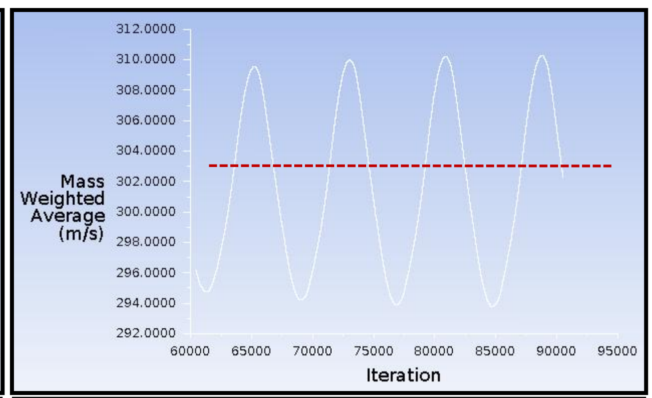
- Mass flow rate at the nozzle exit is calculated as the mean value
- Static pressure at the nozzle exit is calculated as the mean of its area-average
- Velocity at the nozzle exit is calculated as the mean of its mass-average
- Mean values used to calculate nozzle thrust



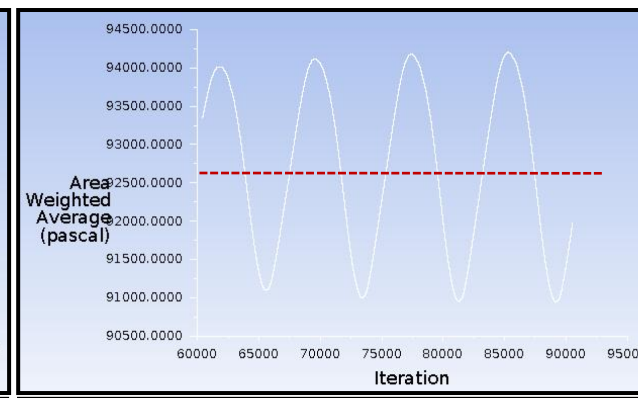
Residuals



Mass Flow Rate



Mass Weighted Velocity



Area Weighted Pressure

Test Set 1 : Nozzle Parameters

	C_d (Exp)	C_d (CFD)	C_d (Error %)	Cfg (Exp)	Cfg (CFD)	Cfg (Error %)
PBCS (2nd Order)	0.97	0.9651	-0.502	0.71	0.6519	-8.180
PBCS (2nd Order) Adapted	0.97	0.9653	-0.485	0.71	0.6531	-8.013
PBCS (PRESTO, QUICK)	0.97	0.9651	-0.505	0.71	0.6609	-6.921
PBCS (PRESTO, QUICK), Adapted	0.97	0.9648	-0.534	0.71	0.6609	-6.910
PBCS (PRESTO, QUICK), Adapted twice	0.97	0.9651	-0.506	0.71	0.6618	-6.793
DBNS	0.97	0.9648	-0.534	0.71	0.6700	-5.635
DBNS, Adapted	0.97	0.9649	-0.528	0.71	0.6701	-5.624

Test Set 1 : Conclusions

- Both PBCS and DBNS solvers provide excellent match for discharge coefficient with the experimental data
- DBNS solver provides the best match with experimental data for the thrust parameter
- Shock diamond phenomena is captured quite well with all solvers
  - DBNS 2nd Order and PBCS with QUICK and PRESTO offer better resolution than PBCS 2nd Order
- Adaption doesn't seem to affect the nozzle internal pressure distribution
  - As the number of cells on the walls remain unchanged after adaption
- Adaption leads to very small improvement in the matching of thrust parameter with experimental data
  - As the mesh is fine enough to capture the shocks quite accurately

