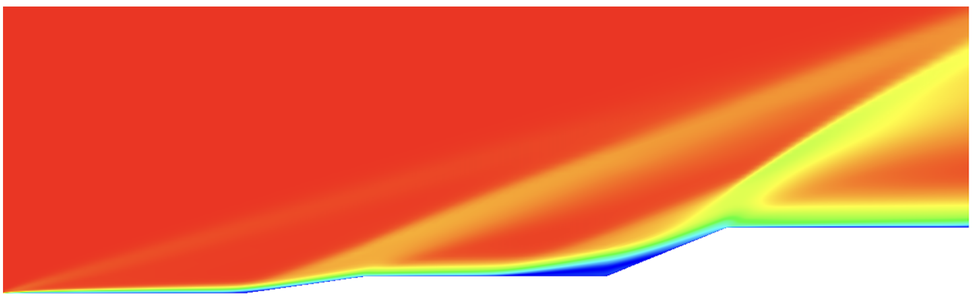
Niki Balestrieri
03782128
niki.balestrieri@tum.de

Gabriel Haga
03776720
gabriel.haga@tum.de

Supersonic Flow over a Double-Ramp Configuration

Project RB17



Abstract

In this study, simulations are conducted to analyze the behavior of **supersonic flow over a double ramp configuration**. The working fluid is **Air as Ideal Gas**, modeled as incompressible, with constant density and viscosity. The advection scheme used is both a **high-resolution** and **second-order scheme**, in order to capture in a detailed way convective effects. A **timestep control** was employed with a size of $5e-6s$ to ensure stability and convergence. **Boundary conditions** are evaluated at the **inlet**, comprending temperature, static pressure and Mach number, with no-slip wall at the surfaces. The grid consists of a **structured mesh**, **H-Grid**, refined next to the ramps.

1 Computational Setup

1.1 Geometry of Domain

In this discussion we will be dealing with the **supersonic flow over a double ramp**: each ramp presents a distinct inclination, respectively 8° and 22°, detail of major importance due to the formation and propagation of **shock waves**.
The domain is tailored on the given geometry and flow conditions, as illustrated in Figure 1, using **centimeters** as measurement unit. [1]
The length of the ramps are calculated respectively as:

$$5cm \cdot \tan(8°) = 0.7027\ cm; \qquad 5cm \cdot \tan(22°) = 2.7228\ cm; \tag{1}$$

For what concerns height and width, they have been arbitrarily set as 30 cm and 2 cm, constant coordinates for all the geometry.

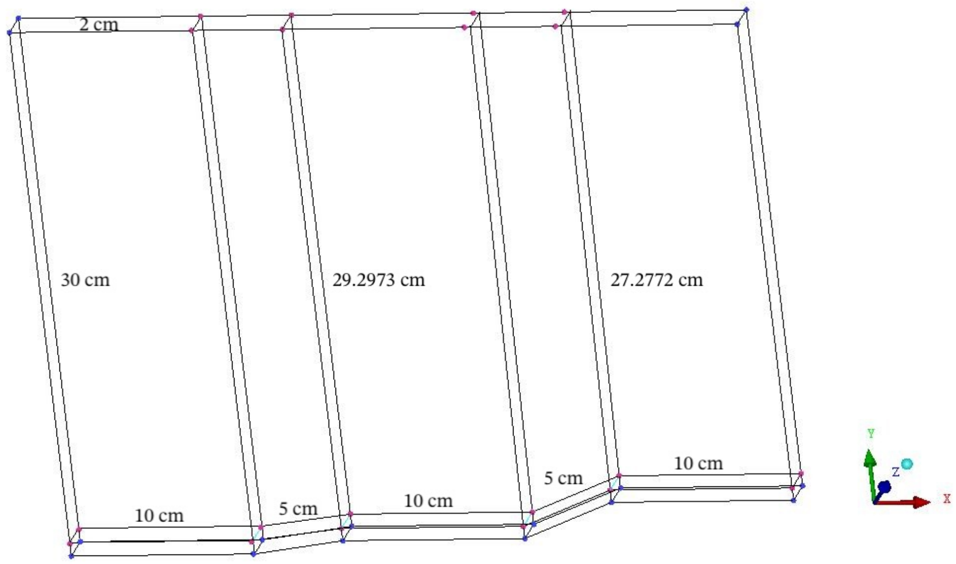
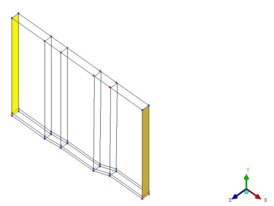
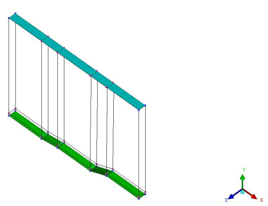


Figure 1: Geometry - Global structure and dimensions

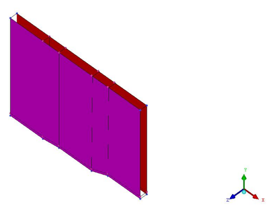
Subsequently, the geometry has been divided in operational **parts** in order to facilitate the simulation process, such as: inlet, outlet, ramp, upper wall, left and right symmetry.



(a) Outlet-Inlet



(b) Ramp-Upper Wall



(c) Left-Right Walls

Figure 2: Geometry - Parts division

[1]The choice of using centimeters is justified by reference Verma et al. [1].

1.2 Boundary Conditions

For obtaining the boundary conditions, first we derive the properties at the **inlet**, since at the outlet we don't have any type of constraint.
Knowing that we are working with **Air as Ideal Gas**, so having a isentropic expansion factor of $\gamma = 1.4$, we can then calculate and assume the following:

- **Temperature** - having a $T_0 = 900K$:

$$T_{inlet} = \frac{T_0}{1 + \frac{\gamma-1}{2}M^2} = 231.5K \tag{2}$$

- **Static Pressure** - assuming a $p_0 = 100\text{kPa}$:

$$p_{inlet} = \frac{p_0}{(1 + \frac{\gamma-1}{2}M^2)^{\frac{\gamma}{\gamma-1}}} = 0.863\text{kPa} \tag{3}$$

- **Normal Mach Number** - having a $M = 3.8$
- **Imposed Symmetry**- as we deal with a 2D problem, we specified left and right side of the ramp as a symmetry boundary.

Another important factor is the **velocity** of the fluid at the inlet, derived by definition of Mach number and speed of sound:

$$u = M\sqrt{(\gamma \cdot R \cdot T_{inlet})} = 1159m/s \tag{4}$$

Where R is the specific gas constant[2].

The boundary conditions and the corresponding derived data are then summarized in Table 1.

Property	Value	Unit
Temperature	231.5	K
Static Pressure	0.863	kPa
Mach Number	3.8	
Velocity	1159	m/s

Table 1: Inlet Data

1.3 Time discretization

In regards to the time discretization, the used value it has been obtained through the following calculation:

$$dt = \frac{dx}{u + c} = \frac{0.01cm}{1159m/s + 304m/s} = 5e^{-6}s \tag{5}$$

Respectively the **smallest element of the grid**, the **fluid velocity** u and the **speed of sound**.

