Seminario nell’ambito del corso di
Aerodinamica degli Aeromobili

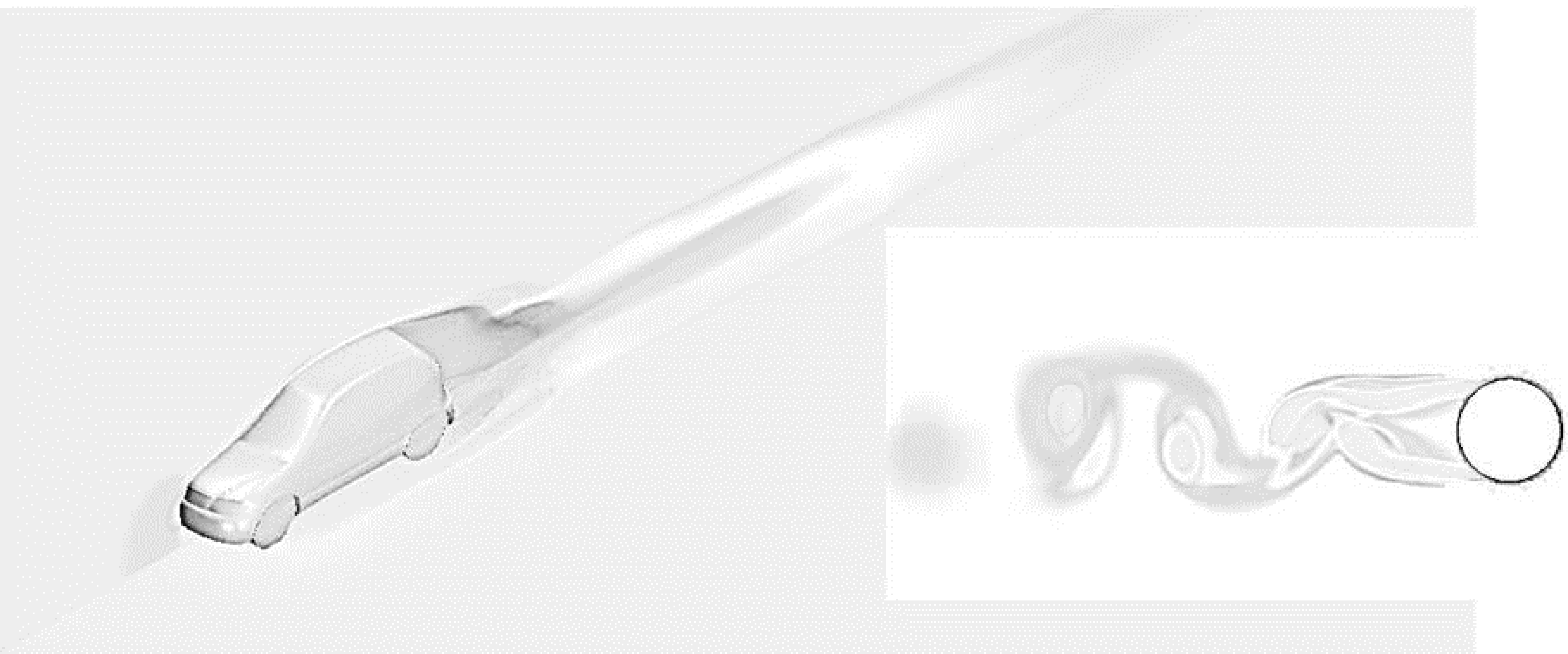
Analisi del profilo NACA 63012A
con il solutore STAR-CCM+