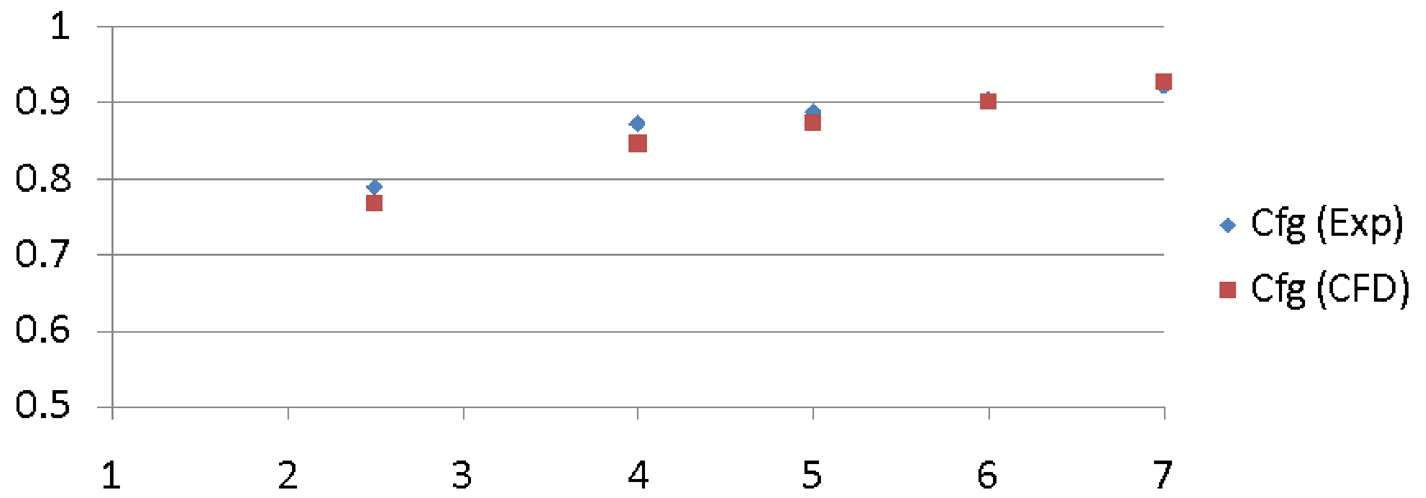
Test Set 2

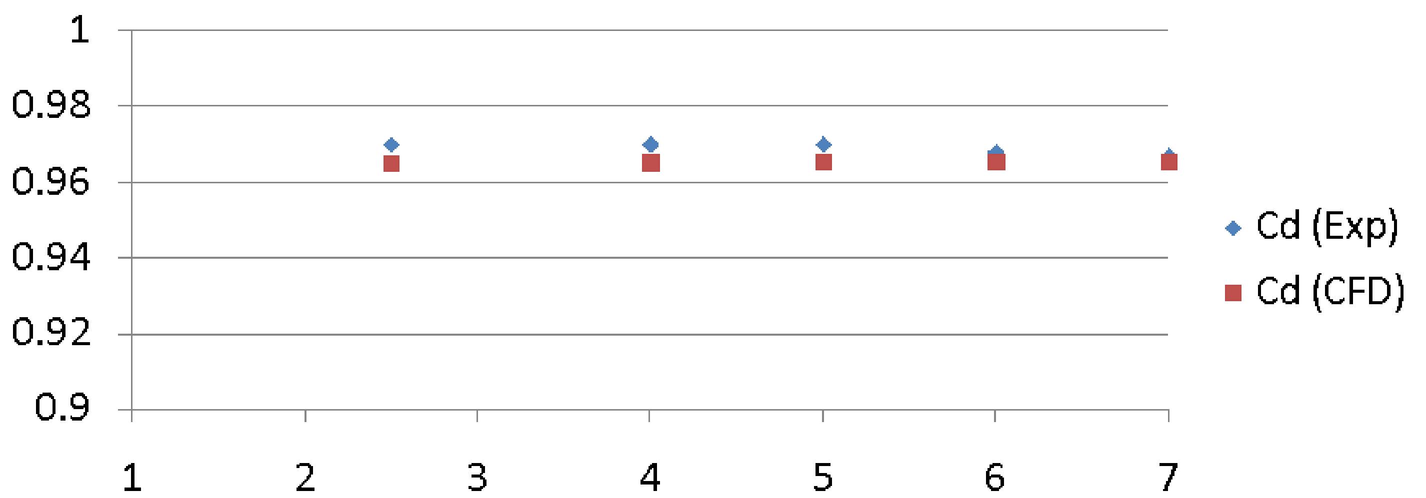
- Test cases
  - Nozzle in still air: Far-field Mach Number = 0
  - Nozzle Pressure Ratios (NPR)
    - 2.5, 4, 5, 6 and 7
- Solver Setup (based on the Test Set 1 results)
  - DBNS solver
  - Flux type: AUSM
  - Gradients: Least Squares cell based
  - Flow, Turbulence: Second Order