[2]Considered as $R = 287J/kg \cdot K$

1.4 Turbulence Model

The used turbulence model is the Shear Stress Transport, or **SST Model**, which integrates both the $k - \epsilon$ and $k - \omega$ models. This combination makes it suitable for problems showing near-wall flows and free-stream regions.
While the SST model is very versatile for a broad range of flow conditions, it is less accurate then using individually the other two models in the specific areas.

2 Grid

2.1 Blocking structure

Once established the geometry's topology, the blocking is shaped and divided in **ten** main parts, as is shown in Figure 3 and Figure 4: this choice is justified by the presence of the ramps and the desire of having a better management of coarser and wider mesh where needed.

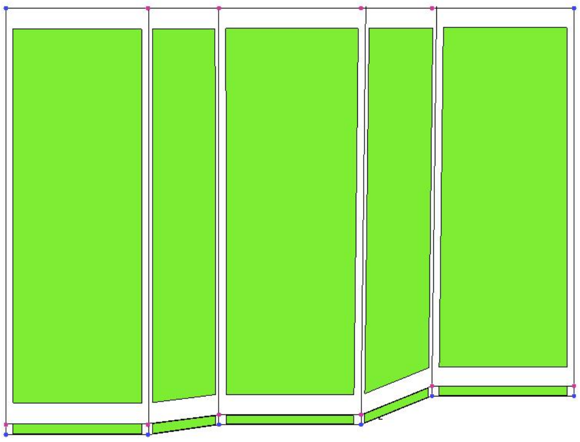


Figure 3: Blocks - Frontal perspective

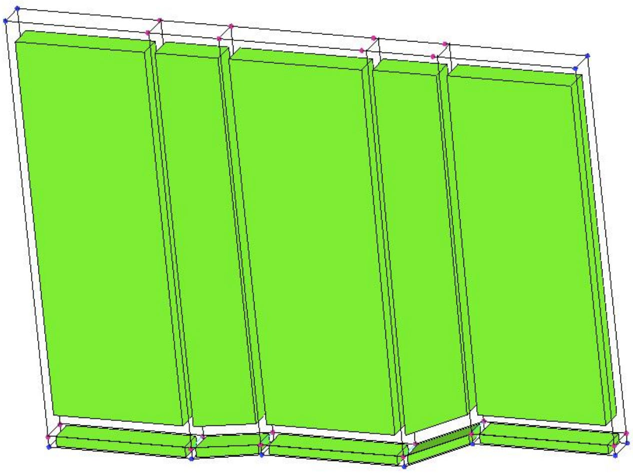


Figure 4: Blocks - Tilted perspective

Due to the conformation of the geometry and the absence of round corners, the topology of the grid is restricted to a **H-Grid**.

2.2 Resolution and Quality of the Grid

The refinement of the grid it has been made considering the presence of the shock waves and their spacial propagation. Therefore, in order to congruently define nodes and elements, the following mesh laws have been chosen, as summarized in Table 2. If not specified, the function "Copy parameter –> Apply to all parallel edges" has been used.

	Nodes	Meshing Law	Spacing
Vertical Edges 1	15	Exponential 1	0.01
Vertical Edges 2	74	Exponential 1	0.15
Horizontal Block 1	75	Exponential 2	0.0999998
Horizontal Block 2	60	Uniform	0.0844193
Horizontal Block 3	60	Biexponential	0.0844193
Horizontal Block 4	100	Uniform	0.0544713
Horizontal Block 5	150	Uniform	0.0671141

Table 2: Meshing Laws

The mesh has been built with this parameters for being more refined in locations of structural discontinuation such as the ramps, both in vertical and horizontal direction as illustrated in Figure 6.

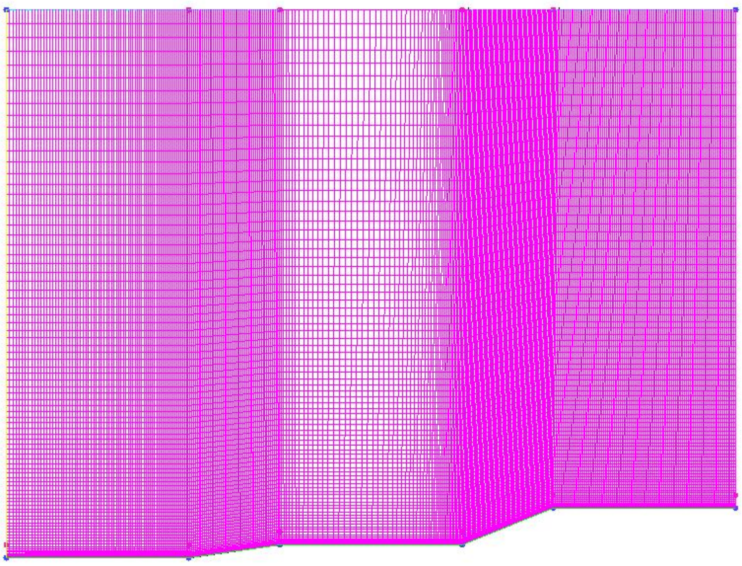
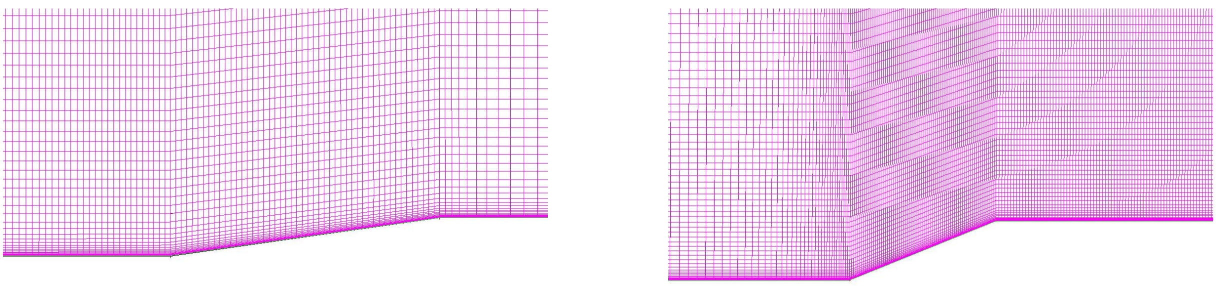


Figure 5: Grid - Frontal perspective



(a) Detail - First Ramp (b) Detail - Second Ramp

Figure 6: Grid - Details

The mesh has also been visibly refined next to the wall due to the boundary layer. This resulted in **82016 nodes** and **122469 elements**, with 82014 *QUAD4* and 40455 *HEXA8*. Last, we performed the quality control, summarizable in three main points: **volume**, **determinant** and **minimum angle**.
The values are presented in Table 3 and are coherent for a stable simulation.

	Minimum Value	**Maximum Value**
Volume	$9.08e{-4}$	0.493
Determinant	0.998	1
Minimum angle	67.3	90

Table 3: Grid quality control values

2.3 Y+ parameter

Next, we proceed evaluating the **resolution of the boundary layer** through the analysis of the y+ parameter for no-slip walls.
The optimal resolution is attained when $y \approx 1$, indicating that the constructed mesh satisfies the objective, as is shown in Figure 7.
The evaluation sees the **viscous simulation** and the **special task** altogether for better comparison.

