

Trans-sonic Flow

The flow between Mach number 0.8 and 1.3 is generally defined as transonic flow, which is widely found in civil aviation vehicles. The purpose of this example is to simulate the aerodynamic characteristics of a transonic airfoil, which will give the user an idea of the calculation process and the configuration of the transonic parameters.

3D

1 Import Mesh

Click the *New Project* button in the menu bar "File" to open the *Create a project* window, select the Preferences as *Import Mesh*, select the mesh file named *inviscid_ONERA.su2* from the installation path, set the project name and location, and click the *OK* button to finish creating the project.

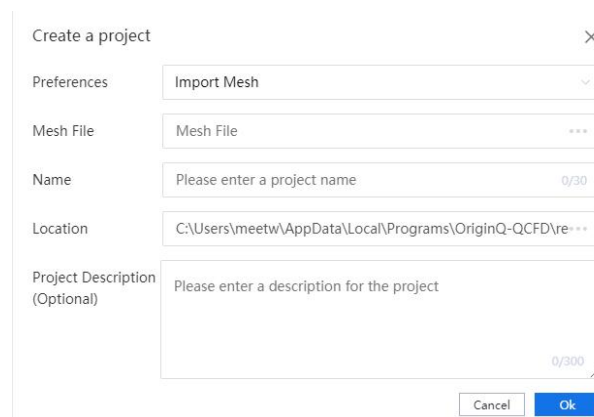
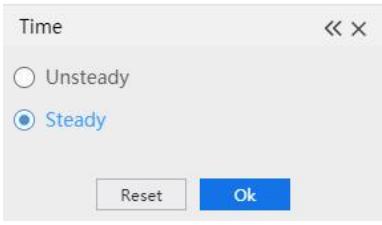
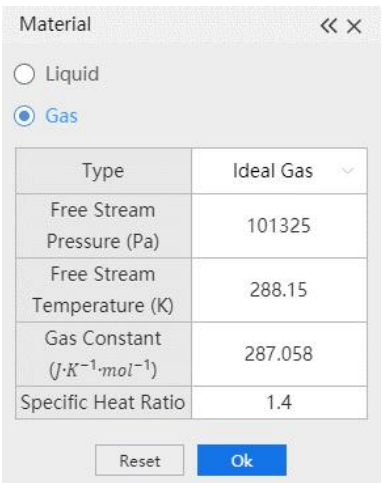
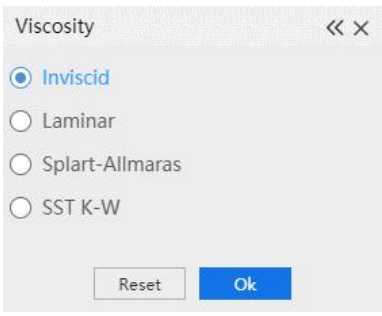
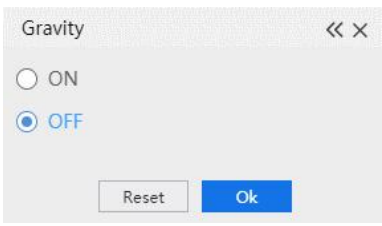
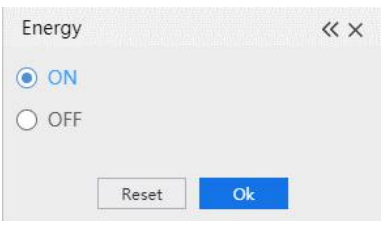


Figure1 *Create a project*

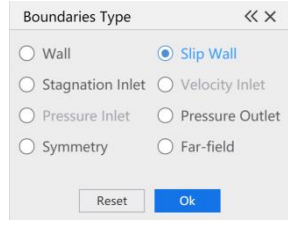
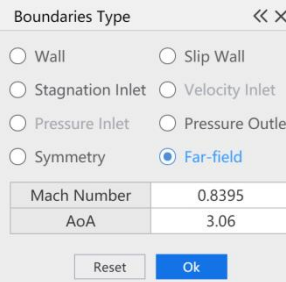
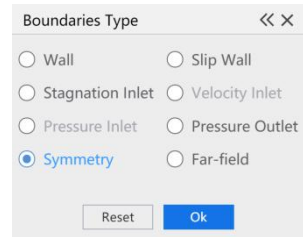
2 Physcis

Table1 Option and configuration of Physcis

option	configuration	option	configuration
Time		Material	
Viscosity			
Gravity		Energy	