Test Set 2 : Nozzle Parameters

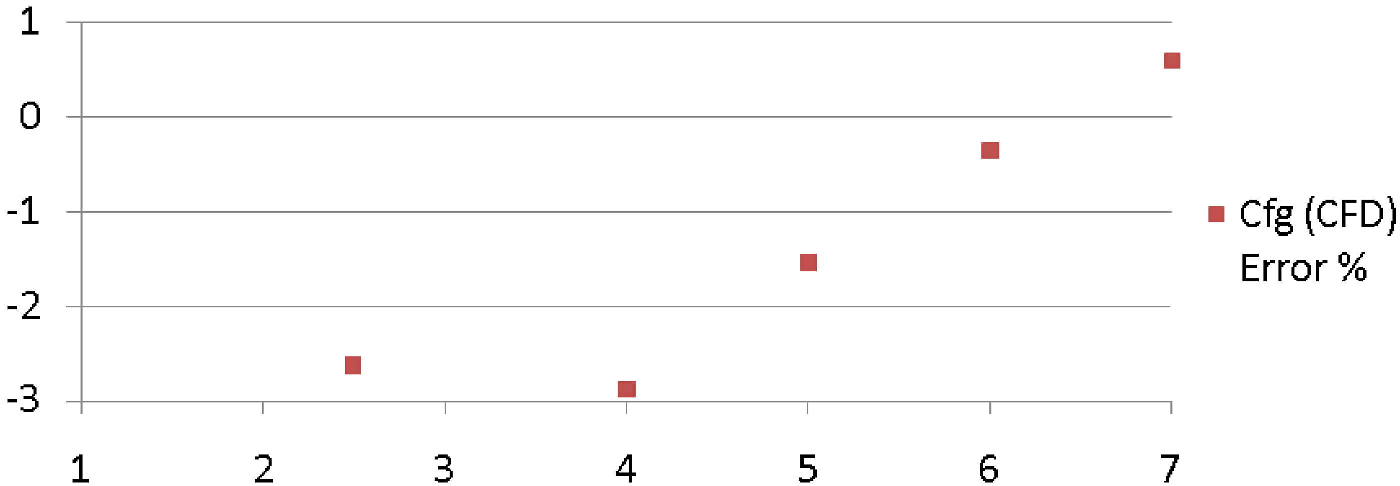
NPR	C_d (Exp)	C_d (CFD)	C_d (Error %)	Cfg (Exp)	Cfg (CFD)	Cfg (Error %)
2.5	0.97	0.9650	-0.519	0.79	0.7693	-2.626
4.0	0.97	0.9651	-0.505	0.873	0.8479	-2.872
5.0	0.97	0.9652	-0.496	0.889	0.8753	-1.536
6.0	0.968	0.9653	-0.284	0.906	0.9028	-0.358
7.0	0.967	0.9653	-0.176	0.923	0.9284	-0.590