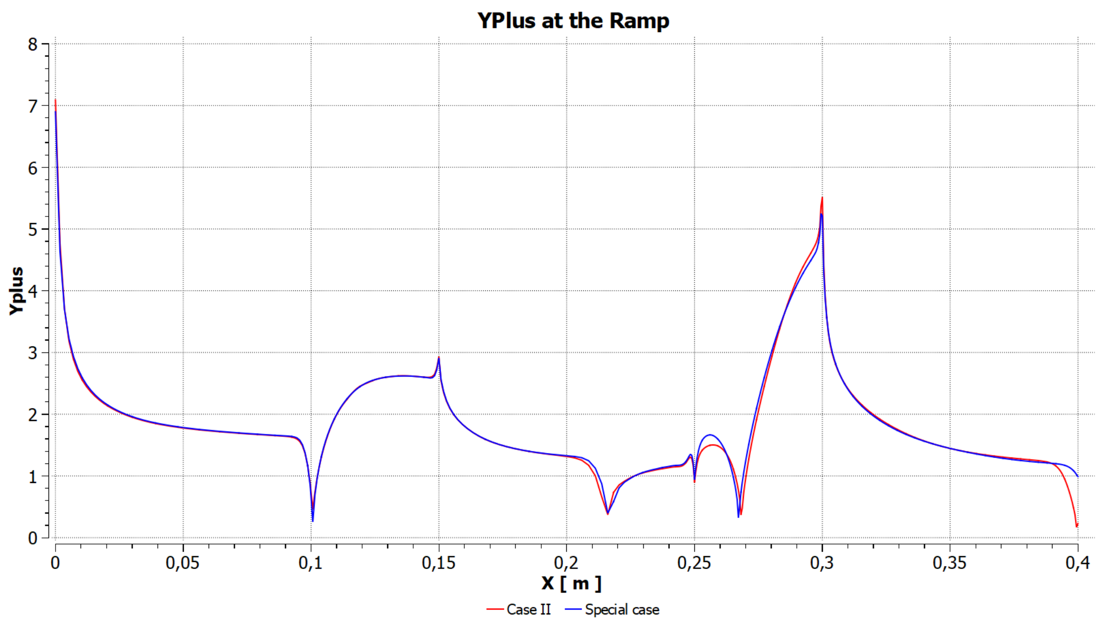


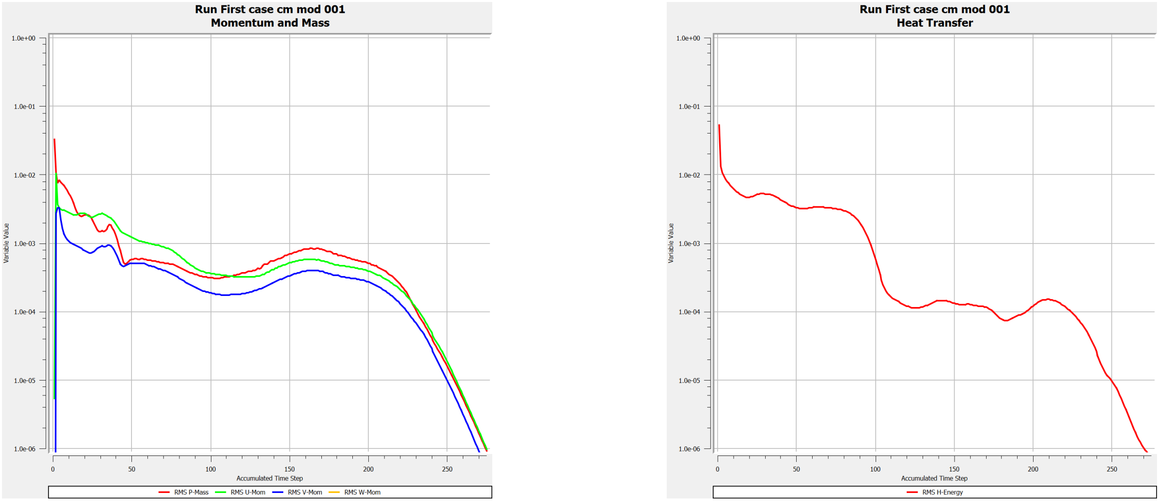
Figure 7: Y+ plots of simulations

3 Convergence Behavior

Regarding the simulation, we will address two distinct cases: first, a basic **inviscid simulation**, followed by a more complex **viscid one**. The former serves as a preliminary step to guarantee coherent results.

3.1 I CASE - Inviscid simulation

For this analysis of the residuals, a **physical timestep** of $5e-6s$ was defined.



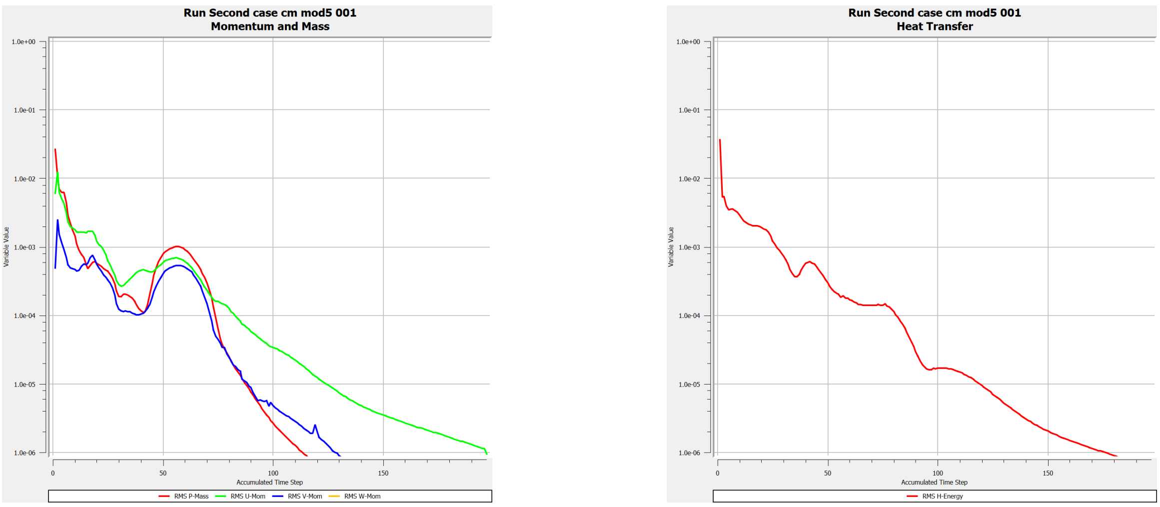
(a) Momentum and Mass (b) Heat Transfer

Figure 8: Case I - Residual Convergence

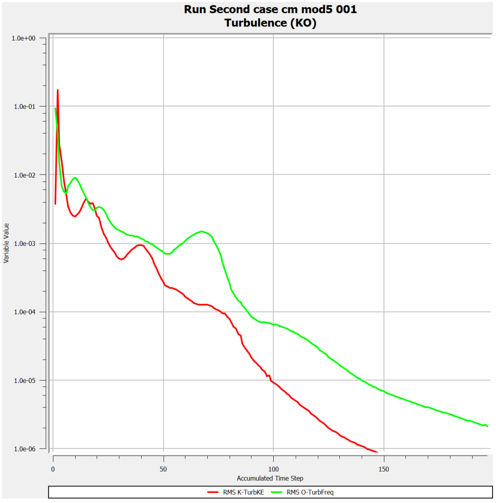
As illustrated in Figure 8, the residuals presents a good convergence behavior, especially towards the end, where they start decreasing with a greater rate.

