# Objectives and motivation of the project

In the aeronautical industry, it is of vital importance the study of the fluid’s behaviour in different conditions as the air behaves as a fluid and in order to design an aircraft (airfoil, sustaining surfaces, etc ) it is necessary these study with the aim of being able to perform their functions at all times.

In fluid dynamics, the study of the fluid behaviour has a great complexity; several manipulations of complex equations are required and some simplifications are needed to simplify them and attempt achieving an analytical solution.

However, apart from these simplifications, nowadays we have help of computers and computational fluids dynamics to facilitate even more the process of obtaining the solutions of these problems related with the behaviour study of fluids, giving more visual results and using numerical resolution techniques instead of complex analytical solutions.

This project is orientated to the simulation of one of the process that occurs inside the supersonic air flux, in concrete, the study of an expansive wave, also known as Prandtl-Meyer expansion.

En la industria aeronáutica es de vital importancia el estudio del comportamiento del fluido en diferentes condiciones ya que el aire se comporta como un fluido y para el diseño de una aeronave (perfil aeronáutico, superficies de sustentación, etc) es necesario este estudio con el objetivo de poder realizar sus funciones en todo momento.

En dinámica de fluidos, el estudio del comportamiento de los fluidos es de gran complejidad; se requieren múltiples manipulaciones de ecuaciones complejas y algunas simplificaciones para intentar lograr una solución analítica.

Sin embargo, además de estas simplificaciones, hoy en día contamos con la ayuda de los ordenadores y la dinámica de fluidos computacional para facilitar aún más el proceso de obtención de las soluciones de estos problemas relacionados con el estudio del comportamiento de los fluidos, dando lugar a resultados más visuales y utilizando técnicas de resolución numérica en lugar de complejas soluciones analíticas.

Este proyecto está orientado a la simulación de uno de los procesos que ocurren dentro del flujo de aire supersónico, en concreto, el estudio de una onda expansiva, también conocida como expansión de Prandtl-Meyer.

A la indústria aeronàutica és de vital importància l'estudi del comportament del fluid en diferents condicions ja que l'aire es comporta com un fluid i per al disseny d'una aeronau (perfil aeronàutic, superfícies de sustentació, etc) cal aquest estudi amb l'objectiu de poder fer les seves funcions en tot moment.

En dinàmica de fluids, l’estudi del comportament dels fluids és de gran complexitat; es requereixen múltiples manipulacions d'equacions complexes i algunes simplificacions per intentar assolir una solució analítica.

Tot i això, a més d'aquestes simplificacions, avui dia comptem amb l'ajuda dels ordinadors i la dinàmica de fluids computacional per facilitar encara més el procés d'obtenció de les solucions d'aquests problemes relacionats amb l'estudi del comportament dels fluids, donant lloc resultats més visuals i utilitzant tècniques de resolució numèrica en lloc de complexes solucions analítiques.

Aquest projecte està orientat a la simulació d'un dels processos que tenen lloc dins el flux d'aire supersònic, en concret, l'estudi d'una ona expansiva, també coneguda com a expansió de Prandtl-Meyer.

# Description of the physic problem and its relevance

## 2.1 Introduction of the problem



Figure 1. Prandtl-Meyer expansive wave [1]

The Prandtl-Meyer expansion or expansive wave is the process that occurs inside a supersonic flux when it expands over a convex corner (that forms an angle θ with the horizontal), creating infinity of Mach waves.

This expansion is caused by the drastic change in the direction of the geometry of flux, as it is a convex corner. This phenomenon affects the fluid properties in a gradual and continuous way, making the temperature, pressure and density decrease at the same time that velocity increases.

It has to be empathized that the increase in Mach number and velocity is soft and, as the variations of temperature, pressure and density are also infinitesimal, we can consider the flux as isentropic.

This flux has a leading edge with which creates an angle and a trailing edge with which the final flux forms an angle . These angles are the Mach angles and, as their name already indicates, they are related with the Mach numbers at the beginning () and final () with the following expressions:

## 2.2 Application of the expansion wave to a practical case

This problem is of great interest to us in the aeronautical industry for the design of wing profiles, since practically all aircraft move within supersonic flows (due to the high speeds they reach) where the wing of an aircraft in question acts in a similar to the convex corner of the Prandtl-Meyer model. The flow at the wall (in this case the airfoil) must be tangent to it; hence, the streamline at the wall is deflected downward through the angle θ. Therefore, the airflow will maintain its direction parallel to the wing surface at the leading edge and it will be when, upon reaching the trailing edge, the flow direction will change producing an expansion wave and accelerating the air behind the wing while the pressure, temperature and density will decrease continuously.

The physical mechanism by which these waves are created in this environment has to do with the propagation of a sound wave through molecular collisions. In a gas, information is transmitted from one point to another by sound waves that propagate in all directions away from the origin of the disturbance.

Considering a body within an air flow, the molecules that collide with the surface of the body undergo a change in momentum. This change is transmitted to its neighboring molecules through molecular collisions and is thus transmitted throughout the entire flow. Therefore, the information is propagated upstream at more or less the local speed of sound. If the upstream flow is subsonic, the disturbances have no problem by working their way far upstream, thus the incoming flow will move out of the way of the body but, if the upstream flow is supersonic, the disturbances cannot work their way upstream and, at some finite distance from the body, the disturbance waves will come together forming a standing wave in front of the body.

Therefore, the physical generation of waves in a supersonic flow is due to the propagation of information (physical properties of the flow such as speed, pressure, temperature ...) through molecular collisions and due to the fact that this propagation cannot travel correctly in certain regions of the supersonic flow.

# Description of the math’s involved

The objective of this project is to simulate and obtain a numerical solution of a flow over a Prandtl-Meyer expansion corner.

In this simulation, we are going to suppose that the flow that moves on the surface is two-dimensional, supersonic and invisible and we are going to establish a series of initial conditions in order to simplify the problem and obtain the 5 properties of the flow that interests us to study its behavior: velocity (both vertical and horizontal), Mach number, density, temperature and pressure. For that purpose, we will solve the problem numerically using the MacCormack’s predictor-corrector explicit finite-difference method solution technique.

## Physical equations

The governing Euler equation for a fluid with the mentioned properties can be expressed as:

Where F and G are the flux variables, two columns of vectors that contain different values ​​of the properties that we want to study to describe the behavior of the fluid, defined as in the following equation.

Moreover, assuming a calorically perfect gas, we can use the next relation between the energy and the temperature of the gas in favor of the pressure and density of the fluid:

, where is the specific heat at constant volume, a constant that measures the amount of heat that a gas can release or absorb during a change on temperature keeping constant the volume. Introducing this condition we obtain the following equations for F and G.

Where are the density, horizontal speed, pressure, vertical speed and module of the speed, respectively.

As had be seen in equation , vectors F and G are related to each other thanks to the Euler equation for a stationary and two-dimensional flow as follows:

So we can assume that if we know the flow variables at a point as a function of y (initial data line), the y derivative of G is known along this entire line and therefore the x derivative of F is known (from where we can extract F). Therefore, the problem can be solved by making Δx jumps along the x-direction of the flow and calculating the next line above at each step.