Ing. Giuseppe Calise
Ph.D. Student
e-mail: giuseppe.calise@unina.it

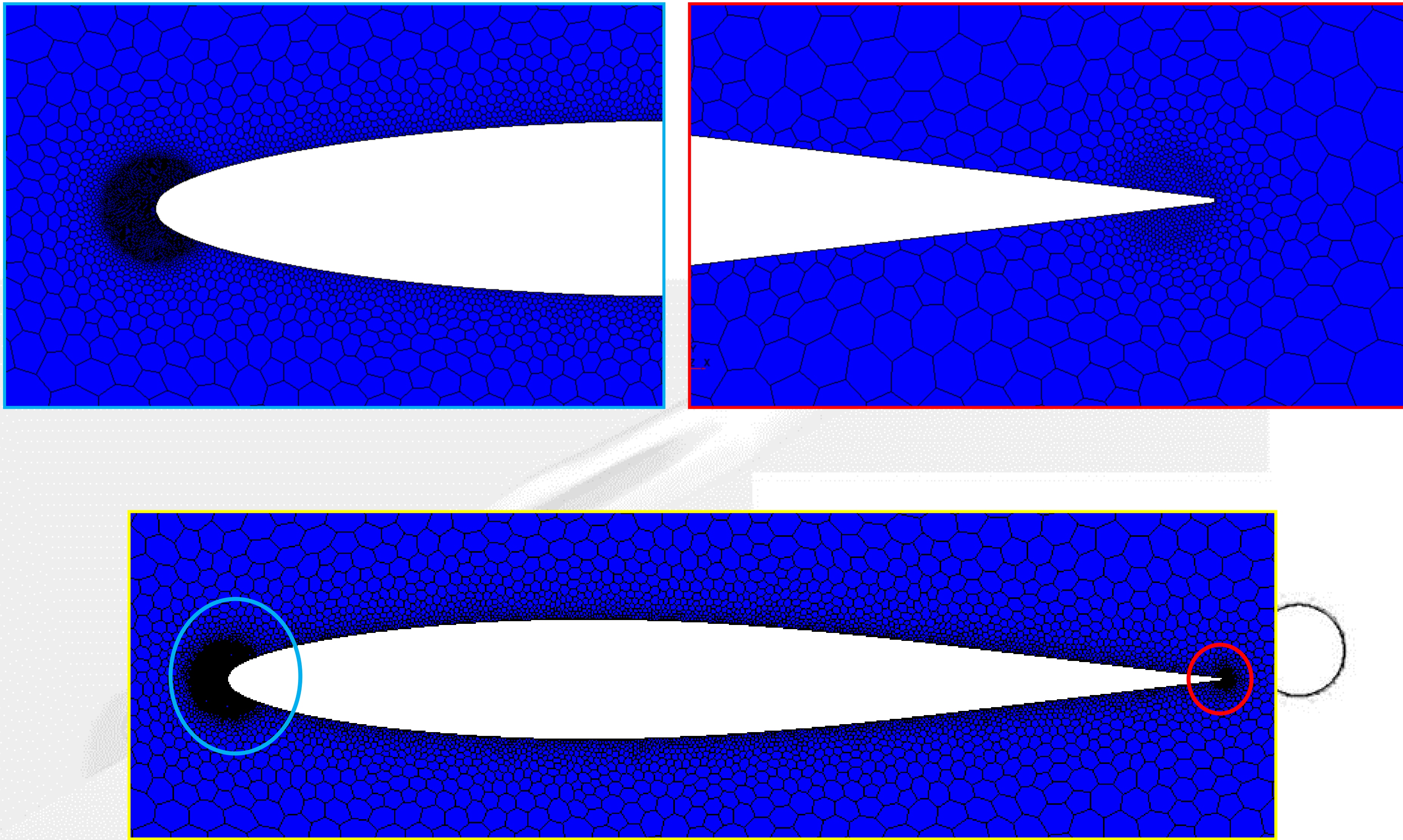
SOMMARIO

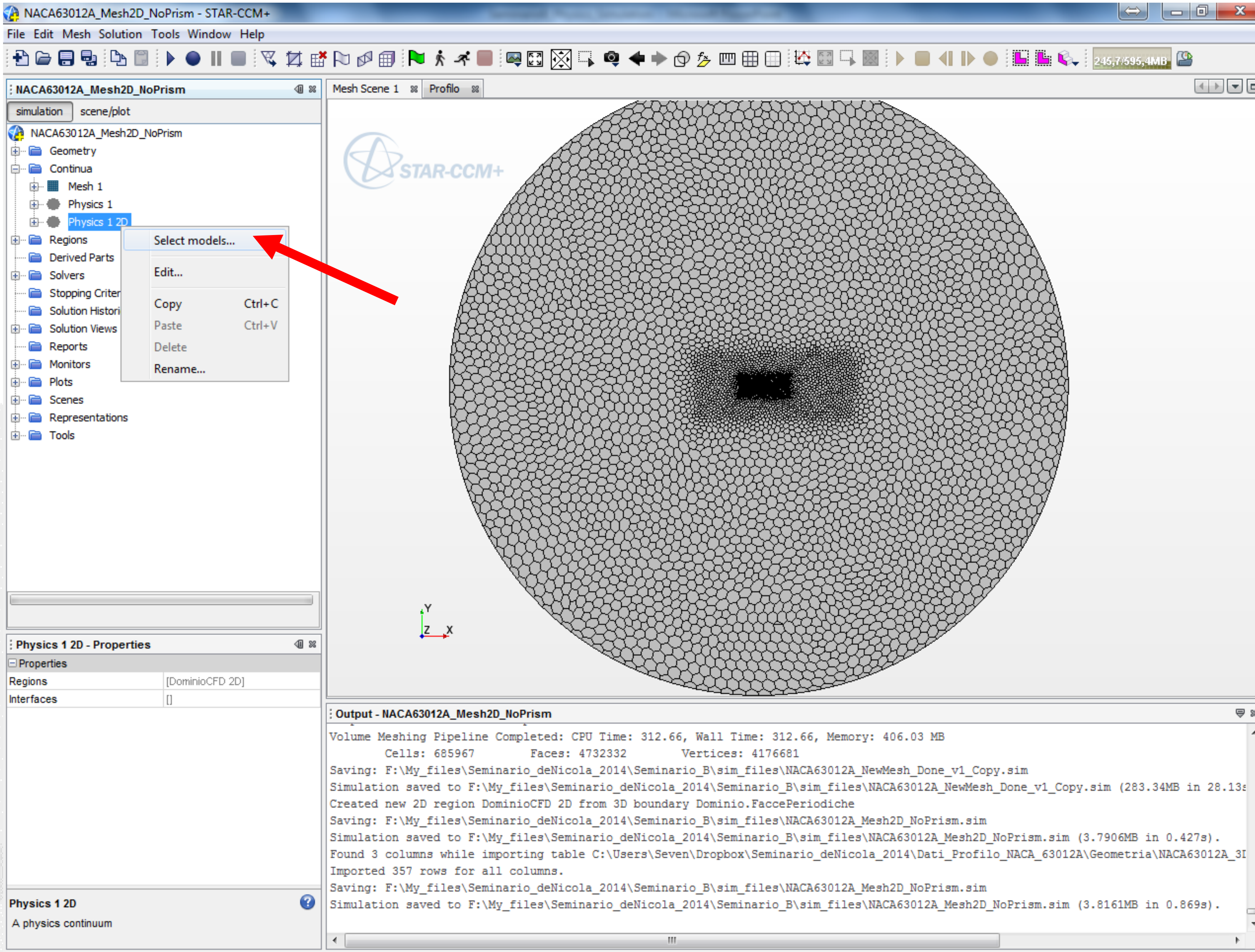
- Parte 1
  - Introduzione al software CFD
  - Importazione geometria CAD
  - Generazione griglia di calcolo
- **Parte 2**
  - **Impostazione della fisica del problema**
  - **Generazione di un report e di un grafico**
  - **Impostazione dei criteri di convergenza e avvio del calcolo**
    - **Cosa fare se il calcolo non converge?**
- Parte 3
  - Confronto dei risultati ottenuti con altri metodi
  - Variazione della geometria e/o delle condizioni fisiche
  - Avvio di un nuovo calcolo e confronto dei risultati