3.2 II CASE - Viscid simulation

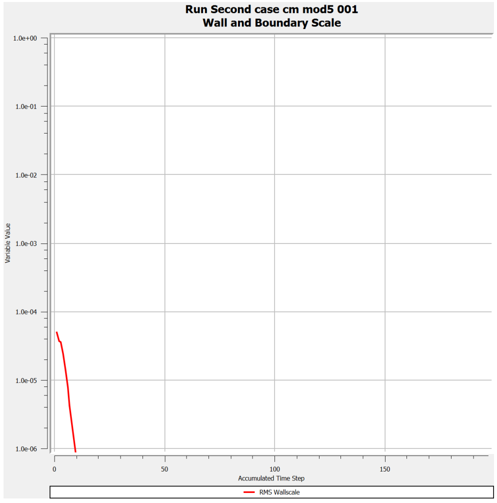
For the viscid case, the same physical timestep value was initially used.
However, once the residuals stabilized, a **local timescale factor** was applied in order to speed up the convergence, starting with a value of 5 and later decreased to 2.
The switch to **local timescale factor** requires that the wall and boundary scale residual **have already converged**, satisfied condition as shown in Figure 10.
Overall, the residuals exhibit proper convergence behavior.



(a) Momentum and Mass (b) Heat Transfer



(a) Turbulence



(b) Wall and Boundary Scale

Figure 10: Case II - Residual Convergence

4 Final results

First of all, since the flow is in a **supersonic regime**, shock waves are expected at the beginning of the ramps while expansion fans at the end of them. The inviscid case with free-slip wall will align with the results obtained using oblique compression shock wave and expansion fan theories.
The following figures show the results for the **inviscid case**. The black lines represent the compression shock waves that have been also calculated theoretically. As expected, the results in Figure 11 and Figure 12 are close to the theoretical calculations.