Figure 2. Model for the downstream solution

To obtain the so-called primitive variables (, u, v, p, T) we must perform a decoding task of the flow variables in the following way:

And knowing the Fs and all the primitive variables we can obtain the Gs in order to calculate the next point (at ):

To end-up with this section, note that, since the grid where the problem develops (the physical grid) is not completely rectangular, we must transform it into a fully rectangular computational plane. That is why we introduce the variables and . In a plane, denotes horizontal step and denotes vertical step. The conversion from x,y coordinates to coordinates is as follows:

Where is the y location of the lower surface and h(x) is the local height from the lower to the upper boundary in the physical plane. So, now transforming the Euler equation into coordinates:

Writing the column vectors F and G with these partial derivatives we obtain a system of equations, which physically are the equations of conservation of mass (continuity equation), conservation of momentum and conservation of energy.

* Continuity equation:
* X-momentum conservation equation:
* Y-momentum conservation equation:
* Energy conservation equation:

## MacCormack’s technique

As we said before, we will use the finite-difference method of the MacCormack’s technique in order to numerically solve the problem. This method is represented in the following general equation: , where we discretize A with respect to b. In our case, A will be both of the flow variables (F and G) and b will be the or components of the computational plane.

From the previous conservation equations we can set up an explicit finite-difference solution for the two-dimensional flow by the MacCormack technique from the space derivatives isolated from the conservation equations of mass, momentum and energy written in terms of forward differences:

Next, with the previous equations and following the same explicit method in space, we can obtain the predicted values ​​of the fluid properties, which can be identified with a bar above each property:

## Predictor-Corrector method

In the previous point, what we have obtained are predicted values ​​since they do not have an accuracy of second order. Therefore, to improve the accuracy of our results, we must implement a predictor-corrector method in our calculations with which we will obtain the final value of each one of the flow properties.

The Predictor-Corrector method uses the same general equation mentioned from the MacCormack’s technique with explicit finite-differences but using an average derivative that corrects the predicted values of the used method. The name "average derivative" comes from the fact that the derivative in question comes from averaging two derivatives, that of point i, j of the mesh with that of point i + 1, j of the same:

Where the derivatives on the right-hand side of the equations are known numbers; as it can be written analog to the predictor step equation:

And as we already have the values, the can be computed as had be done at the beginning with the primitives

In this equation we are considering the space derivative of the conservation equations in a point and the time derivative of the equations by using the MacCormack’s technique in the next interval of space (point). Finally, we obtain the following equations:

## Artificial viscosity

To finish with the calculation of the flow properties, we just have to introduce one more aspect to take into account in our calculations, the artificial viscosity. This is because, when solving the problem using finite differential equations, the change in the calculation of the flow properties for one point and the next is discontinuous. These discontinuous changes can introduce oscillations in our numerical solutions and to eliminate them, we have to formulate this artificial viscosity as follows on the predictor step:

Where n can be 1, 2, 3 and 4 and refers to each of the Fs.

Therefore, adding these SF values ​​to our previous equations we have:

Artificial viscosity is also added on the corrector step in the following way:

From where we obtain the following flow properties:

And the primitive variables can be decoded from the previous values.

## Boundary conditions

Last but not least, a boundary condition must be applied to the system: tangent flow to the wall. Following the Abbett’s boundary condition treatment, we first calculate the u and v values at the wall and compute the direction of the resulting velocity (Ф) as tan-1(v/u) and the Mach number.

With the formulas previously seen, Pcal, Tcal, ρcal can be obtained. Once we have the calculated values Pcal, Tcal, ρcal we must compute the actual values. They can be computed as:

However the Mact is not defined yet and it must be obtained by trial and error. As Mcal is known it can be computed the fcal with the formula:

And as fact=fcal + Ф it can be solved the value of fcal. With this value it is proposed a trial and error resolution yet we are using a numerical method (secant).

With the secant method two initial random but realistic values of Mach must be defined and the tangent line is computed. Then, it is evaluated the value when the tangent line is 0 and this new point, and the last of the previous Mach points defined, is used to compute the next tangent line to find the new value of 0. This is done iteratively until a value is found, defining some tolerances.

With this method a good approach of Mach is obtained and all the remaining actual parameters can be obtained.

# Description of the simulator

The simulator developed aims to compute the Prandtl-Meyer expansion wave flow field (in the way we have explained in the previous “mathematics” section).

With the previous equations calculated for the initial conditions (initial state of the flow), all the flow properties can be calculated from now on and displayed graphically in a series of tables and graphs that we will comment on later in this report for a better understanding of the physical problem analyzed.

In order to make a comprehensible simulator and user-friendly, the code has been organised in diverse classes and windows.

## 4.1. Classes

The “Classes” part of the code is the part where we calculate all the equations involved in obtaining the parameters that describe the behavior of the flow. All the discretized equations according to the corresponding method (in this case the Predictor-Corrector with the MacCormack technique), the artificial viscosity and the boundary conditions are implicit in this section.

The principal classes of the program are:

* Normas. In this class there are all the parameters defined by the user; from the initial data line to the properties of the physics domain. The values of this class can be modified according the user’s will through the simulation window.

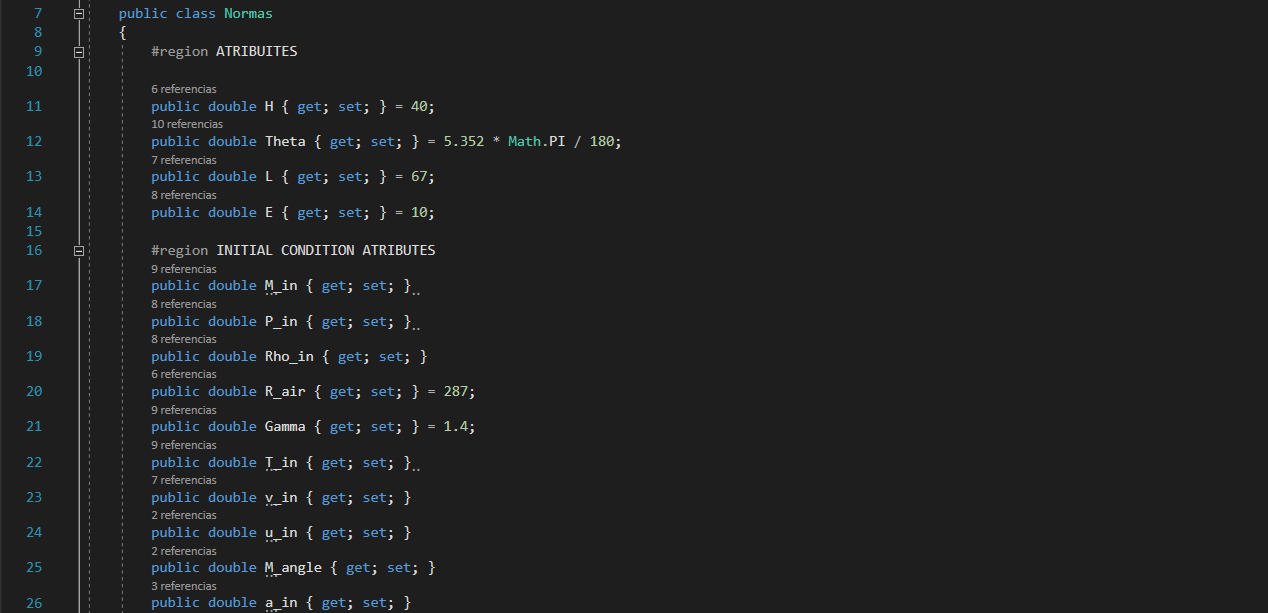


Figure 3.1.1 Normas class attributes

This class has also a method that allows us to compute the rest of initial conditions attributes:

Figure 3.1.2 Computation of other initial conditions attributes

This class will be used by the rest of the classes discussed later since it is where all the numerical information of the parameters and variables with which we will play to obtain the resolution of the problem is contained.