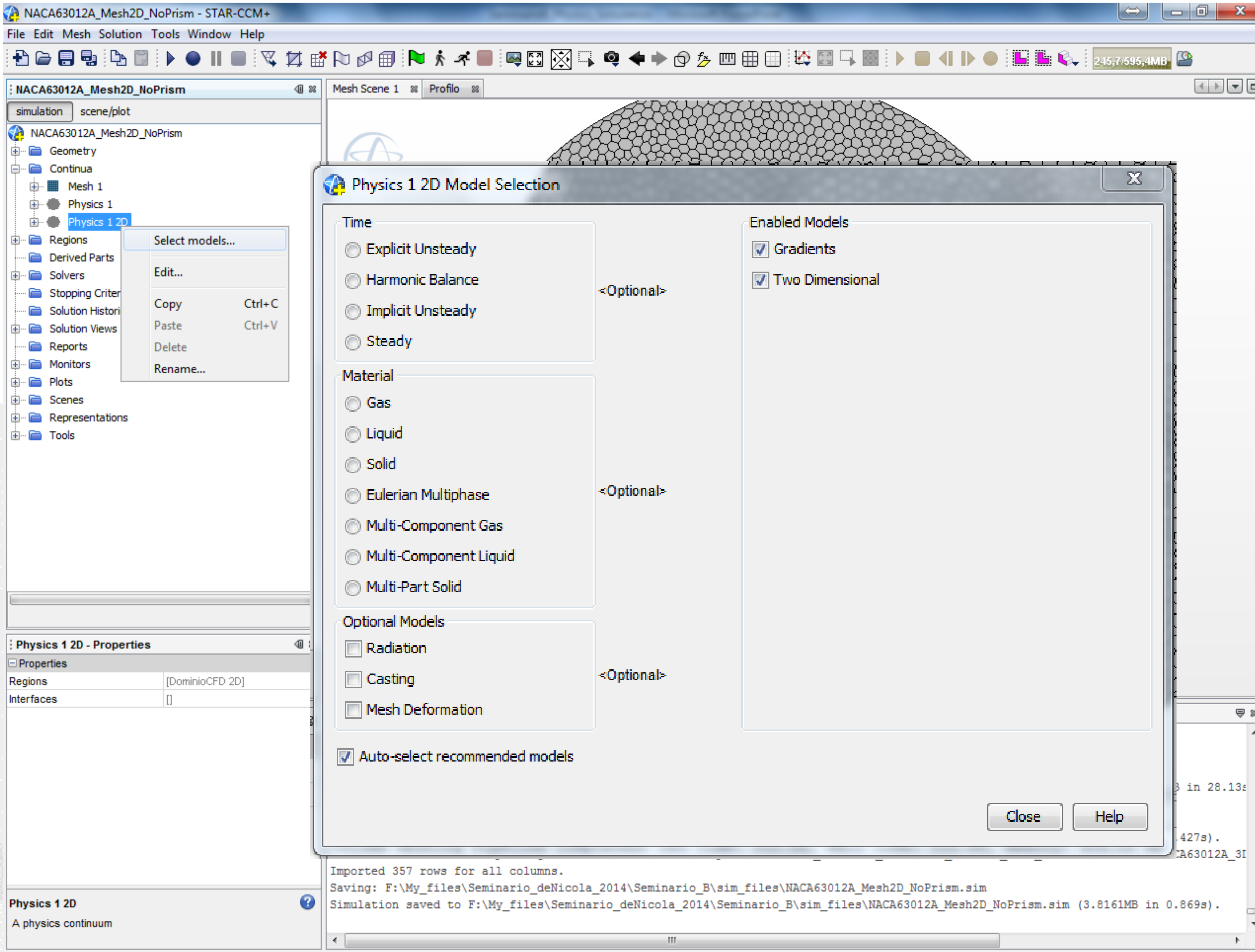
Impostazione della fisica del problema



Impostazione della fisica: soluzione non viscosa, alto subsonico (M_∞=0.5)




NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
Derived Parts
Solvers
Stopping Criter
Solution Histori
Solution Views
Reports
Monitors
Plots
Scenes
Representations
Tools
Select models...
Edit...
Copy Ctrl+C
Paste Ctrl+V
Delete
Rename...
Mesh Scene 1
Profilo
STAR-CCM+
Physics 1 2D - Properties
Properties
Regions [DominioCFD 2D]
Interfaces []
Physics 1 2D
A physics continuum
Output - NACA63012A_Mesh2D_NoPrism
Volume Meshing Pipeline Completed: CPU Time: 312.66, Wall Time: 312.66, Memory: 406.03 MB
Cells: 685967 Faces: 4732332 Vertices: 4176681
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_NewMesh_Done_v1_Copy.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_NewMesh_Done_v1_Copy.sim (283.34MB in 28.13s
Created new 2D region DominioCFD 2D from 3D boundary Dominio.FaccePeriodiche
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.7906MB in 0.427s).
Found 3 columns while importing table C:\Users\Seven\Dropbox\Seminario_deNicola_2014\Dati_Profilo_NACA_63012A\Geometria\NACA63012A_3I
Imported 357 rows for all columns.
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).


NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
Derived Parts
Solvers
Stopping Criter
Solution Histori
Solution Views
Reports
Monitors
Plots
Scenes
Representations
Tools
Select models...
Edit...
Copy Ctrl+C
Paste Ctrl+V
Delete
Rename...
Mesh Scene 1
Profilo
Physics 1 2D Model Selection
Time
Explicit Unsteady
Harmonic Balance
Implicit Unsteady
Steady
<Optional>
Material
Gas
Liquid
Solid
Eulerian Multiphase
Multi-Component Gas
Multi-Component Liquid
Multi-Part Solid
Optional Models
Radiation
Casting
Mesh Deformation
Enabled Models
Gradients
Two Dimensional
Auto-select recommended models
Close
Help
Physics 1 2D - Properties
Properties
Regions [DominioCFD 2D]
Interfaces []
Physics 1 2D
A physics continuum
Imported 357 rows for all columns.
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).

Physics 1 2D Model Selection

(STAR-CCM+ User Guide)

Selecting Physics Models (pp. 2421)

A physics model in STAR-CCM+ defines how a physical phenomenon in a continuum is represented. Essentially, physics models define the primary variables of the simulation (such as pressure, temperature, and velocity) and what mathematical formulation is used to generate the solution. Models in STAR-CCM+ have varying levels of complexity and functionality, but their major purpose is to work with solvers to obtain a solution and to help present the information to you. The tasks of a typical model include making relevant field functions available and placing initial conditions and reference values in its continuum. In any continuum, models are selected and changed using the **Physics Model Selection dialog**, which assists you in recommending models and in selecting the appropriate combination of models.

What Is the Steady Model? (pp. 2498)

The Steady model is used for all steady-state calculations. When this model is activated, the concept of a physical time-step is meaningless.

General Navier-Stokes (pp. 2498)

All of fluid dynamics is based on three physical principles:

- **Mass is conserved.**
- **Newton's second law,**
- **Energy is conserved.**

The form of governing equations that is generally used in Computational Fluid Dynamics (CFD) is known as the Navier-Stokes equations. The Navier-Stokes equations include the effect of viscosity on the fluid flow.
In STAR-CCM+, the **finite volume method** is used to transform the continuous governing equations into a form that can be solved numerically.
Two approaches are taken to solve the governing equations:

- The segregated approach, where the flow equations are solved one after the other and linked using a correction equation.
- The coupled approach, where the coupled system of equations is solved simultaneously.

The Coupled Flow Model (pp. 2774)
The Coupled Flow model solves the conservation equations for mass, momentum, and energy simultaneously using a pseudo-time-marching approach. One advantage of this formulation is its robustness for solving flows with dominant source terms, such as rotation.
Running a Steady State Analysis (pp. 2784)
For steady simulations, the coupled solver in STAR-CCM+ employs a time marching scheme to drive the unsteady form of the governing equations to a steady state. In this case, a pseudo-transient term replaces the physical time-derivative. The solution advances in pseudo-time to drive this term to zero. The solution in each cell is advanced independently with an optimal pseudo-time step computed locally according to stability constraints. In this way, convergence to steady state is achieved in the most efficient manner.

Physics 1 2D Model Selection (cont.)

(STAR-CCM+ User Guide)

How Do I Choose Between Coupled and Segregated? (pp. 2773)

To guide the choice between the Segregated Flow model and the Coupled Flow model, consider their relative strengths and weaknesses:

- The segregated algorithm uses less memory than the coupled.
- The coupled algorithm yields more robust and accurate solutions incompressible flow, particularly in the presence of shocks.
- The coupled algorithm is more robust for high-Rayleigh number natural convection.
- The number of iterations that the coupled algorithm requires to solve a given flow problem is independent of mesh size. However, the number of iterations that the segregated algorithm requires increases with mesh size.
- In some situations the coupled algorithm can be combined with the implicit solver to permit large CFL numbers. This scenario would be analogous to an under-relaxation factor of 1 for all variables in a segregated algorithm. In contrast, the segregated algorithm needs significant under-relaxation for both velocity and pressure and, in compressible flows, energy.