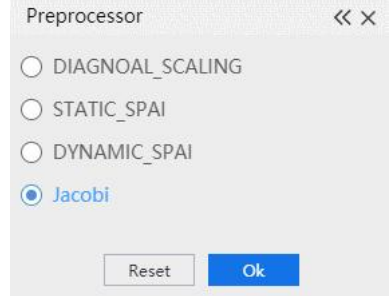
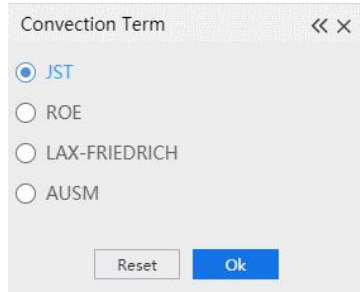
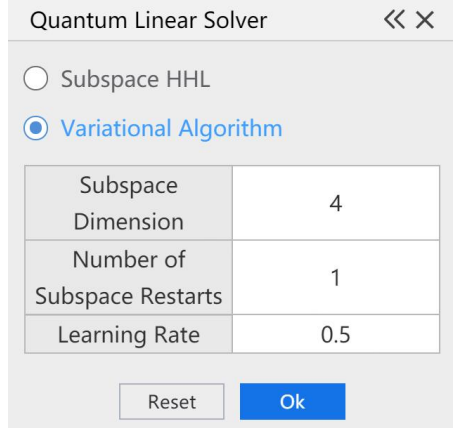
3 Boundary Type

Table2 Boundary type setting

boundaries name	LOW_SIDE、TIP、UPPER_SIDE	XNORMAL_FACES、YNORMAL_FACES、ZNORMAL_FACES	SYMMETRY_FACE
Boundaries Type			

4 Numerical Method

Table3 Option and configuration of Numerical Method

option	configuration	option	configuration
Spatial Gradient		Preprocessor	
Convection Term		Quantum Linear Solver	

5 Control Parameters

Table4 Configuration of Control Parameters

parameters	CFL Number	Relaxation Factor	Convergence Error	Maximum Iteration Steps
configuration	25	0.5	0.00001	5000

6 Report

Click the *New Report* button to create a report named Airfoil_Lift, select the type as *Force Coefficient*, and select UPPER_SIDE as the *Object*, the *Direction* as y, and the *Reference Area* as 1. This report is used to

quantitatively analyze the relationship of the lift coefficient of the wing with the iteration step.

Click the *New Report* button to create a report named surface, select the type as *Surface data*, and select LOW_SIDE、TIP、UPPER_SIDE as the *Object*. This report is used to analyze the trend of various physical quantities on the wing surface.

7 Compute

Click the "*Start Simulation*" button in the menu bar "*Compute*" to do the simulation. The *Local Host* is the default choice for the computing backend, and *Cluster* and *Quantum computer* will be supported in subsequent releases.

8 Result

This example will create diagram, pressure scalar and streamline in the results to analyze aeromechanical properties and flow field distribution of the wing. Hover over the *Result* option, click on the *Result* button  , select the Type as *diagram*, *Data Type* as *Force Coefficient*, and select the report created in the *Report* named Airfoil_Lift as the *Data Source* to create a diagram representing the lift coefficient of the wing airfoil with iteration steps.

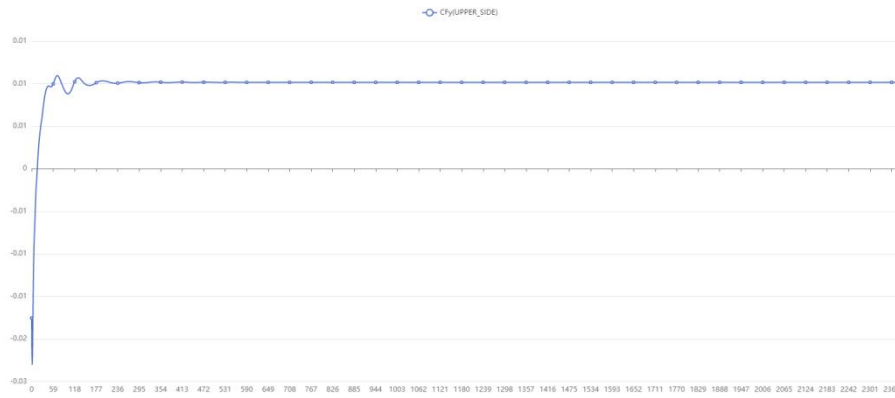


Figure2 Airfoil lift coefficient diagram

Hover the mouse over the *Result* option, click on the *Result* button , select the *Data* as *Surface*, select the *Type* as *Scalar*, click the *Ok* button, and then click the created scalar below the *Result* to open the scalar scene in the view window after the scalar rendering is complete. In the function button area on the left side of the view window, click the *Property* button , and in the dialog box that pops up in the upper right corner of the view window, select the *Physical Quantity* as *Pressure_Coefficient* and the *Data Range* as -1.2-0.6.