* Celda. This class is where the most important functions are defined. The computational field is converted in a grid hence a set of cells are then defining the ensemble. Each cell has its own properties of temperature, velocity, density, x and y position, F and G values, intermediate predicted values, etc. These properties are the attributes of the Celda class. Besides, in this class can also be found the mathematical and computing functions that enable computing the attributes of the neighbouring cell.

In addition, each cell can be represented as a rectangle of a given dimensions and colour, relying on its location in the physical plane and the value of its attributes. Because of that, in the Celda class are also defined this aspects.

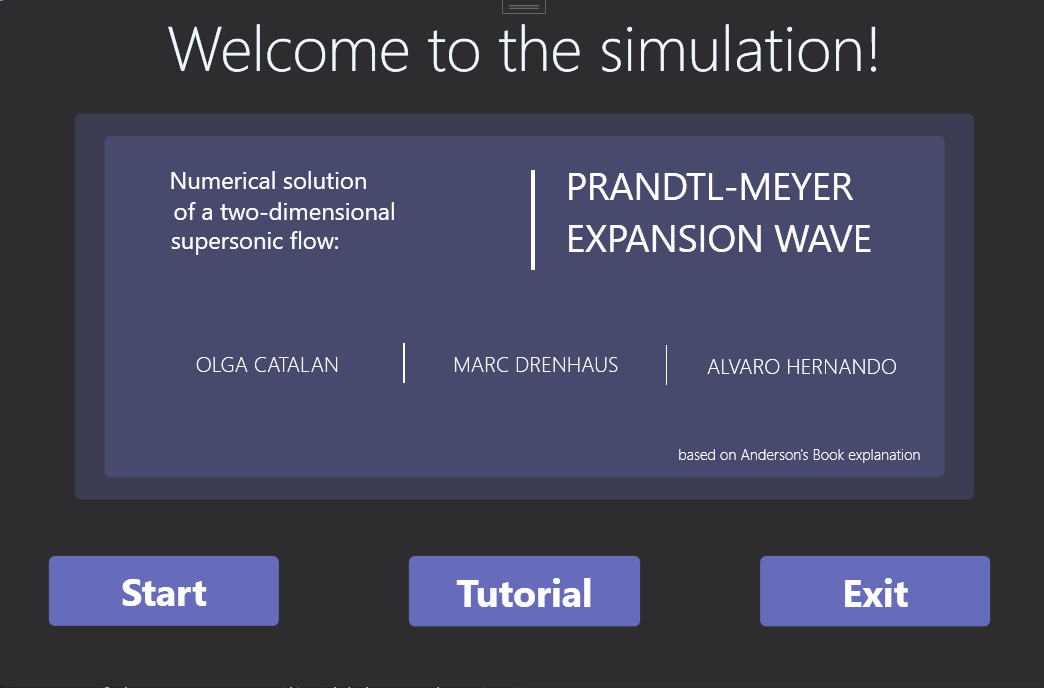
* Malla. This class represents the assembly of cells and the interaction between them. Is where is found the function that computes all the flow properties, using the functions defined in the Celda class. Furthermore, DataTables are created gathering all the results and are filled with the most interesting values that are going to be shown in the windows.

Other secondary classes are created as support of specific functionalities. For instance, for generating the plots a class of points has to be defined.

## 4.2. Windows

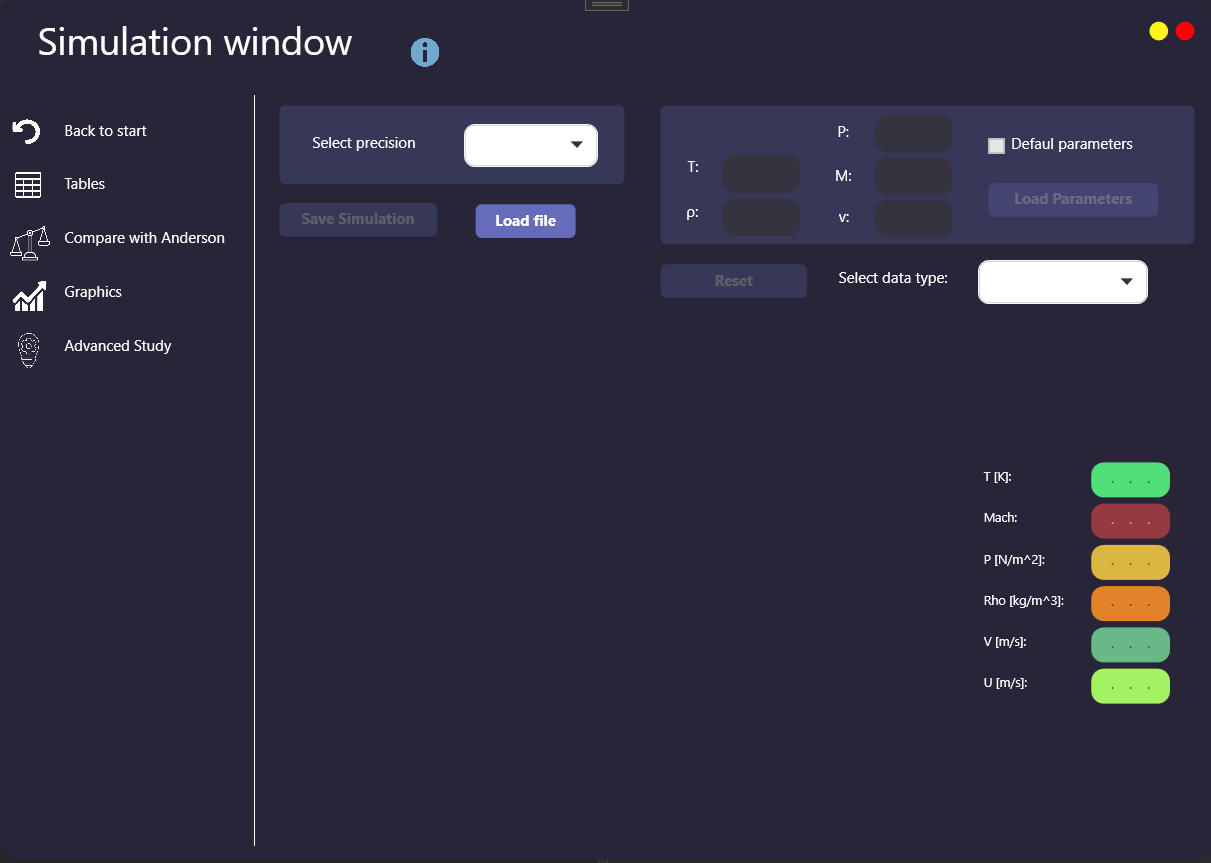
The visible part of the simulator and the one that the user is going to interact with are the windows. Because of that, with the goal of provide a friendly and understandable graphic interface, a simple and intuitive design is applied.