With these strengths and weaknesses in mind, it is suggested that you proceed as follows to select the algorithm:

- Choose the Coupled Flow and Coupled Energy models for compressible flows, natural convection problems, and flows with large body forces or energy sources.
- If computational resources are not an issue, choose the Coupled Flow model for incompressible and/or isothermal flows.
- Choose the Segregated Flow model for incompressible or mildly compressible flows.

Equation of state (pp. 2748)

The Equation of State model is used to compute the density and the density derivatives with respect to temperature and pressure. This section describes the various models that are available in STAR-CCM+:

- Constant Density
- IAPWS-IF97
- Polynomial Density
- **Ideal Gas:** the Ideal Gas model, available for gases only, uses the ideal gas law to express density as a function of temperature and pressure.
- Real Gas
- Thermal Non-Equilibrium
- User Defined Equation of State (EOS)

Selecting an Equation of State Model

Selection of an Equation of State model depends on the selection of a single-phase (single- or multi-component) or multiphase material model.

Physics 1 2D Model Selection (cont.)

(STAR-CCM+ User Guide)

Modeling the Viscous Regime (pp. 2862)

STAR-CCM+ lets you specify the viscous regime to use in your simulation. The following types of flows can be modeled:

- **Inviscid Flow:**

  Inviscid flows are an idealization resulting from neglecting the viscous effects in simulating the equations of motion. The solution of the resulting Euler equations (as opposed to the Navier-Stokes equations) generally results in significant savings of computer resources. Boundary layers and other viscous effects are not resolved. This approximation is only valid for certain physical situations, such as high-Reynolds number compressible aerodynamics.

- **Viscous Flow:**

  Viscous flows can be classified as either laminar or turbulent. Laminar and turbulent flows occur in nature. Both types are described using the Navier-Stokes equations which include the effects of viscosity, thermal conductivity, and mass diffusion.

- **Laminar Flow:**

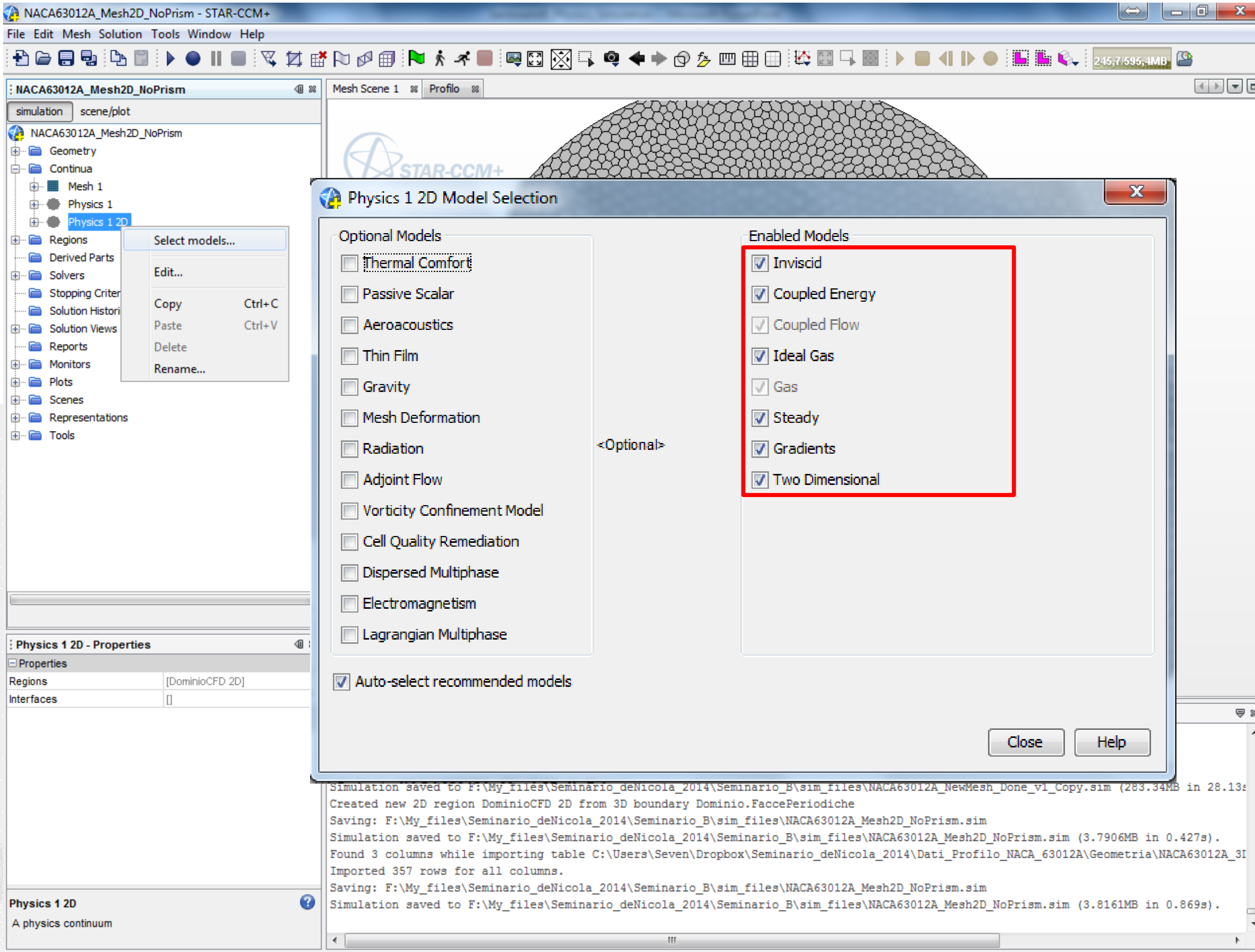
  The term laminar refers to a well-ordered flow, free of macroscopic, non-repeating fluctuations. Laminar flows occur in nature when the Reynolds number (the ratio of viscous to inertial forces) is low enough that transition to turbulence does not occur. In computational simulations, numerical instabilities can arise from simulating laminar flows at Reynolds numbers that are too large. If you are seeking a steady solution, these instabilities can impede convergence. Therefore, laminar flow simulation is appropriate if you already know that the Reynolds number of the problem is sufficiently low.