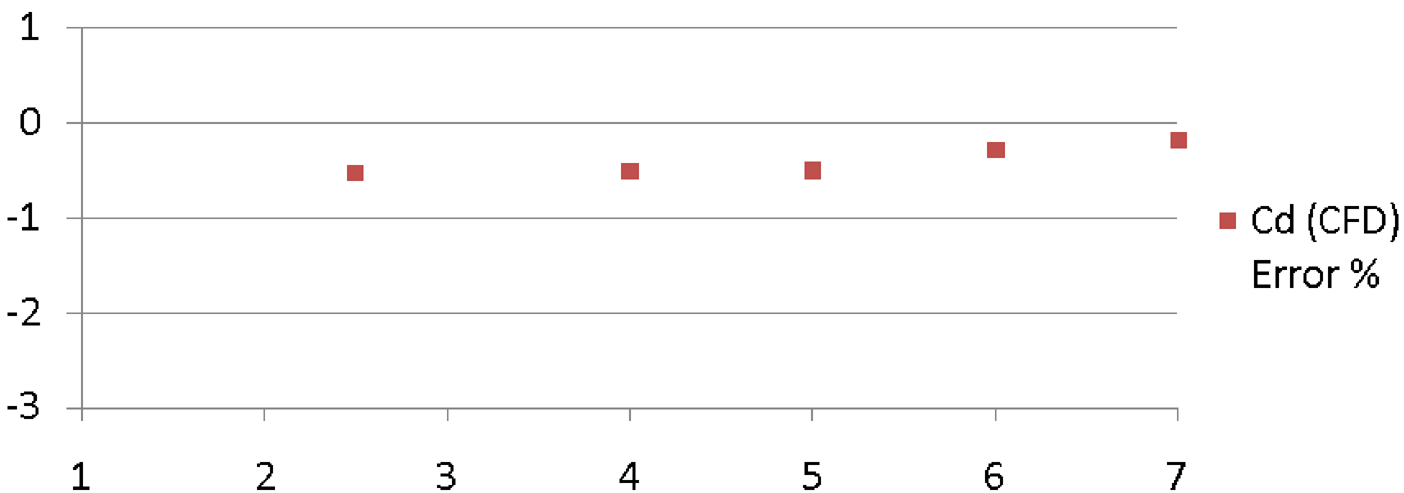
Test Set 2 : Plot of Nozzle Parameters





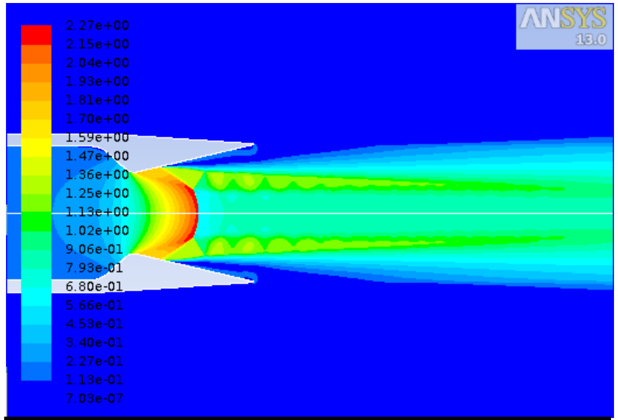
Test Set 2 : Error % in Nozzle Parameters