First, when we initialize the program a welcome window pops up.

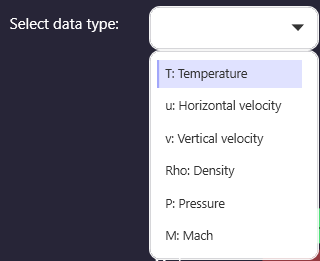


This first window is the first contact with the simulation and because of that it allows the user to access a tutorial of use. For this, it consists of three buttons: the "Start" button by which, by pressing it, the user is directed to the window where the problem simulation is carried out. The "Tutorial" button with which we can access a video tutorial on how the simulator works and the "Exit" button to close the simulator.

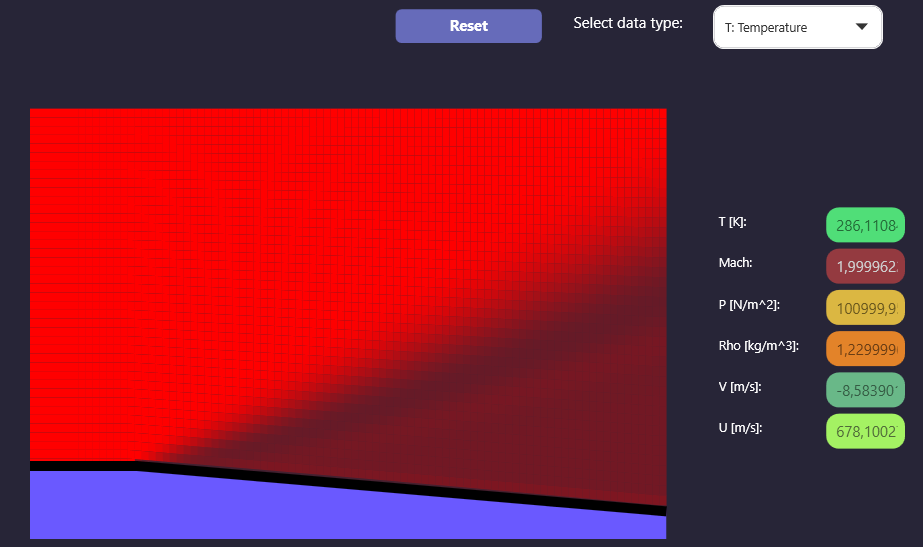
The next window is the simulating one. In this window is where we find the main utility of the program: the computation of the Prandtl-Meyer expansion wave flow field.



To begin with, we find the following interface. In the central part we have different buttons that will allow us to select the way in which we want to carry out the simulation and, once all the parameters involved in solving the analyzed problem have been selected, a visual grid-like representation can be observed of each one of the attributes of the flow. This attribute represented can be changed to visualize another set of values with the following ComboBox:



Where, as we can see, we can choose between displaying the numerical representation of the evolution of Temperature, Horizontal Speed, Vertical Speed, Density, Pressure or Mach number throughout the entire domain of the fluid. Here is an example screenshot of what the Temperature representation would look like:

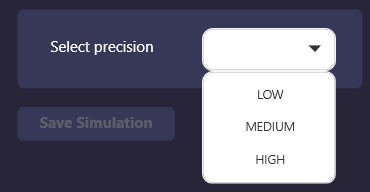


As we can see, we have also implemented some labels to the right of this window where each one of the attributes of each specific cell that we pass over with the mouse are represented.

In addition, a Reset button allows us to reset the entire simulation and clear all the parameters in order to select new ones and start a new simulation from the beginning.

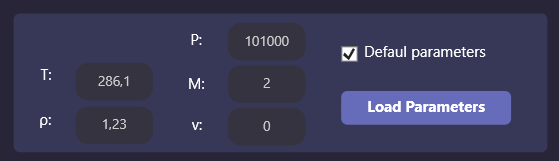
Let us now comment on the buttons that will allow us to control the way in which we want to carry out the simulation.

The first thing we must select before starting with the simulation is the precision of the grid: LOW, MEDIUM or HIGH depending on how we want the precision of the grid with the results of the simulation to be for each attribute:



Depending on the precision, the grid size is going to variate.

In the upper right part of the window, we must enter the initial parameters of the flow with which the simulation of the problem will be carried out. We can choose between entering all these parameters by hand or using the default parameters which are the following ones:

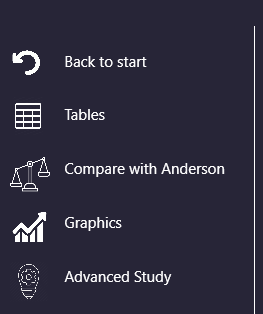


Once these parameters have been selected, we must press the "Load Parameters" button to carry out the simulation.

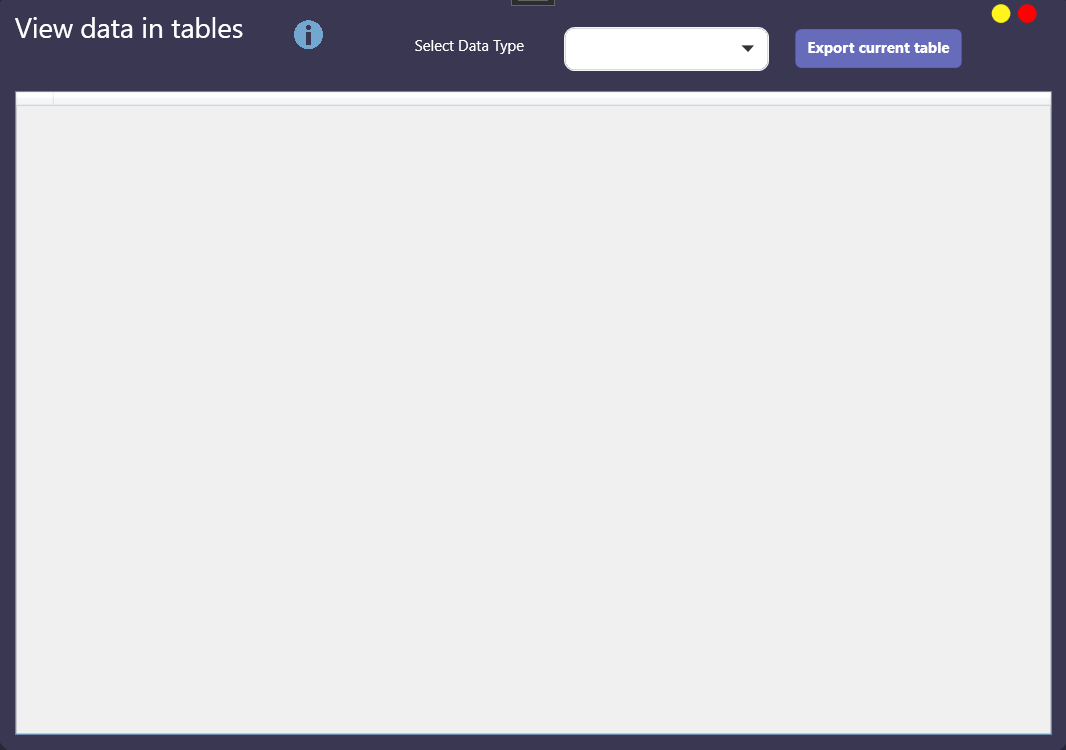
We also have the simulation load and save buttons with which we can load files with previous simulations to continue analyzing them or save the current simulation to analyze it later:



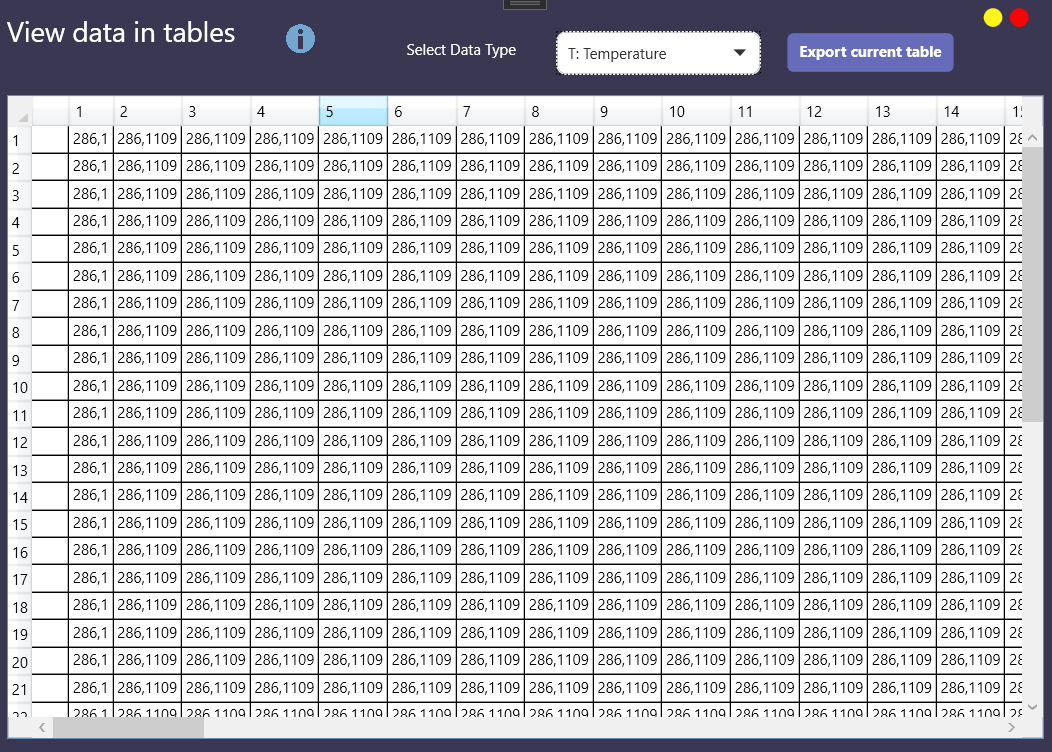
This window also has a column of buttons on the left with various additional functions. The "Back to start" button allows us to return to the initial window where we could access the tutorial for using the simulator and exit it. The "Tables" button opens a new form with tables with all the numerical results of the simulation. The "Compare with Anderson" button opens a form that allows us to compare the results of our simulator with the results that appear in the Anderson tables. The "Graphics" button opens a form with the plots of the evolution of each attribute and finally the "Advance Study" button with the results of the advanced study simulation. All these mentioned forms will be discussed in depth next.



The next window is the tables of velocity, temperature, pressure, etc. These variables that we had observed in the visual grid-like representation are now shown in tables enabling knowing its specific value in each position of the field.



With the ComboBox we can choose the data that we want to display in the tables. For example, in the case of the Temperature the tables have the following shape:



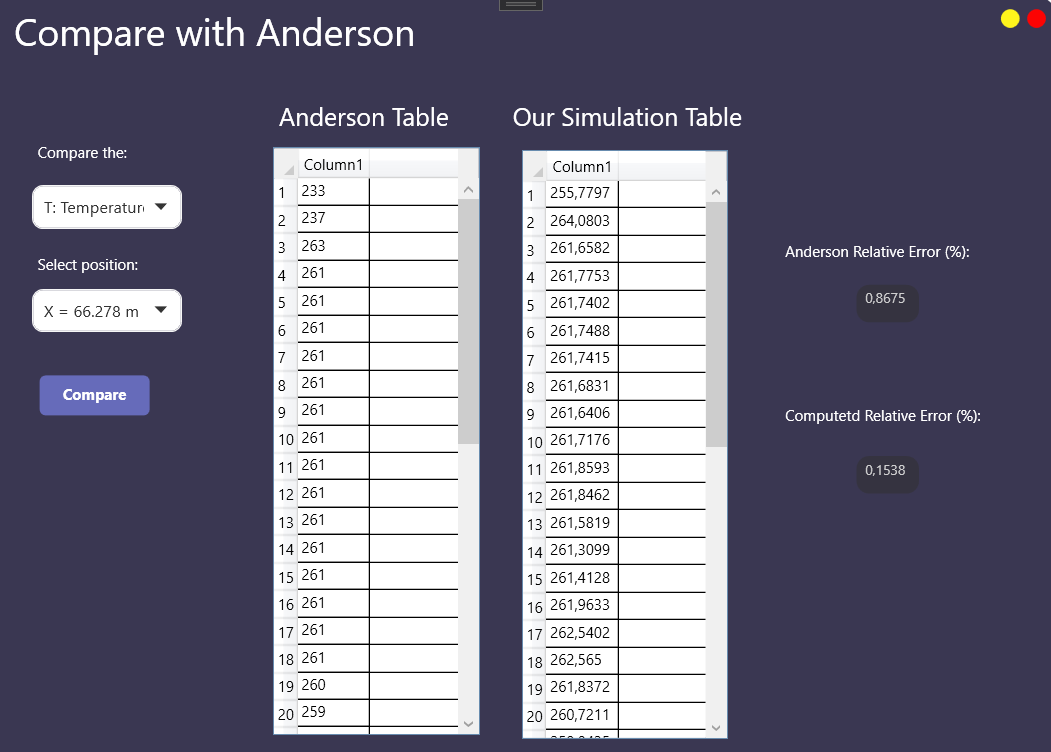
This table results can be exported by pressing the export button, so any further investigation, or simply save it, can be done.

Then, the following window enables checking the validity of the results.