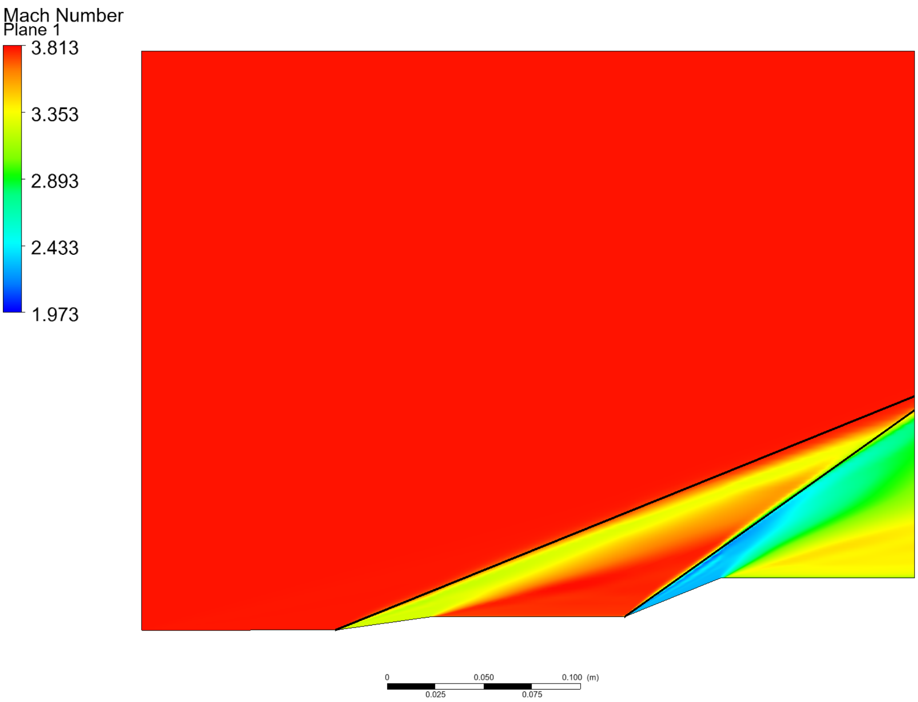


Figure 11: Case I - Mach Number

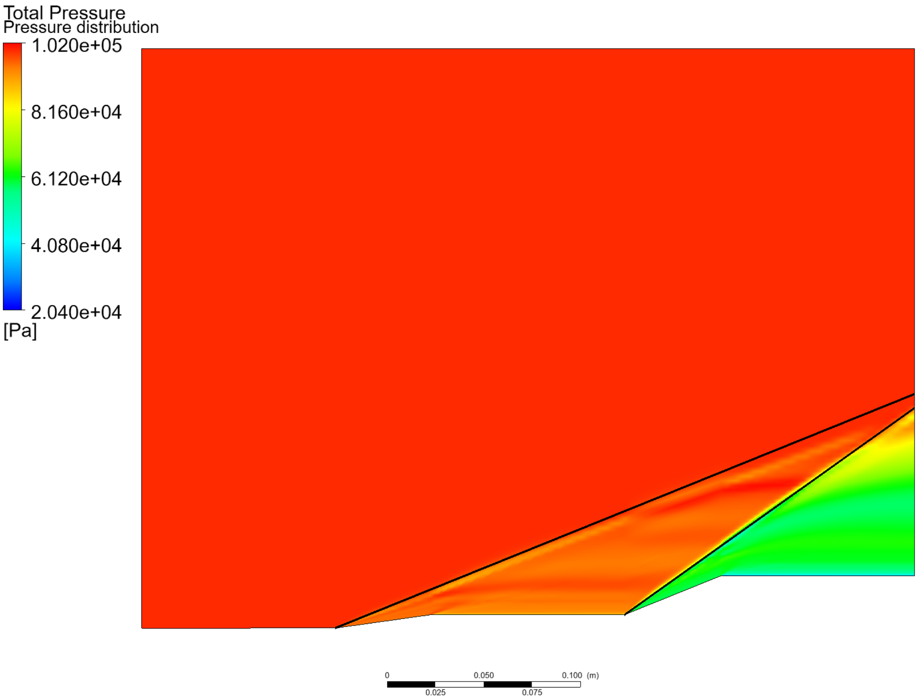


Figure 12: Case I - Total Pressure

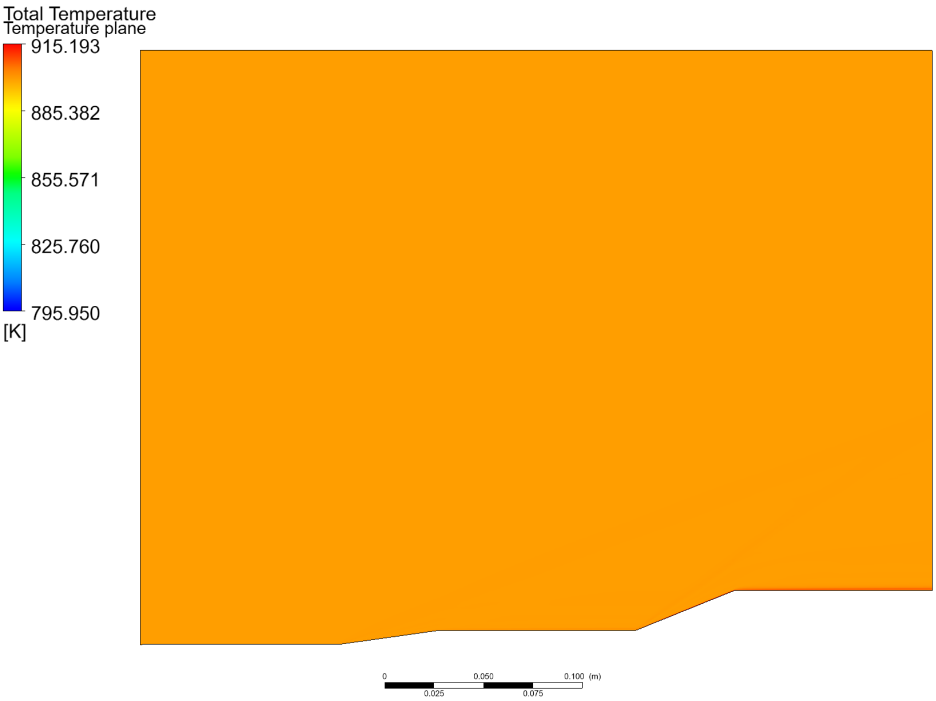


Figure 13: Case I - Total Temperature

As an **inviscid fluid with free-slip wall**, the only mechanism for pressure drop are compression shock waves as clarified in Figure 12. Shock waves does not affects the Total temperature which is demonstrated in Figure 13.

For the second case, ergo **viscid flow**, the results are presented in the following figures. In this scenario, it is expected that the viscosity combined with the no-slip wall condition results in a decrease in velocity near the wall, which is precisely what Figure 14 shows. Additionally, the Mach number gradient across the compression shock waves is lower compared to the inviscid case.