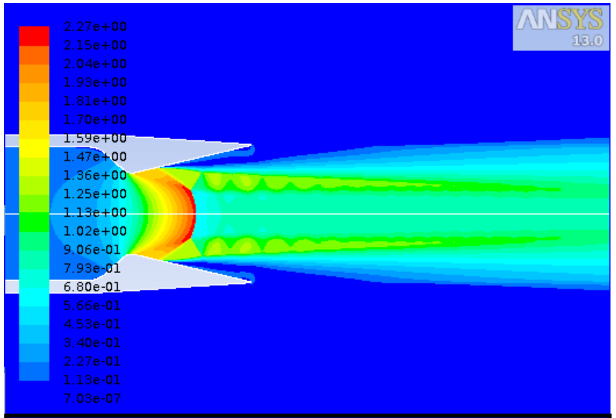


Test Set 2 : Mach Contours at different Pressure Ratios

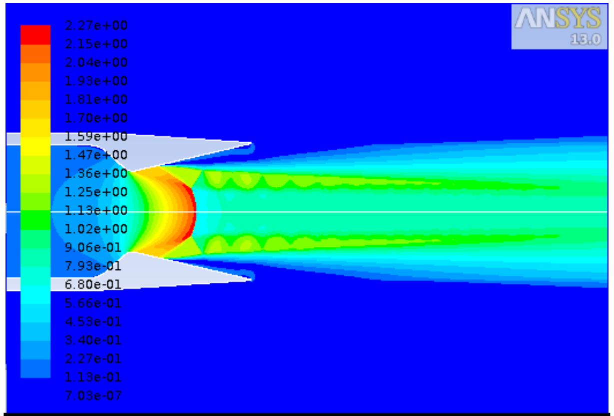
- Mach Contours
  - For different pressure ratios
  - Note: The color range corresponds to the minimum and maximum value for each case to highlight the shock location