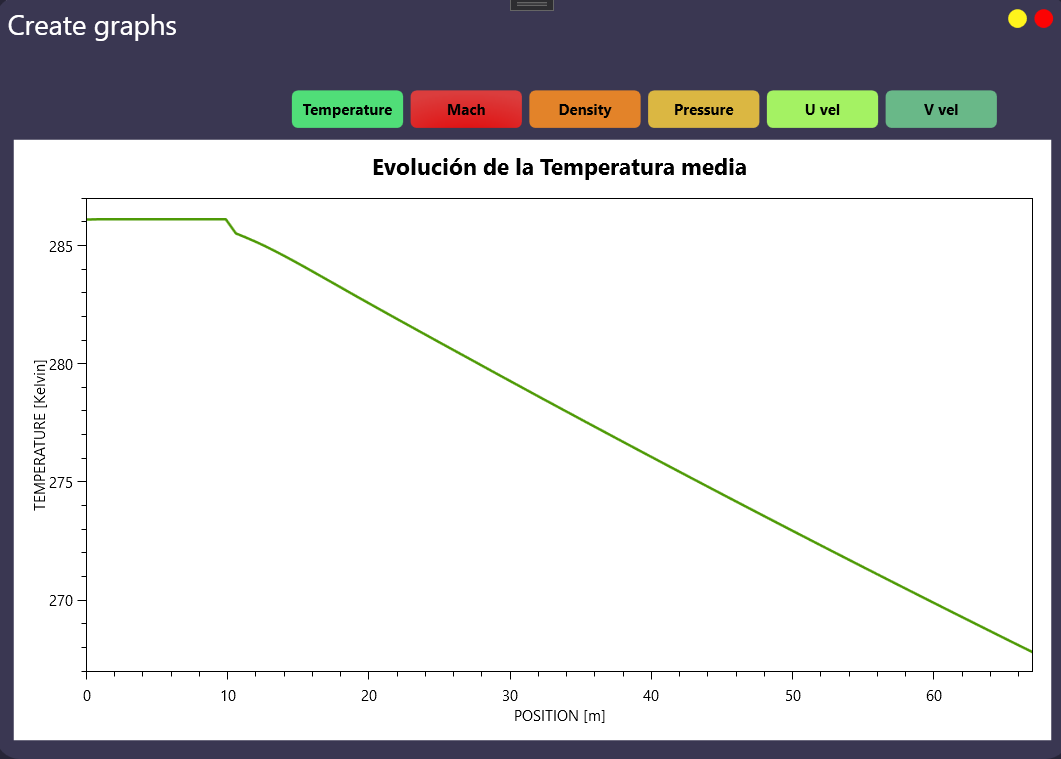
Here can be compared the Anderson results with the specific columns computed by the simulator. Moreover, some further study is done analysing the variation of the results.

To compare these results, we will select with the ComboBox above the attribute we want to study and with the ComboBox below the position (upstream or downstream) where we want to compare it with the result of the Anderson tables. For the temperature, the tables present the following shape downstream:



In this position, as we can see, we can also study the Relative error of the Anderson and the computed results with respect to the analytical results.

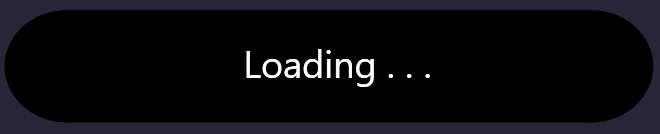
The next windows we have are the plots and the advanced study. In the plots window is where can be observed the evolution of the parameters in the field and interesting results of the simulator. For example, for the case of the temperature we can study the following graph representing the evolution of it along the fluid domain:



In the advance study is where it is found in what consists the advance study of the project, accompany with visual results.



Finally, special attention has been paid in the robustness of the windows, so potential mistakes of users do not provoke a fatal error that disables the whole simulator. In addition, to indicate that the code does not hang, each time the simulator takes lot of time to present the simulation data (because it has to analyze a very large data grid and there are a huge number of operations to analyze) it appears on screen the next message:



## Challenges and decisions

One of the biggest challenges in the development of the simulator was the adaptation itself of the Prandtl-Meyer expansion wave flow field, as many complex equations and functions are considered for the computation.

To overcome the difficulties led of the computation, a MATLAB code was initially done and online research was done to check different implementations. Once the MATLAB was fully functional and the results matched with the Anderson tables, the code was transferred and adapted to C#.

Another significant challenge was the advance study, as no verifications could be done to ensure it was correct. To solve this challenge the advance study problem was divided in smaller problems and they were solve it separately and then gathered to obtain a solution.

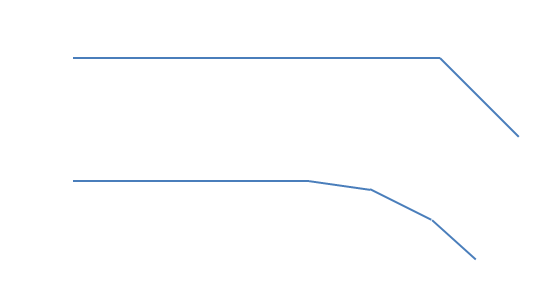
Finally, the results representations have been done in two different ways; the first one in the same window where the simulator computes the flow field. This first representation corresponds to the physical plane representation and different colours representing the variation of the temperature, density, etc. This first representation of results was selected as is a visual but effective way of obtaining a general knowledge of how the fluid behaves. Additionally, in the next window, tables with the results are obtained. This other way of visualising the results allows more specific values and also usher into further studies, as this results can be exported and then can be handled by the user for further studies.

# Advance study

As has been explained, this problem is of great interest as since practically all aircraft move within supersonic flows and several aircraft geometries can be modelled as a convex corner of Prandtl-Meyer model, for example the wing.

Nevertheless, the geometry represented of a convex corner is not much accurate with the geometries found in aircrafts; this geometry has softer changes.

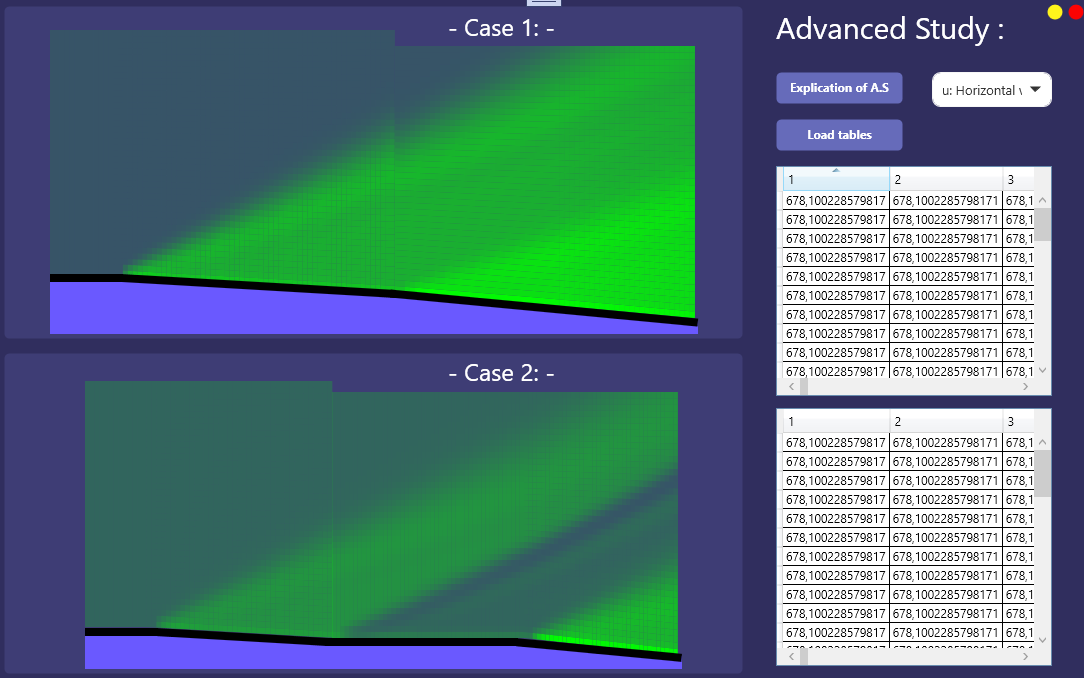
Because of that, the advance study is intended to analyze a way of approximating these curved surfaces with the Prandtl-Meyer model. These soft changes can be better approximated by several inclined planes rather than one unique change, as observed in the following image.