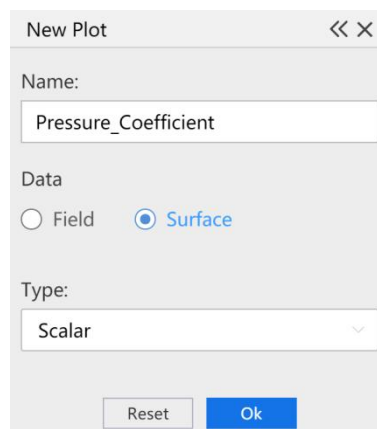


Figure3 New Plot



Figure4 *Property*

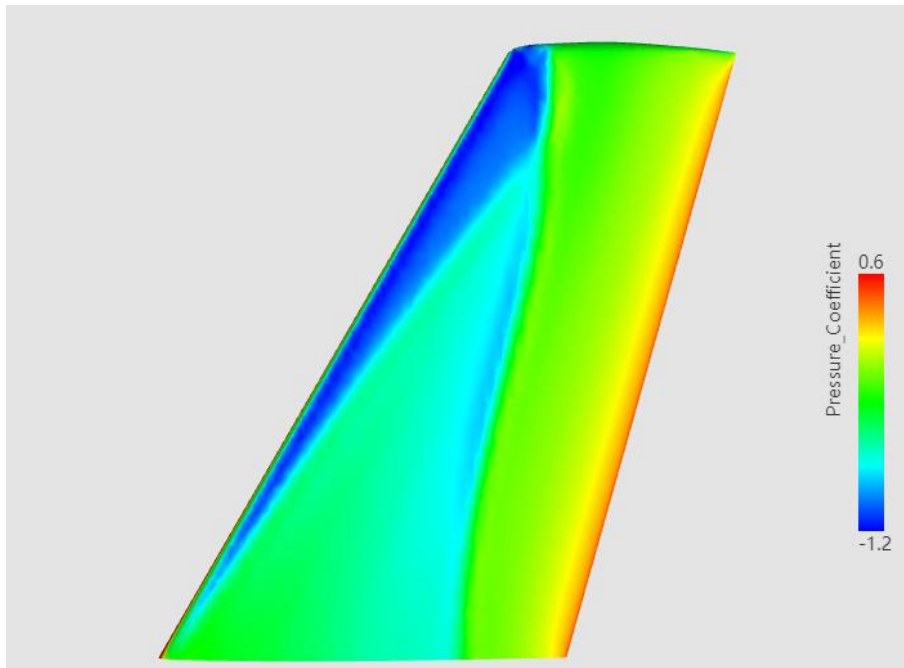


Figure5 *Pressure_Coefficient scalar*

Hover the mouse over the *Result* option, click on the *Result* button , select the *Data* as *Surface*, select the *Type* as *Scalar*, click the *Ok* button, and then click the created scalar below the *Result* to open the scalar scene in the view window after the scalar rendering is complete. In the function button area on the left side of the view window, click the *Property* button , and in the dialog box that pops up in the upper right corner of the view window, select the *Physical Quantity* as *Mach* and the *Data Range* as 0.5-1.5.