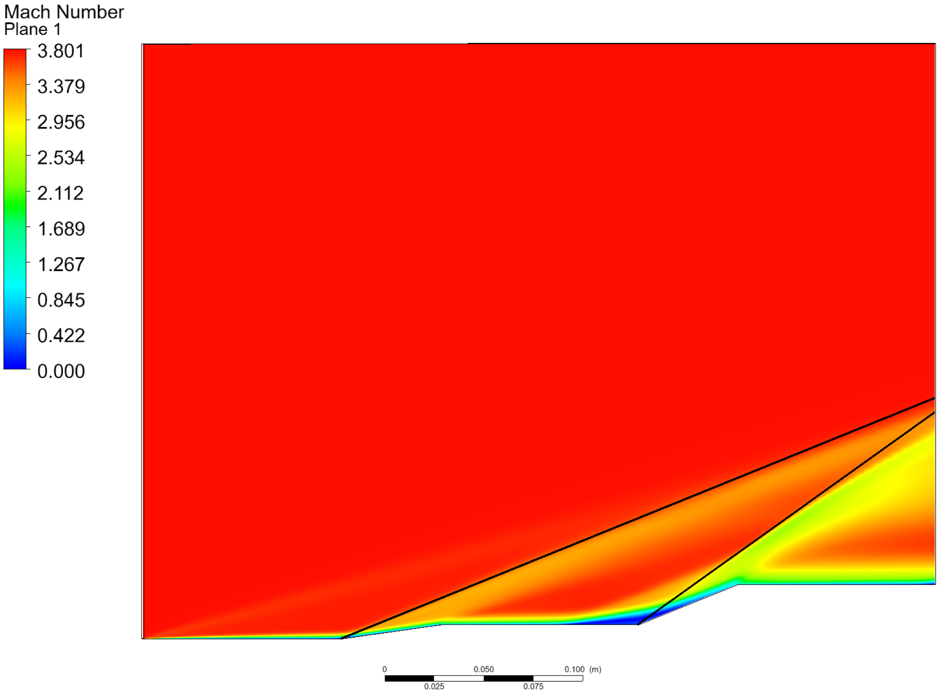


Figure 14: Case II - Mach Number

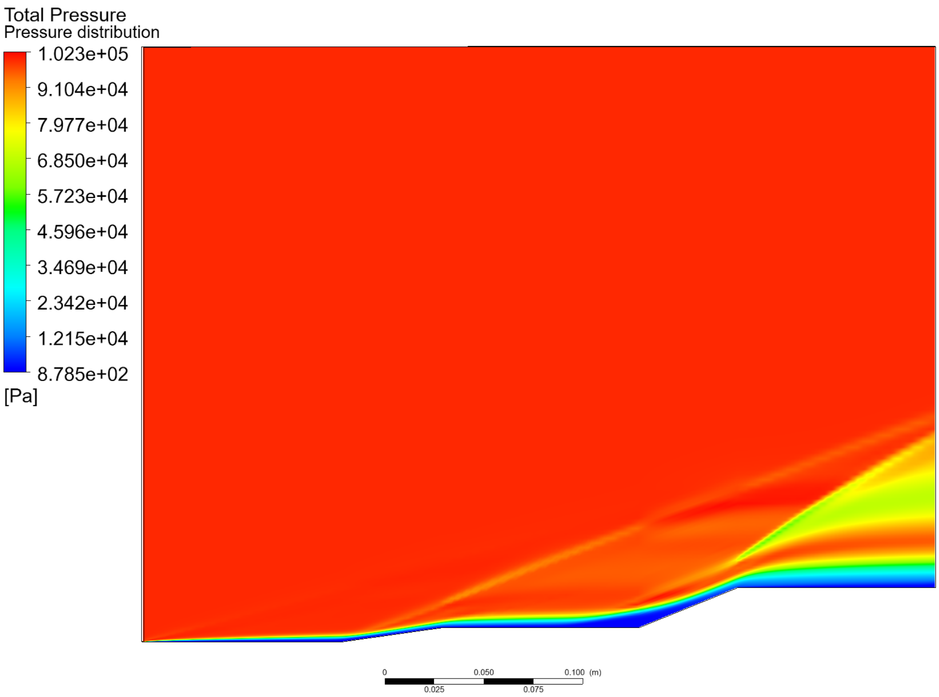


Figure 15: Case II - Total Pressure

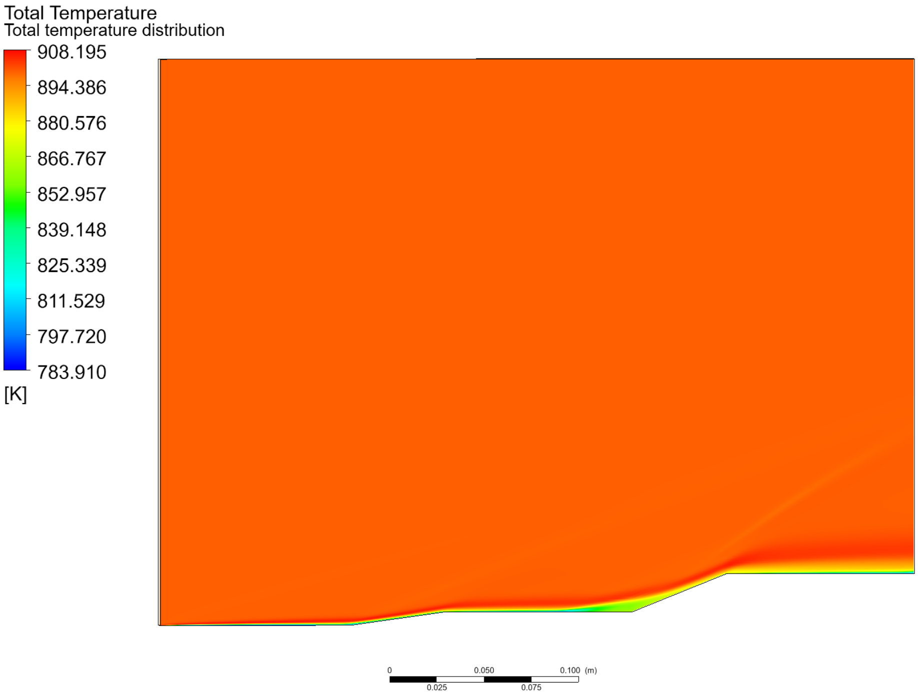
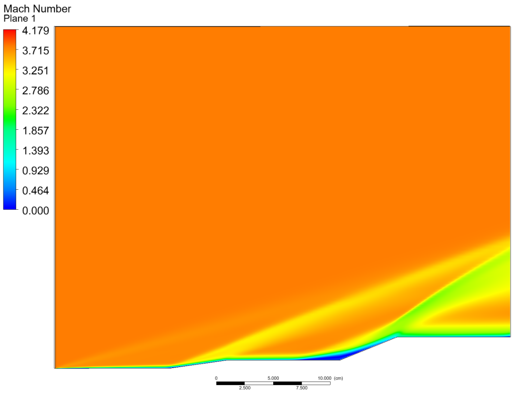


Figure 16: Case II - Total Temperature

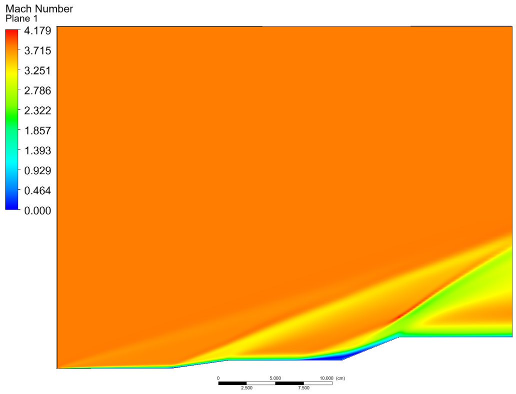
Another key aspect of this case is the new mechanism for **total pressure loss**, which arises because of the viscous force near to the wall (Figure 15). This figure also shows the decrease of the pressure drop, due to the compression shock waves, which explains the **lower Mach number gradients**.
The last comparison between the two cases is regarding the **total temperature**. Different than the last case, the viscosity leads to a loss of total temperature near to the wall as in Figure 16.

5 Special Task

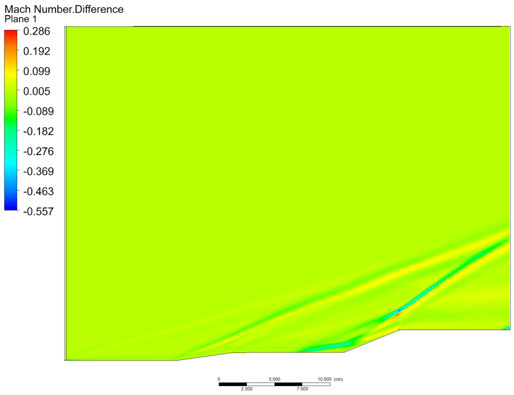
The special task requires changing the advection scheme from **high-resolution** to **second order**. In **Ansys CFX**, the high resolution scheme combines both first order, as Upwind Differencing scheme (**UDS**), and second order methods, as Central Differencing scheme (**CDS**).
The **comparison of Mach number distributions** results are displayed below in Figure 17.



(a) Case II - Mach number



(b) Special case - Mach number



(c) Differences -
Case II and Special Case

Figure 17: Comparison of cases

The solutions differ particularly in regions with higher gradients. This can be attributed to the **dispersive** nature of the **CDS error**, whereas the **UDS error** exhibits a **dissipative** behavior. This difference becomes more pronounced near **discontinuities**, and a shock wave is essentially a type of discontinuity.

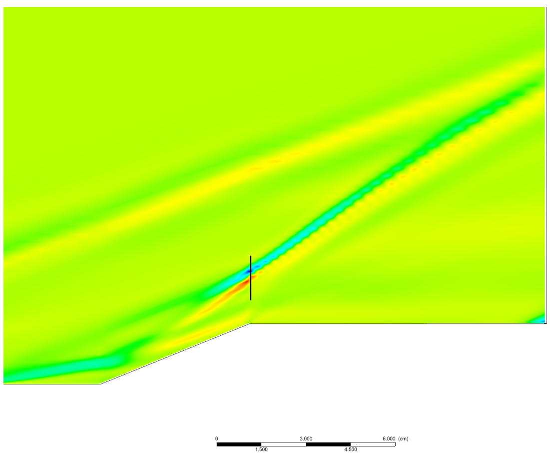


Figure 18: Line comparison

To prove this behavior, a line was drawn as illustrated in Figure 18, with the goal of capturing the regions with the largest difference values. A plot was then generated for each solution of the Mach number.

Figure 19 displays the anticipated behavior for each solution: the special case shows dispersive effects characterized by oscillations, while Case II demonstrates dissipative behavior identified by the absence of overshoots.

