

Hypersonic Flows

Flows with Mach number greater than 5 are generally defined as hypersonic flows and are widely found in aerospace applications such as medium and long-range ballistic missiles. The purpose of this example is to simulate the aerothermal characteristics of a hypersonic warhead, through which the user can gain an understanding of the computational process and the hypersonic parameter configuration.

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 *hypersonic_3D.su2* from the installation path, set the project name and location, and click the *OK* button to finish creating the project.

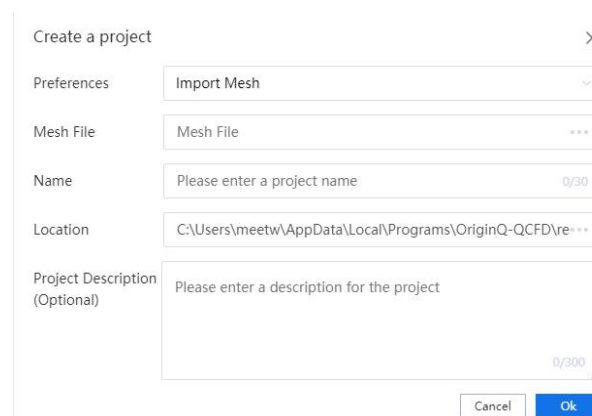
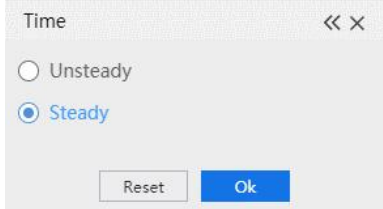
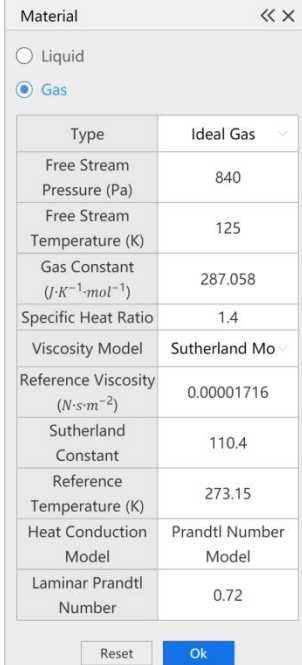
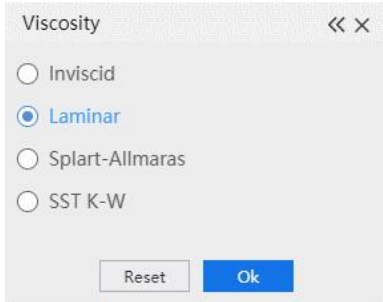
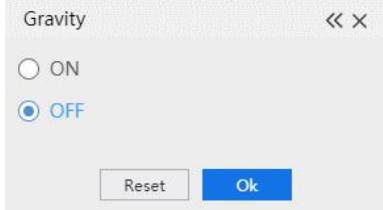
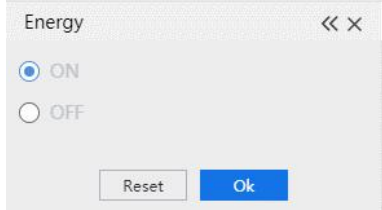


Figure1 *Create a project*

2 Physics

Table1 Option and configuration of Physics

option	configuration	option	configuration
Time		Material	
Viscosity			
Gravity		Energy	

3 Boundary Type

Table2 Boundary type setting

boundaries name	far-field	wall	symmetry
Boundaries Type	Boundaries Type << x ☐ Wall ☐ Slip Wall ☐ Stagnation Inlet ☐ Velocity Inlet ☐ Pressure Inlet ☐ Pressure Outlet ☐ Symmetry ☒ Far-field Mach Number 8 AoA 0 Reset Ok	Boundaries Type << x ☒ Wall ☐ Slip Wall ☐ Stagnation Inlet ☐ Velocity Inlet ☐ Pressure Inlet ☐ Pressure Outlet ☐ Symmetry ☐ Far-field Isothermal T (K) 294 Reset Ok	Boundaries Type << x ☐ Wall ☐ Slip Wall ☐ Stagnation Inlet ☐ Velocity Inlet ☐ Pressure Inlet ☐ Pressure Outlet ☒ Symmetry ☐ Far-field Reset Ok

4 Numerical Method

Table 3-30 Option and configuration of Numerical Method

option	configuration	option	configuration
Spatial Gradient	Spatial Gradient << x ☒ GREEN_GAUSS ☐ WEIGHTED_LEAST_SQUARES Reset Ok	Preprocessor	Preprocessor << x ☐ DIAGNOAL_SCALING ☐ STATIC_SPAI ☐ DYNAMIC_SPAI ☒ Jacobi Reset Ok
Convection Term	Convection Term << x ☐ JST ☒ ROE ☐ LAX-FRIEDRICH ☐ AUSM Entropy fix coefficient 0.5 Reset Ok	Quantum Linear Solver	Quantum Linear Solver << x ☐ Subspace HHL ☒ Variational Algorithm Subspace Dimension 2 Number of Subspace Restarts 4 Learning Rate 0.5 Reset Ok

5 Control Parameters

Table4 Configuration of Control Parameters

parameters	CFL Number	Relaxation Factor	Convergence Error	Maximum Iteration Steps
configuration	10	0.5	0.001	2000

6 Report

Click the *New Report* button to create a report named Avg_Temp_Wall, select the *Type* as *Temperature*, and select *Wall* as the *Object*. This report can be used to quantitatively analyze the relationship between the average temperature of the wall and the iteration steps.

Click the *New Report* button to create a report named surface, select the type as *Surface data*, and select symmetry as the *Object*. This report is used to analyze the trend of temperature changes on the symmetrical 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.