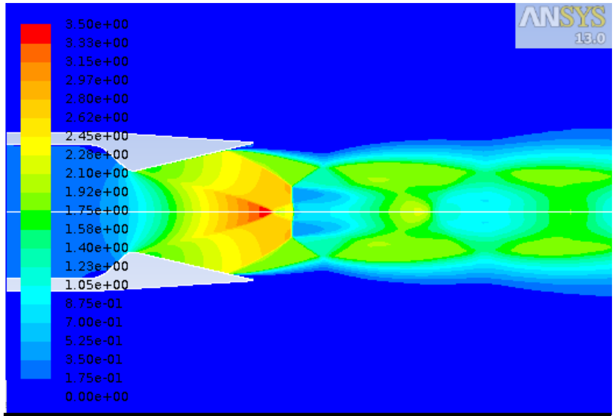
NPR = 2.5



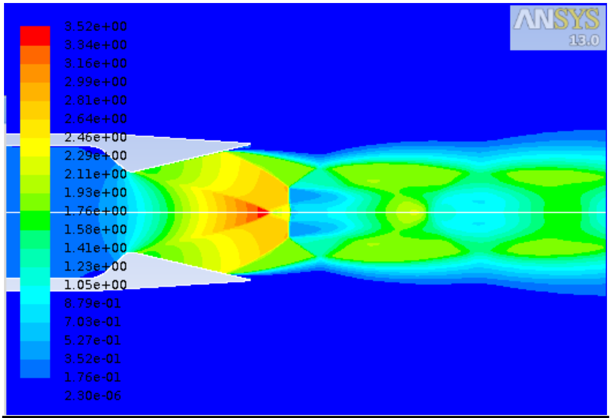
NPR = 4.0



NPR = 5.0



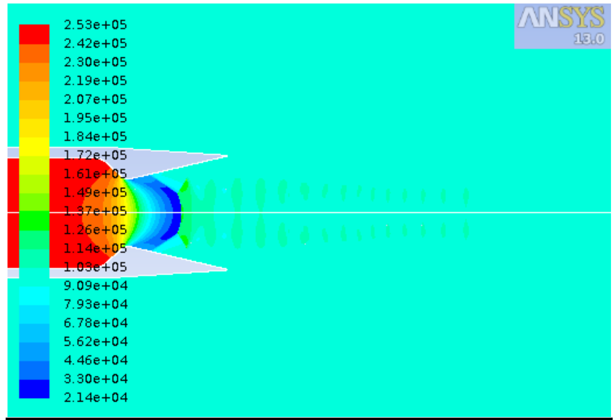
NPR = 6.0



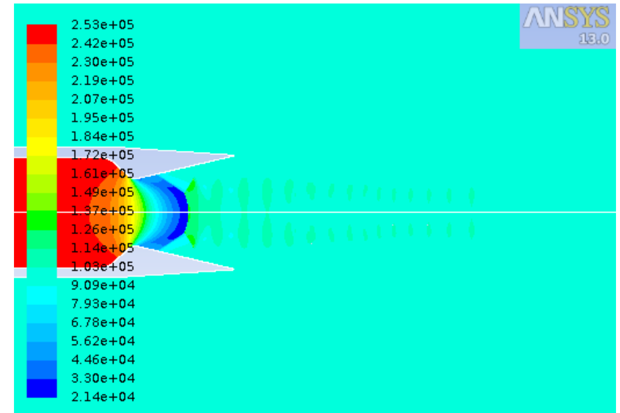
NPR = 7.0

Test Set 2 : Pressure Contours at different Pressure Ratios

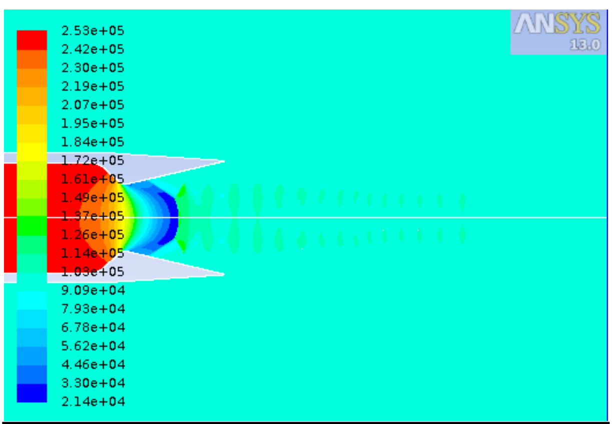
- Pressure Contours
  - For different pressure ratios
  - Note: The color range corresponds to the minimum and maximum value for each case to highlight the shock location



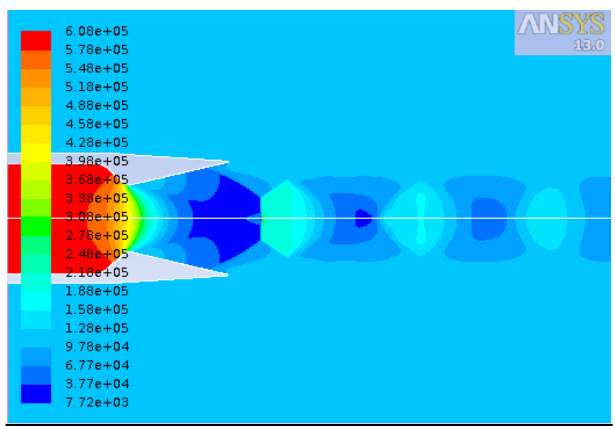
NPR = 2.5



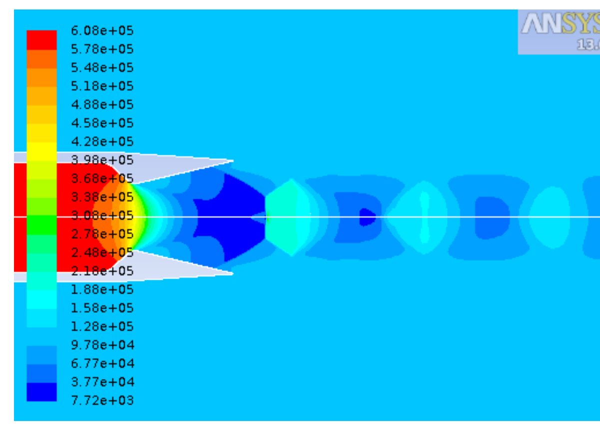
NPR = 4.0



NPR = 5.0



NPR = 6.0



NPR = 7.0

Conclusions

- DBNS solver was selected for Test Set 2 as it provided best agreement with experimental data for Test Set 1
- Excellent match with the experimental data was obtained for Test Set 2
- Shock diamond phenomena was captured very well
- Adaption wasn’t attempted for Test Set 2 as it didn’t affect the results significantly for Test Set 1
  - Mesh is already fine enough
- ANSYS CFD provides Pressure and Density based solvers with easy to use mesh adaptation capability
  - To capture shock-shock and shock-turbulence interactions very accurately