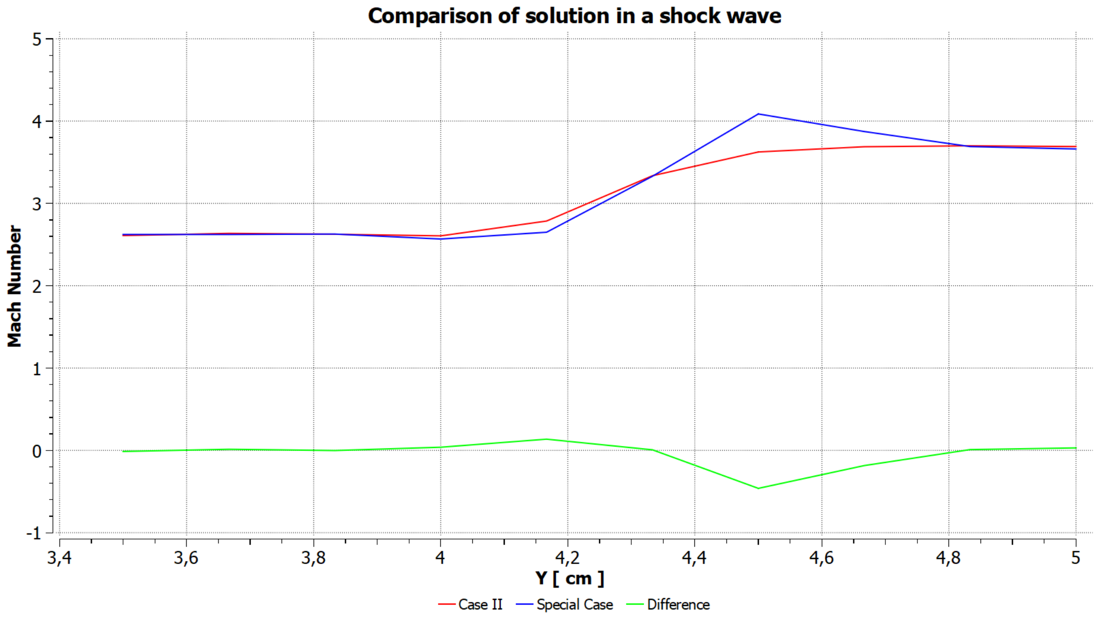


Figure 19: Comparison - Mach number of the two simulations

Annex - First case

Date

2024/09/15 23:23:57

Contents

Annex - First case

1. File Report

Table 1. File Information for First_case_cm_mod_001

Case	First_case_cm_mod_001
File Path	C:/Users/hagag/TUM/Applied CFD/Final Project/First_case_cm_mod_001.res
File Date	15 September 2024
File Time	09:04:30 PM
File Type	CFX5
File Version	24.1

Annex - First case

2. Mesh Report

Table 2. Mesh Information for First_case_cm_mod_001

Domain	Nodes	Elements
Default Domain	65988	32550

Annex - First case

3. Physics Report

Table 3. Domain Physics for First_case_cm_mod_001

Domain - Default Domain	
Type	Fluid
Location	SOLID
Materials	
Inviscid Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	2.0000e-2 [kPa]
Heat Transfer Model	Total Energy
Include Viscous Work Term	True
Turbulence Model	Laminar

Table 4. Boundary Physics for First_case_cm_mod_001

Domain	**Boundaries**	
Default Domain	**Boundary - Inlet**	
	Type	INLET
	Location	INLET
	Settings	
	Flow Regime	Supersonic
	Heat Transfer	Total Temperature
	Total Temperature	9.0000e+2 [K]
	Mass And Momentum	Normal Speed and Total Pressure
	Normal Speed	1.1590e+3 [m s^-1]
	Relative Total Pressure	9.9980e+1 [kPa]
	Boundary - Outlet	
	Type	OUTLET
	Location	OUTLET
	Settings	
	Flow Regime	Supersonic
	Boundary - Symmetry	
	Type	SYMMETRY

Annex - First case

	Location	LEFT_SYMMETRY, RIGHT_SYMMETRY, UPPER_SURFACE
	Settings	
	Boundary - Ramp	
	Type	WALL
	Location	RAMP
	Settings	
	Heat Transfer	Adiabatic
	Mass And Momentum	Free Slip Wall

Annex - First case

4. User Data

Annex - Second case

Date

2024/09/15 19:31:52

Contents

Annex - Second case

1. File Report

Table 1. File Information for Second_case_cm_mod5_001

Case	Second_case_cm_mod5_001
File Path	C:/Users/hagag/TUM/Applied CFD/Final Project/Second_case_cm_mod5_001.res
File Date	15 September 2024
File Time	05:15:53 PM
File Type	CFX5
File Version	24.1

Annex - Second case

2. Mesh Report

Table 2. Mesh Information for Second_case_cm_mod5_001

Domain	Nodes	Elements
Default Domain	82016	40455

Annex - Second case

3. Physics Report

Table 3. Domain Physics for Second_case_cm_mod5_001

Domain - Default Domain	
Type	Fluid
Location	SOLID
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	2.0000e-2 [kPa]
Heat Transfer Model	Total Energy
Include Viscous Work Term	True
Turbulence Model	SST
Turbulent Wall Functions	Automatic
High Speed Model	On

Table 4. Boundary Physics for Second_case_cm_mod5_001

Domain	Boundaries	
Default Domain	**Boundary - Inlet**	
	Type	INLET
	Location	INLET
	Settings	
	Flow Regime	Supersonic
	Heat Transfer	Total Temperature
	Total Temperature	9.0000e+2 [K]
	Mass And Momentum	Normal Speed and Total Pressure
	Normal Speed	1.1590e+3 [m s^-1]
	Relative Total Pressure	9.9980e+1 [kPa]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary - Outlet	
	Type	OUTLET
	Location	OUTLET
	Settings	
	Flow Regime	Supersonic

Annex - Second case

	Boundary - Symmetry	
	Type	SYMMETRY
	Location	LEFT_SYMMETRY, RIGHT_SYMMETRY, UPPER_SURFACE
	Settings	
	Boundary - Ramp	
	Type	WALL
	Location	RAMP
	Settings	
	Heat Transfer	Adiabatic
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

Annex - Second case

4. User Data

Chart 1.

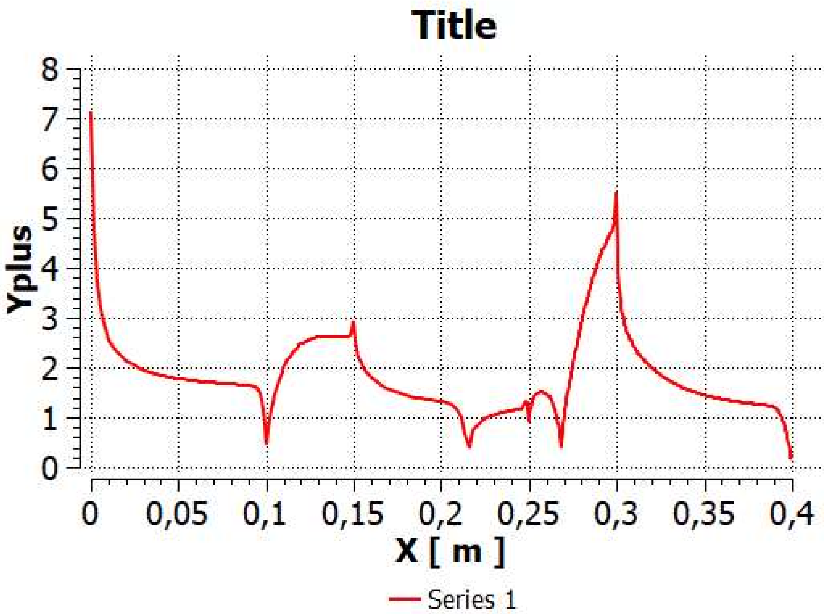
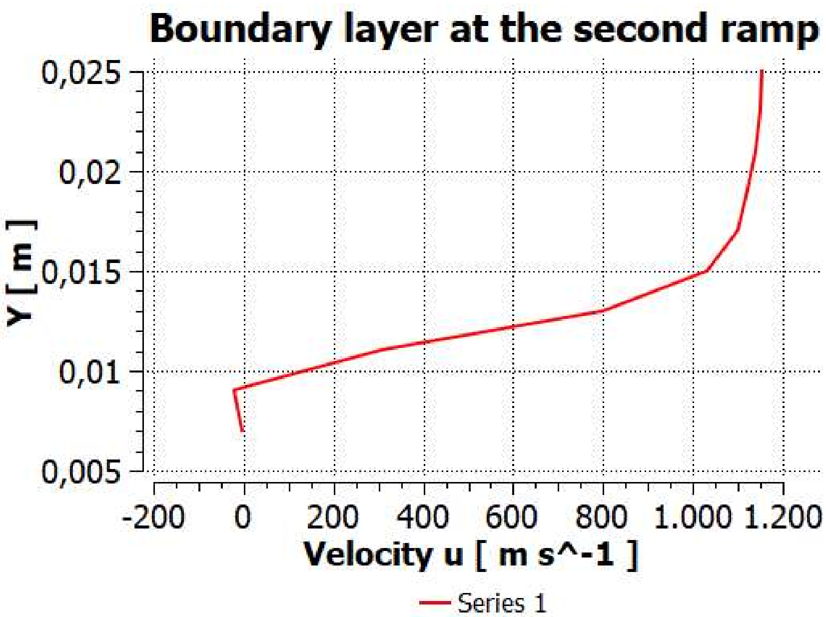