3 Result

This example will create diagram, temperature scalar and temperature

contour in the results to analyze the variation of temperature on and near the surface of the projectile. Hover over the *Result* option, click on the *Result* button  , select the *Type* as *diagram*, *Data Type* as *Temperature*, and select the report created in the *Report* named *Avg_Temp_Wall* as the *Data Source* to create a diagram representing the average temperature of the boundary wall with iteration steps.

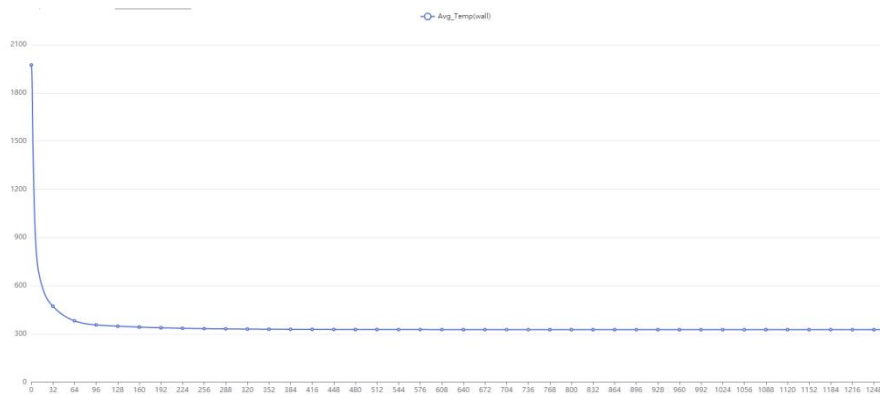


Figure2 diagram of average surface temperature of wall

Hover the mouse over the *Result* option, click on the *Result* button  , select the *Data* as *Filed*, 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 *Temperature* and the *Data Range* as 200-1700.

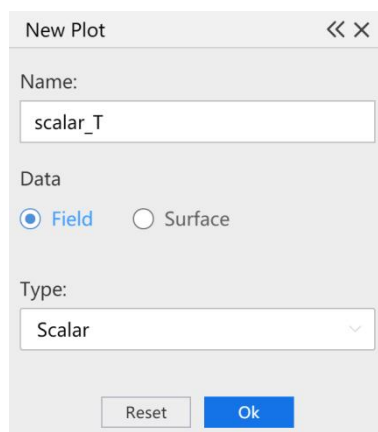


Figure3 New Plot

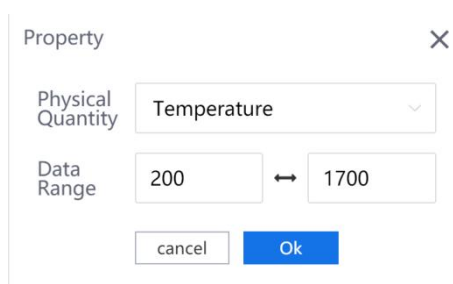


Figure4 Property

Hover the mouse over the *Result* option, click on the *Result* button , select the *Data* as *Surface*, select the *Type* as *Contour*, select the *Physical Variable* as *Temperature*, select the *Range* as 200,1700, select the *Number of Samples* as 50, 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 *Temperature* and the *Data Range* as 200-1700.

New Plot
<< X

Name:

Temperature

Data

☐ Field
☒ Surface

Type:

Contour

Physical Variable:

Temperature

Range	200,1700
Number of Samples	50

Reset
Ok

Figure5 *New Plot*

Property
X

Physical Quantity

Temperature

Data Range

200

↔

1700

Figure6 *Property*

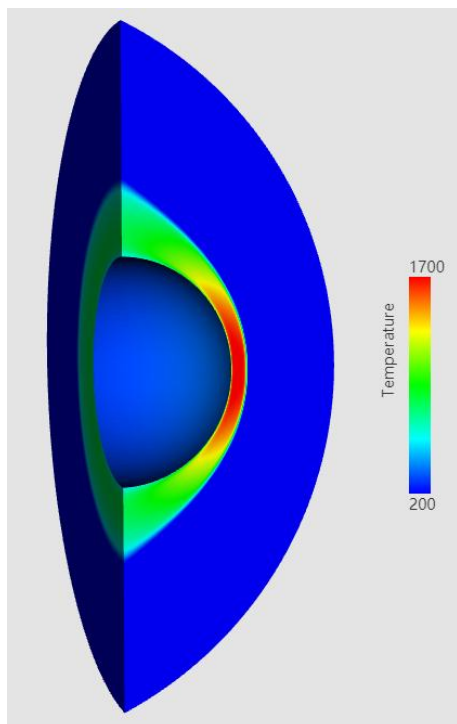


Figure7 *Temperature Scalar*

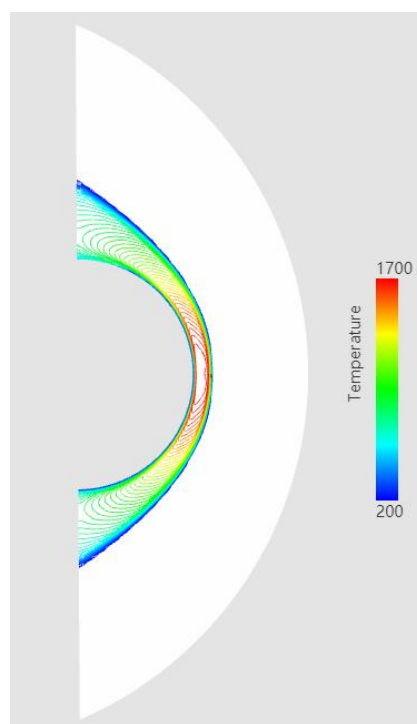


Figure8 *Temperature Contour*