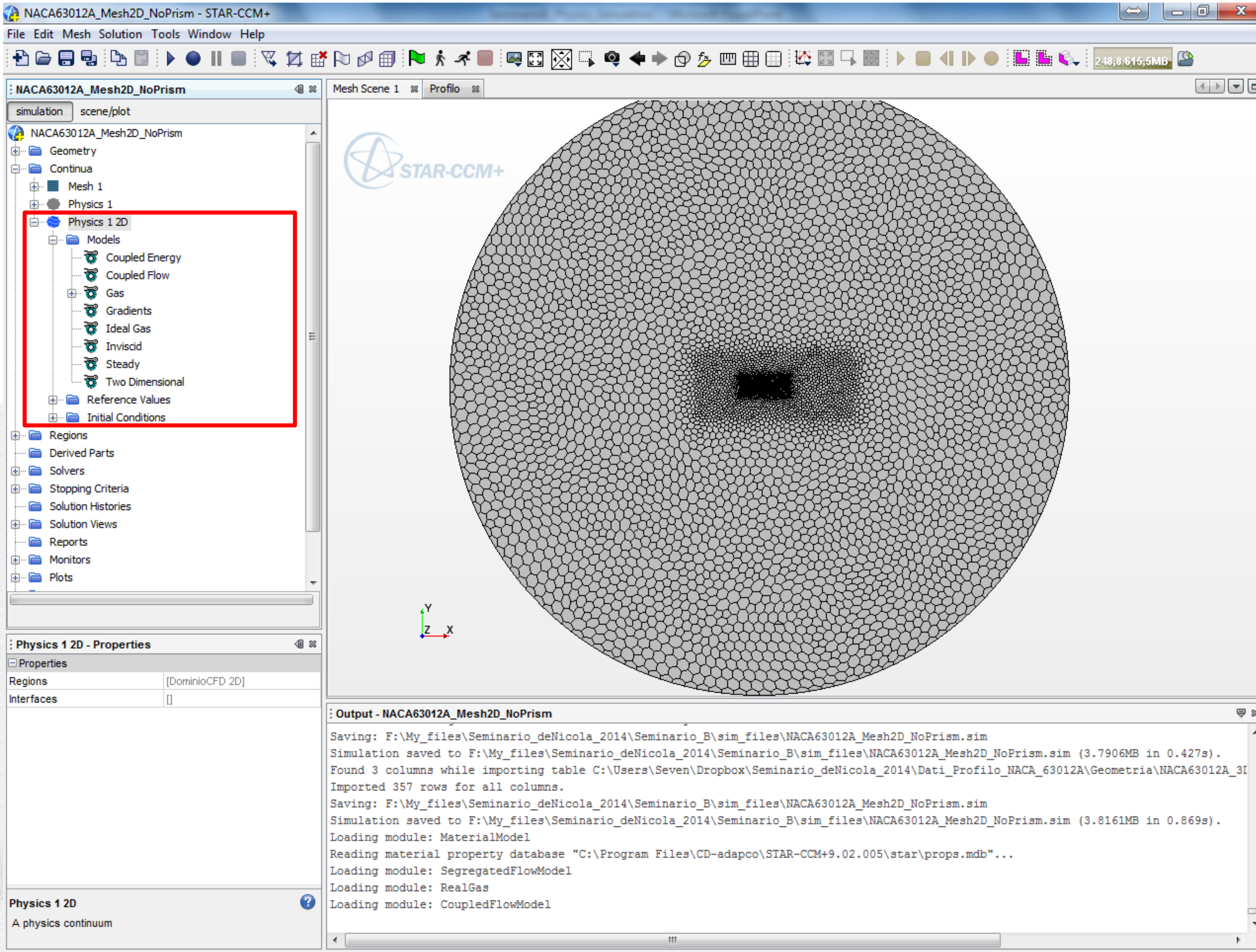
- **Transitional Flow:**

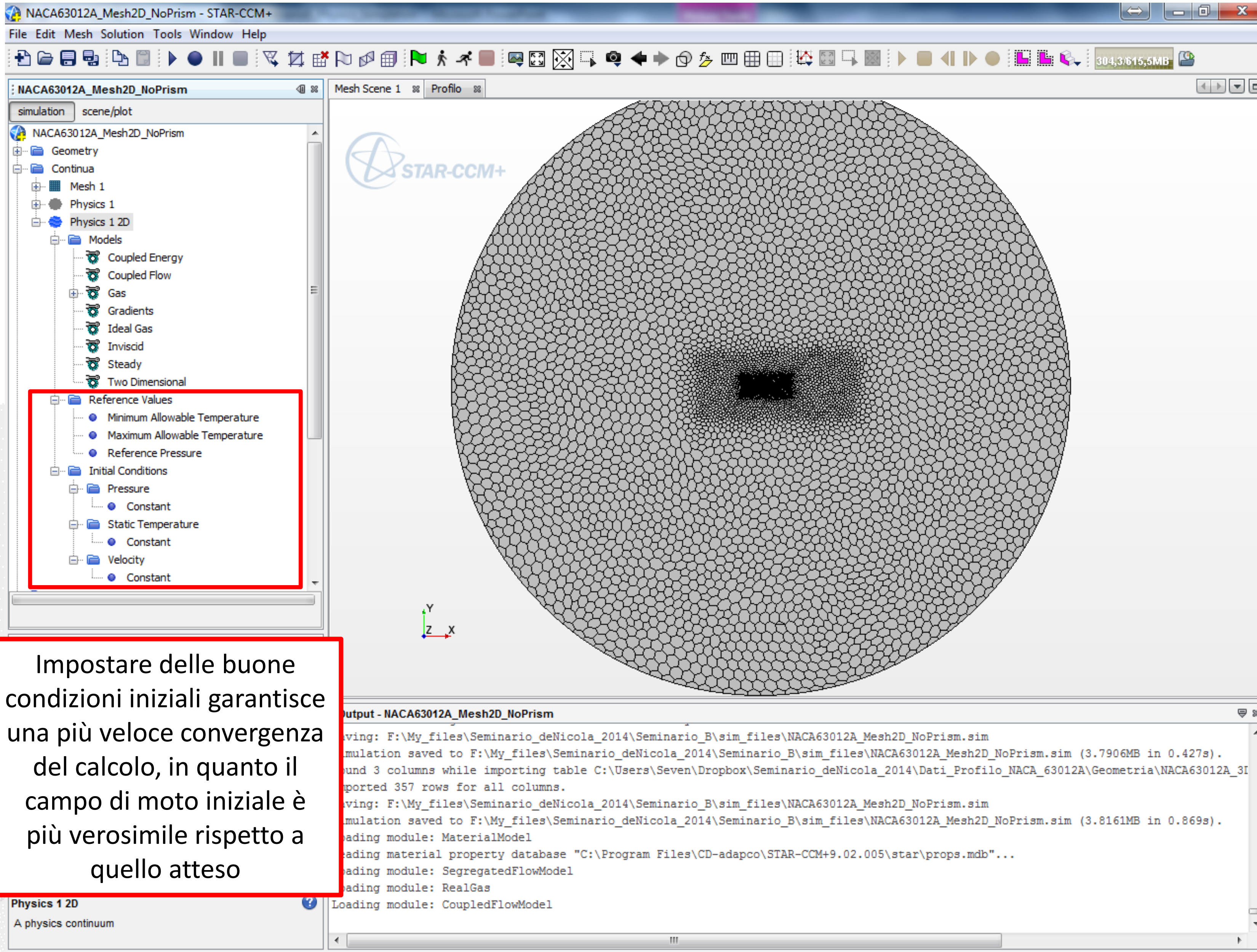
  Transition is a term that refers to the breakdown of laminar flow, through amplification of infinitesimal disturbances, to turbulence. A transitional flow can be defined as one that encompasses this process. The occurrence of physical instabilities in laminar simulations cannot be relied on as an accurate indication of transition. Furthermore, when using turbulence models in low-Reynolds number simulations, the onset of turbulence in viscous layers cannot be predicted with any reliability by the turbulence model itself. Strictly speaking, STAR-CCM+ has no means of predicting transition. However, a Transition model is available that allows you to mimic the effect of transition by suppressing the turbulence in a certain pre-defined region.