This case can be obtained by first solving the Prandtl-Meyer model with a certain angle and then setting the final results obtained of this first computation as the initial data line of the following computation. In this next computation the expansion corner position is set as small as possible, so the general view of both cases together is a flat plane followed of two inclined planes.

Finally, it is also being studied the effect of adding stabilization periods between inclined planes.

Simulating these cases, the following results have been obtained.

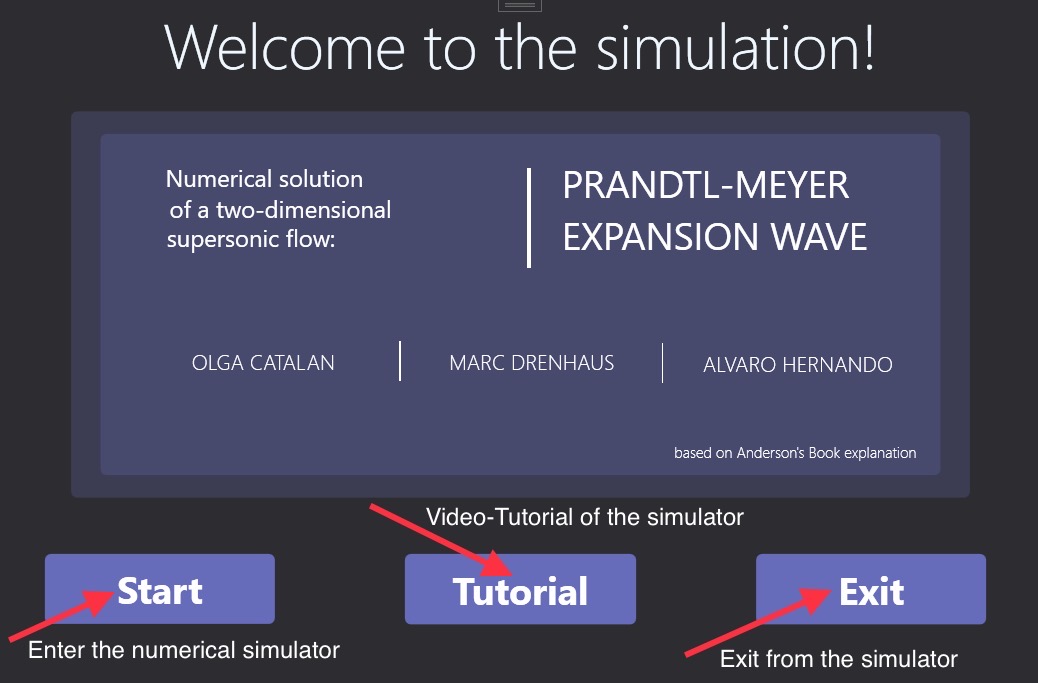


In case 1 it has been simulated two inclined without a stabilization distance and in case 2 with stabilization. Additionally it can be consulted the tables with the exact values of the simulation.

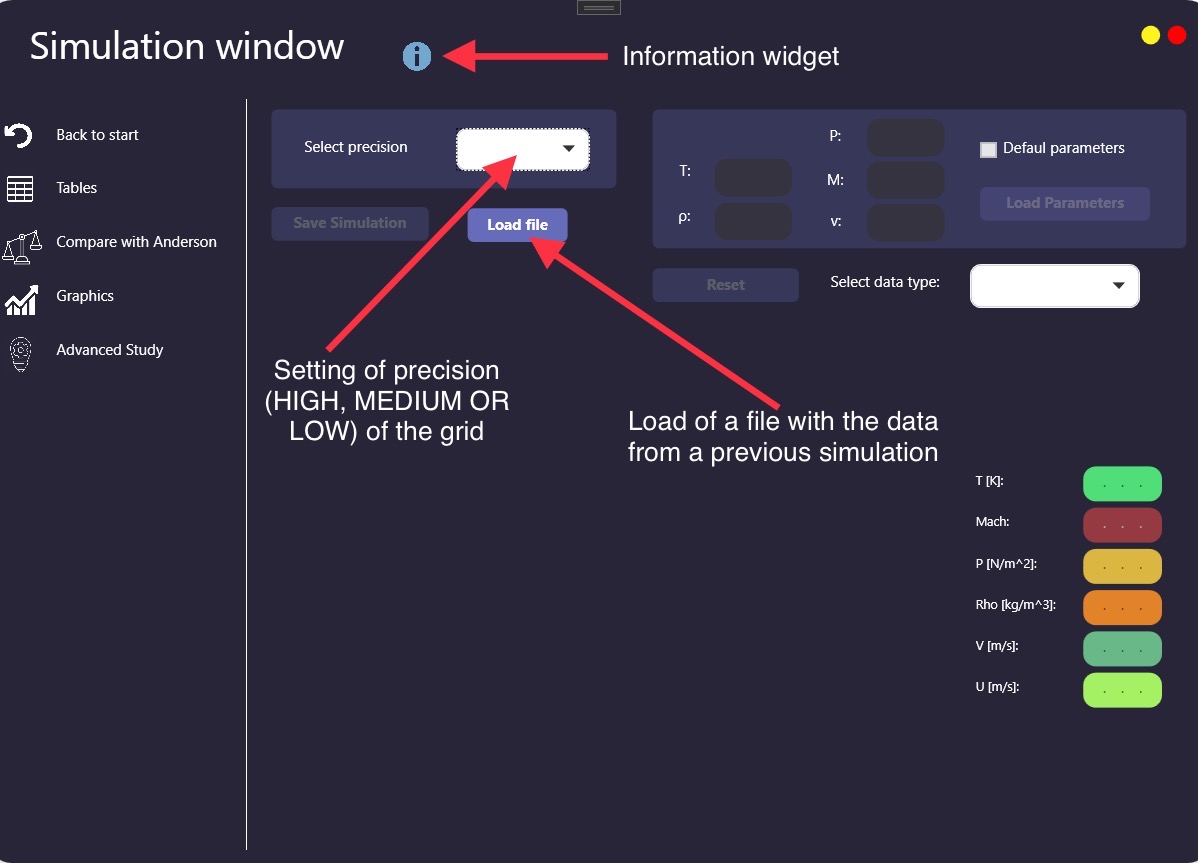
# User manual

The objective of this basic manual is to facilitate user interaction with the simulator of the supersonic two-dimensional flow along an expansive wave. This manual will be accompanied by a short video where the correct operation of the code will be demonstrated through a short tutorial.