Chart 2.



Annex - Special case

Date

2024/09/15 23:27:03

Contents

Annex - Special case

1. File Report

Table 1. File Information for Special_case_cm_mod1_002

Case	Special_case_cm_mod1_002
File Path	C:/Users/hagag/TUM/Applied CFD/Final Project/Special_case_cm_mod1_002.res
File Date	15 September 2024
File Time	09:04:32 PM
File Type	CFX5
File Version	24.1

Annex - Special case

2. Mesh Report

Table 2. Mesh Information for Special_case_cm_mod1_002

Domain	Nodes	Elements
Default Domain	82016	40455

Annex - Special case

3. Physics Report

Table 3. Domain Physics for Special_case_cm_mod1_002

Domain - Default Domain	
Type	Fluid
Location	SOLID
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	2.0000e-2 [kPa]
Heat Transfer Model	Total Energy
Include Viscous Work Term	True
Turbulence Model	SST
Turbulent Wall Functions	Automatic
High Speed Model	On

Table 4. Boundary Physics for Special_case_cm_mod1_002

Domain	**Boundaries**	
Default Domain	**Boundary - Inlet**	
	Type	INLET
	Location	INLET
	Settings	
	Flow Regime	Supersonic
	Heat Transfer	Total Temperature
	Total Temperature	9.0000e+2 [K]
	Mass And Momentum	Normal Speed and Total Pressure
	Normal Speed	1.1590e+3 [m s^-1]
	Relative Total Pressure	9.9980e+1 [kPa]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary - Outlet	
	Type	OUTLET
	Location	OUTLET
	Settings	
	Flow Regime	Supersonic

Annex - Special case

	Boundary - Symmetry	
	Type	SYMMETRY
	Location	LEFT_SYMMETRY, RIGHT_SYMMETRY, UPPER_SURFACE
	Settings	
	Boundary - Ramp	
	Type	WALL
	Location	RAMP
	Settings	
	Heat Transfer	Adiabatic
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

Annex - Special case

4. User Data

Chart 1.

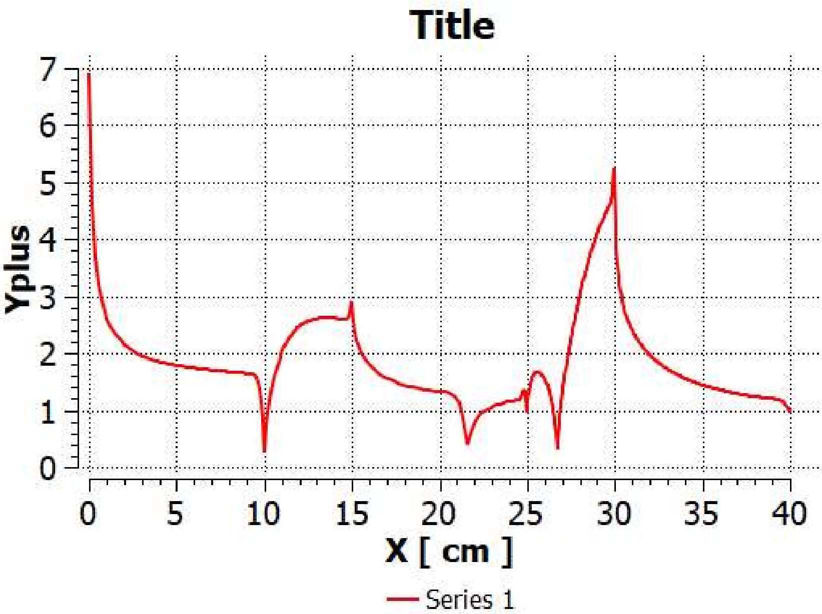
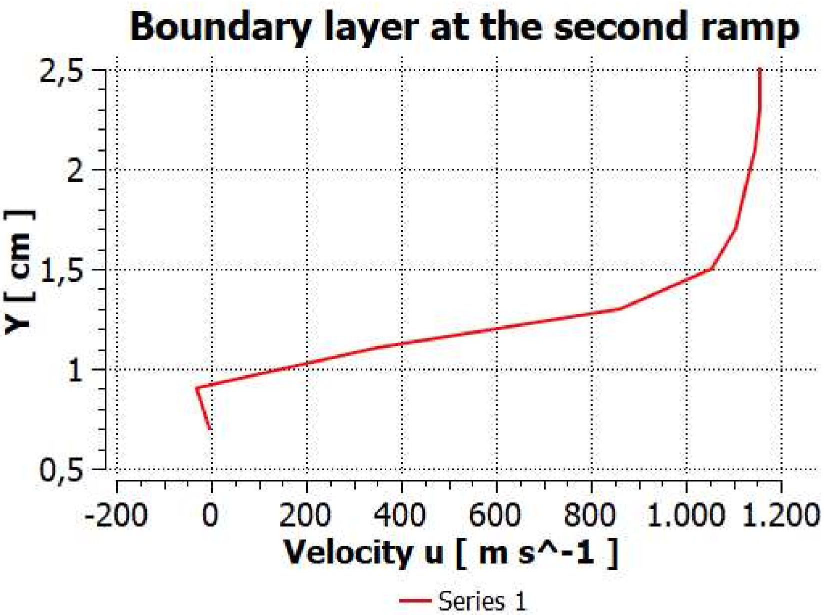


Chart 2.



6 References

[1] S. B. Verma, C. Manisankar, and C. Raju, “Control of shock unsteadiness in shock boundary- layer interaction on a compression corner using mechanical vortex generators,” Shock Waves, vol. 22, no. 4, pp. 327–339, 2012, doi: 10.1007/s00193-012-0369-8.