- **Turbulent Flow:**

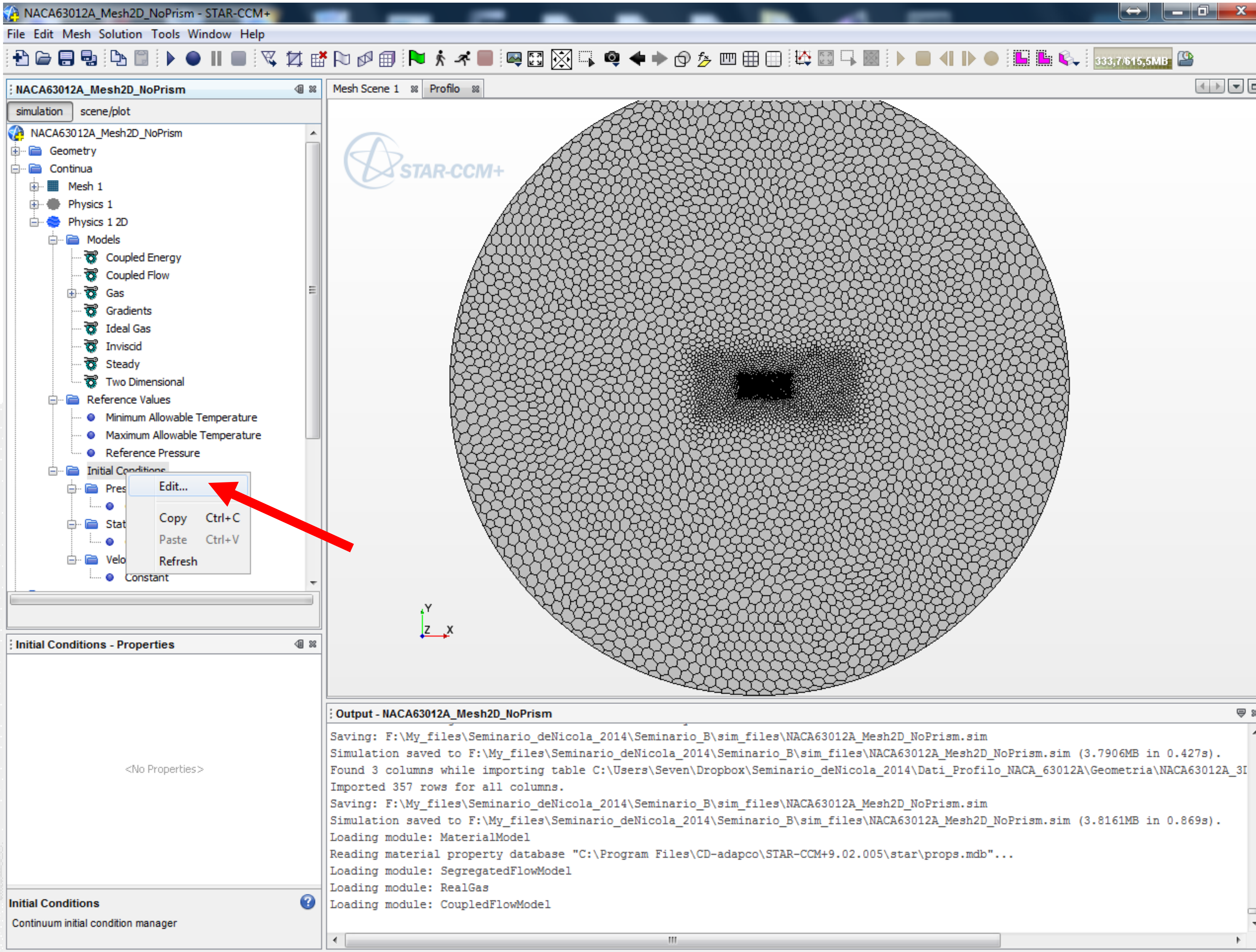
  A flow that is in a state of continuous instability, exhibiting irregular, small-scale, high-frequency fluctuations in both space and time is termed turbulent. It is strictly possible to simulate turbulent flow directly by resolving all the scales of the flow (termed direct numerical simulation). However, the computer resources that are required are too large for practical flow simulations. Therefore, a suitable turbulence modeling approach must be selected.