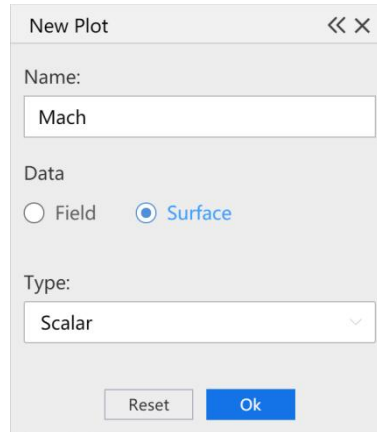


Figure6 New Plot

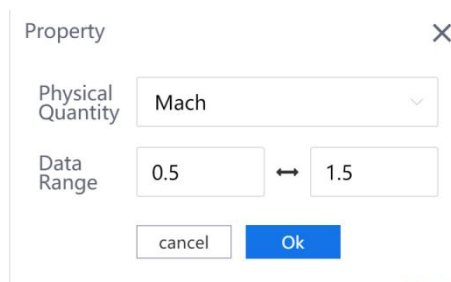


Figure7 Property

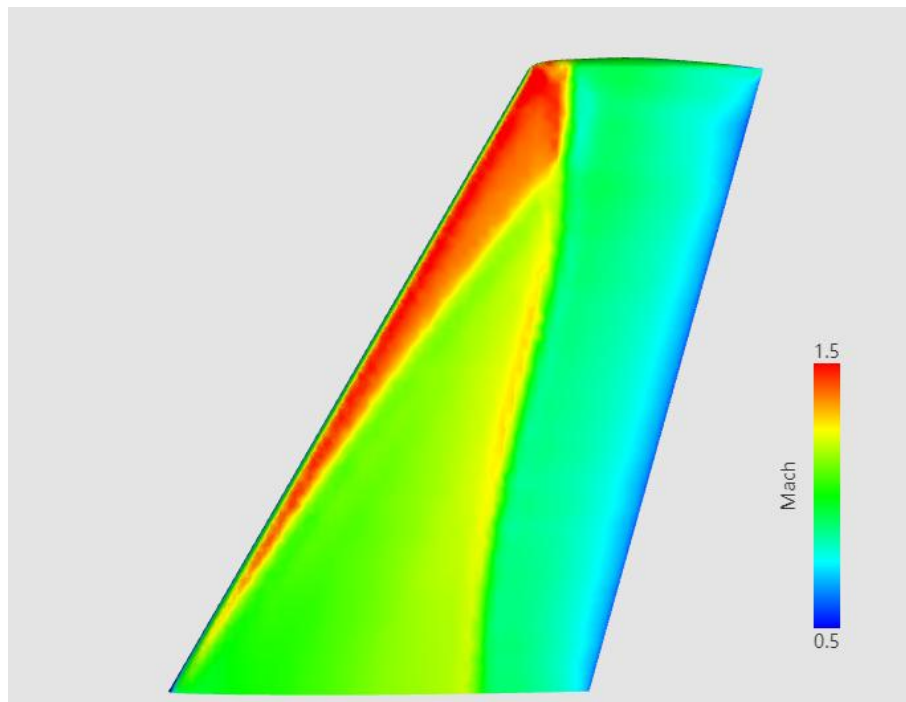
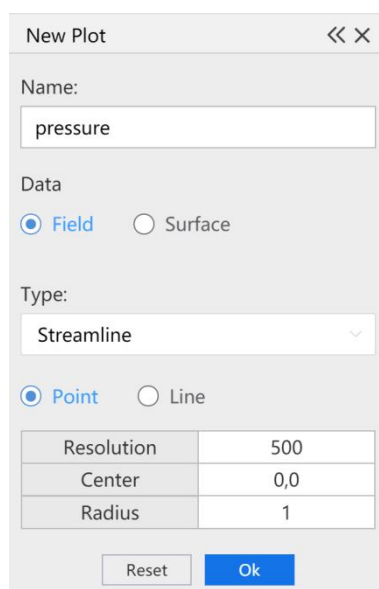


Figure8 Mach scalar

Hover the mouse over the *Result* option, click on the *Result* button , select the *Data* as *Field*, select the *Type* as *Streamline*, choose the fluid mass arrangement as "*Point*", the *Resolution* as 500, the

Center as (0,0), the *Radius* as 1, click the *Ok* button, and after the contour is rendered, click the created contour below the *Result* to open the streamline scene in the view window. In the function button area on the left side of the view window, click the *Property* button , and in the dialog box that pops up in the upper right corner of the view window, select the *Physical Quantity* as *Pressure* and the *Data Range* as 20000-130000.



The 'New Plot' dialog box contains the following settings:

- Name:** pressure
- Data:** ☒ Field ☐ Surface
- Type:** Streamline (dropdown menu)
- Point/Line:** ☒ Point ☐ Line
- Resolution:** 500
- Center:** 0,0
- Radius:** 1
- Buttons:** Reset, Ok

Figure9 *New Plot*



The 'Property' dialog box contains the following settings:

- Physical Quantity:** Pressure (dropdown menu)
- Data Range:** 20000 ↔ 130000
- Buttons:** cancel, Ok

Figure10 *Property*

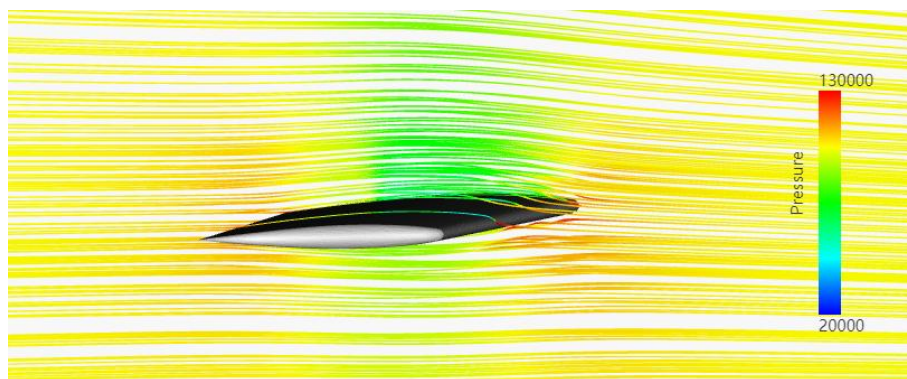


Figure11 *streamline*