When executing the code, the first screen appears through which we can enter the simulation:

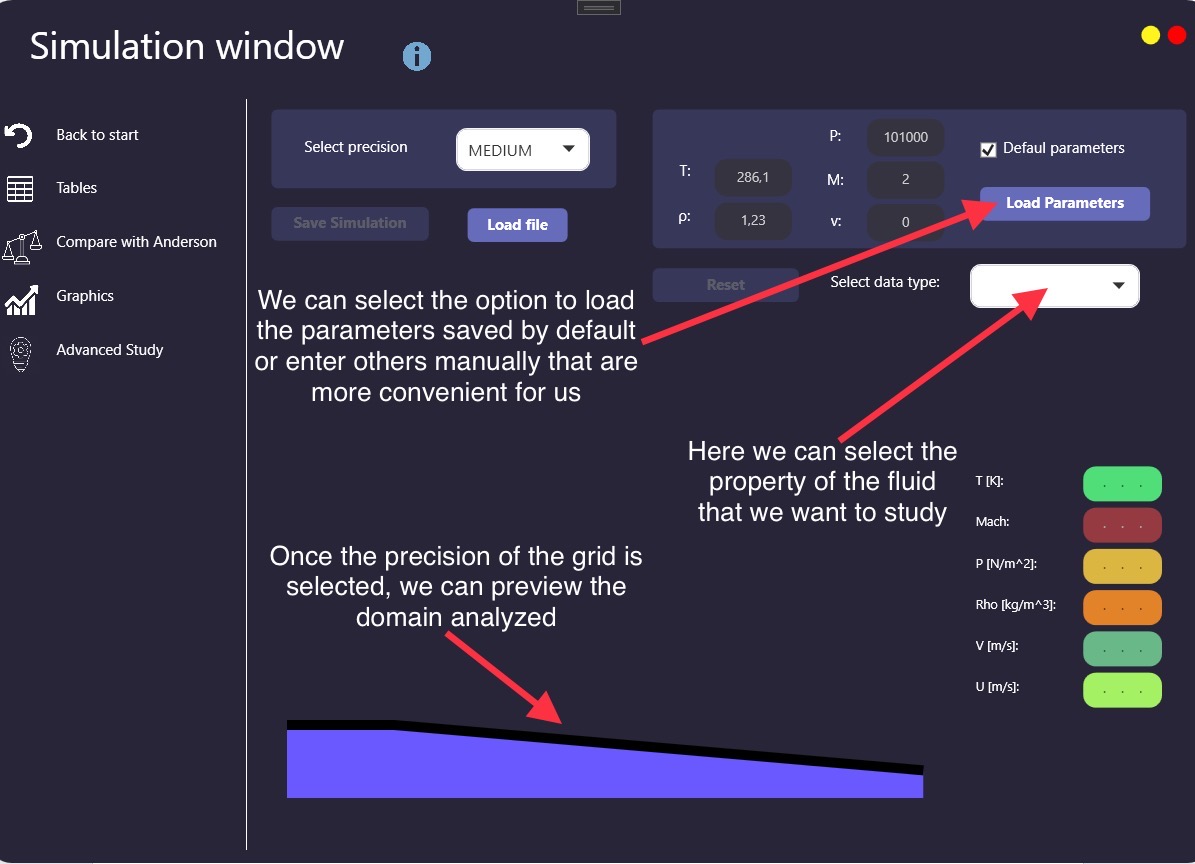


Once inside the simulation, the following window appears with these buttons:

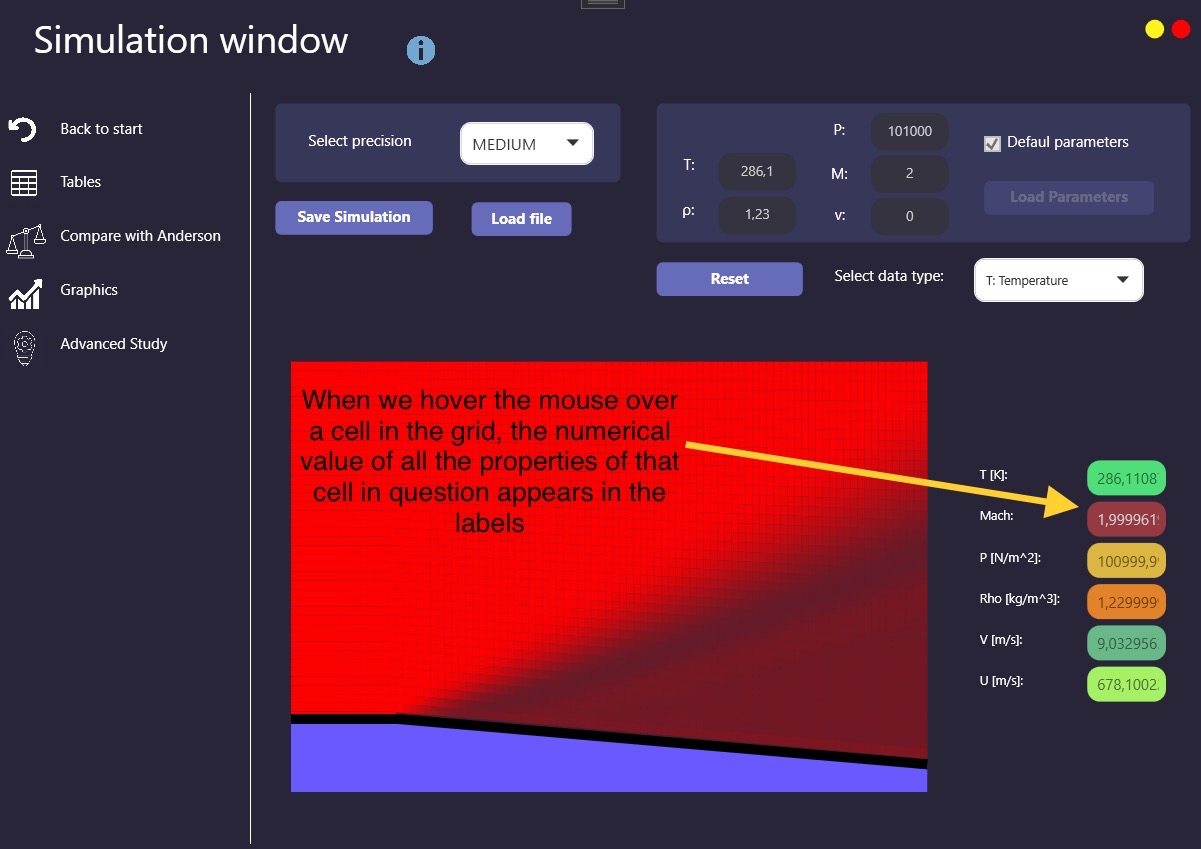


The first thing we have to do is define the precision of the grid where the evolution of each of the fluid attributes will be represented throughout the shock wave or load a file with the parameters of a previous simulation.

After that, we should select the parameters involved in the numerical simulation of the problem and charge them into the simulator.



Once all the simulation parameters have been loaded, we select the type of data that we want to visualize in the grid with the ComboBox.





# Bibliography

[1] J.D. Anderson, Chapter 8: “Numerical solution of a two-dimensional supersonic flow: Prandtl-Meyer Expansion Wave” from Computational Fluid Mechanics, 1995. [Book].

[2] “High-Speed Aerodynamics – Shock Waves”, 2021. [Online]. Available in:

<https://desdelacabinadevuelo.com/2017/11/21/que-ocurre-cuando-un-rayo-impacta-en-un-avion/>

[3] “Oblique Shock and Expansion Waves”, February 2016. [Online]. Available in:

<http://heli-air.net/2016/02/25/oblique-shock-and-expansion-waves/>