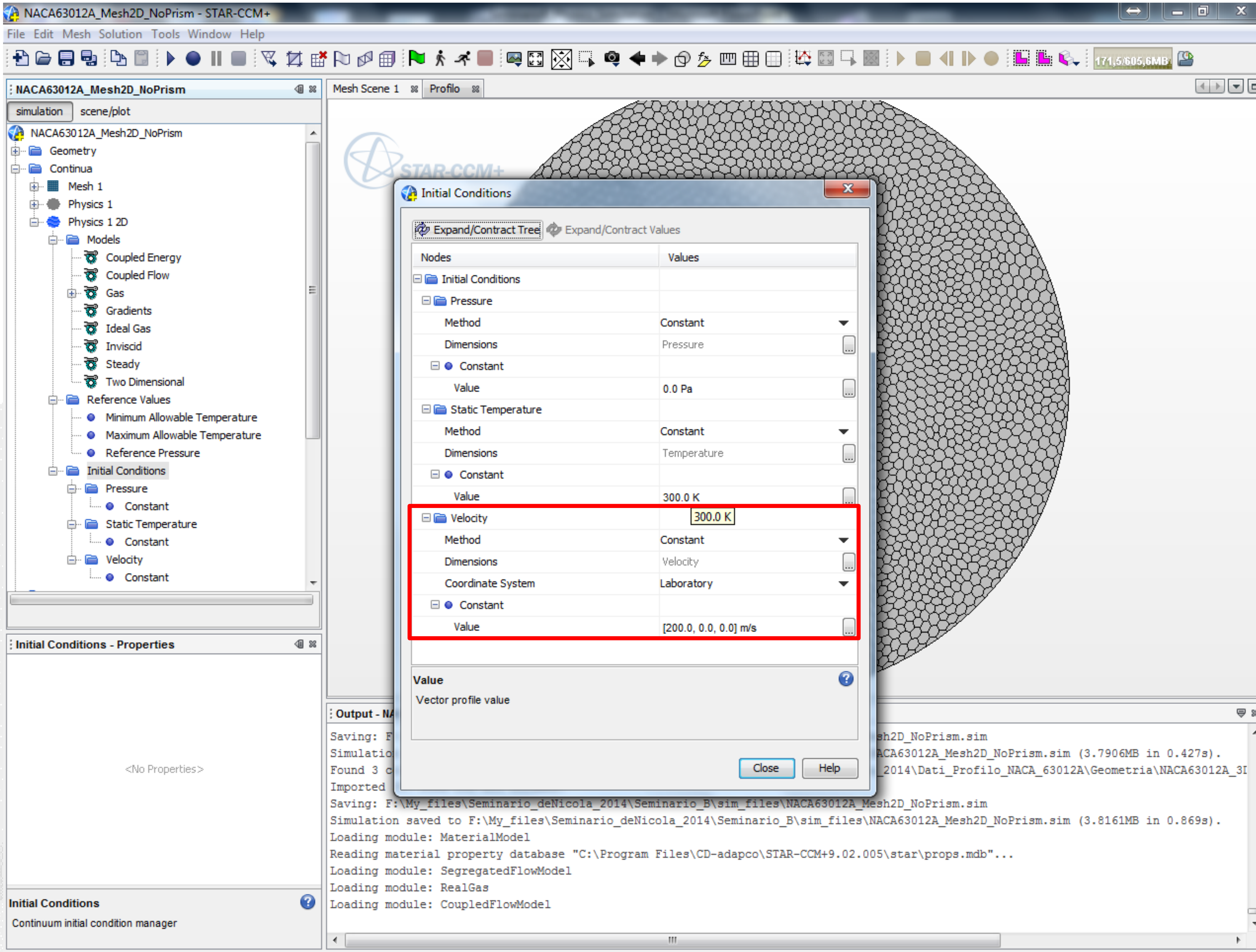

NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
Derived Parts
Solvers
Reports
Monitors
Plots
Scenes
Representations
Tools
Select models...
Edit...
Copy Ctrl+C
Paste Ctrl+V
Delete
Rename...
Mesh Scene 1
Profilo
Physics 1 2D Model Selection
Optional Models
Thermal Comfort
Passive Scalar
Aeroacoustics
Thin Film
Gravity
Mesh Deformation
Radiation
Adjoint Flow
Vorticity Confinement Model
Cell Quality Remediation
Dispersed Multiphase
Electromagnetism
Lagrangian Multiphase
<Optional>
Enabled Models
Inviscid
Coupled Energy
Coupled Flow
Ideal Gas
Gas
Steady
Gradients
Two Dimensional
Auto-select recommended models
Close
Help
Physics 1 2D - Properties
Properties
Regions [DominioCFD 2D]
Interfaces []
Physics 1 2D
A physics continuum
Created new 2D region DominioCFD 2D from 3D boundary Dominio.FaccePeriodiche
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.7906MB in 0.427s).
Imported 357 rows for all columns.
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).

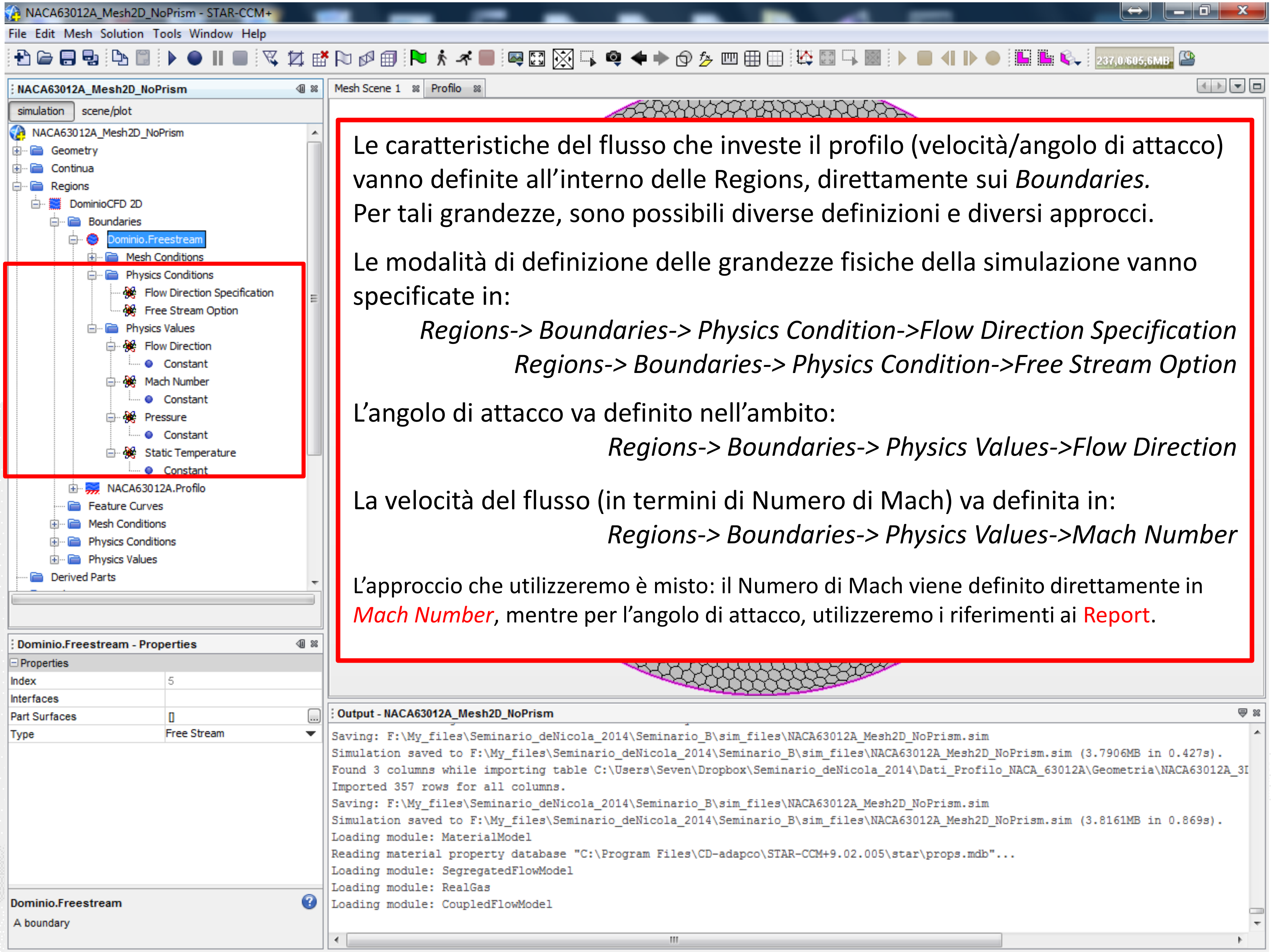

NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Models
Coupled Energy
Coupled Flow
Gas
Gradients
Ideal Gas
Inviscid
Steady
Two Dimensional
Reference Values
Initial Conditions
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Physics 1 2D - Properties
Properties
Regions
[DominioCFD 2D]
Interfaces
[]
Physics 1 2D
A physics continuum
Mesh Scene 1
Profilo
STAR-CCM+
Y
Z
X
Output - NACA63012A_Mesh2D_NoPrism
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.7906MB in 0.427s).
Found 3 columns while importing table C:\Users\Seven\Dropbox\Seminario_deNicola_2014\Dati_Profilo_NACA_63012A\Geometria\NACA63012A_3I
Imported 357 rows for all columns.
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).
Loading module: MaterialModel
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Loading module: SegregatedFlowModel
Loading module: RealGas
Loading module: CoupledFlowModel



Impostare delle buone condizioni iniziali garantisce una più veloce convergenza del calcolo, in quanto il campo di moto iniziale è più verosimile rispetto a quello atteso


NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Models
Coupled Energy
Coupled Flow
Gas
Gradients
Ideal Gas
Inviscid
Steady
Two Dimensional
Reference Values
Minimum Allowable Temperature
Maximum Allowable Temperature
Reference Pressure
Initial Conditions
Edit...
Copy Ctrl+C
Paste Ctrl+V
Refresh
Constant
Mesh Scene 1
Profilo
STAR-CCM+
Initial Conditions - Properties
<No Properties>
Initial Conditions
Continuum initial condition manager
Output - NACA63012A_Mesh2D_NoPrism
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.7906MB in 0.427s).
Found 3 columns while importing table C:\Users\Seven\Dropbox\Seminario_deNicola_2014\Dati_Profilo_NACA_63012A\Geometria\NACA63012A_3I
Imported 357 rows for all columns.
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).
Loading module: MaterialModel
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Loading module: SegregatedFlowModel
Loading module: RealGas
Loading module: CoupledFlowModel


NACA63012A_Mesh2D_NoPrism - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_NoPrism
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Models
Coupled Energy
Coupled Flow
Gas
Gradients
Ideal Gas
Inviscid
Steady
Two Dimensional
Reference Values
Minimum Allowable Temperature
Maximum Allowable Temperature
Reference Pressure
Initial Conditions
Pressure
Constant
Static Temperature
Constant
Velocity
Constant
Initial Conditions - Properties
<No Properties>
Initial Conditions
Continuum initial condition manager
Mesh Scene 1
Profilo
Initial Conditions
Expand/Contract Tree
Expand/Contract Values
Nodes
Values
Initial Conditions
Pressure
Method
Constant
Dimensions
Pressure
Constant
Value
0.0 Pa
Static Temperature
Method
Constant
Dimensions
Temperature
Constant
Value
300.0 K
Velocity
300.0 K
Method
Constant
Dimensions
Velocity
Coordinate System
Laboratory
Constant
Value
[200.0, 0.0, 0.0] m/s
Value
Vector profile value
Close
Help
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_NoPrism.sim (3.8161MB in 0.869s).
Loading module: MaterialModel
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Loading module: SegregatedFlowModel
Loading module: RealGas
Loading module: CoupledFlowModel



Le caratteristiche del flusso che investe il profilo (velocità/angolo di attacco) vanno definite all'interno delle Regions, direttamente sui *Boundaries.*
Per tali grandezze, sono possibili diverse definizioni e diversi approcci.

Le modalità di definizione delle grandezze fisiche della simulazione vanno specificate in:

Regions-> Boundaries-> Physics Condition->Flow Direction Specification

Regions-> Boundaries-> Physics Condition->Free Stream Option

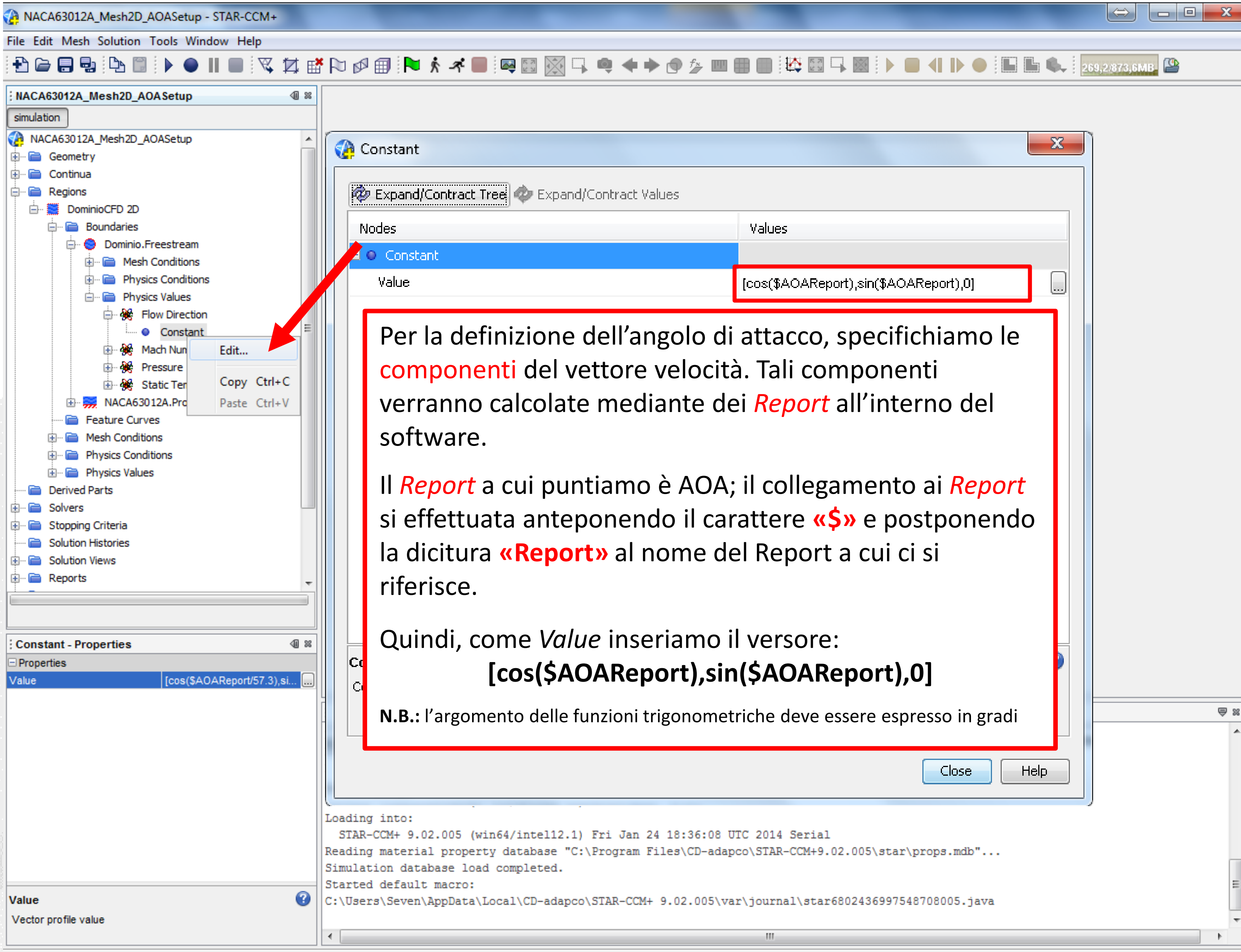
L'angolo di attacco va definito nell'ambito:

Regions-> Boundaries-> Physics Values->Flow Direction

La velocità del flusso (in termini di Numero di Mach) va definita in:

Regions-> Boundaries-> Physics Values->Mach Number

L'approccio che utilizzeremo è misto: il Numero di Mach viene definito direttamente in *Mach Number*, mentre per l'angolo di attacco, utilizzeremo i riferimenti ai Report.



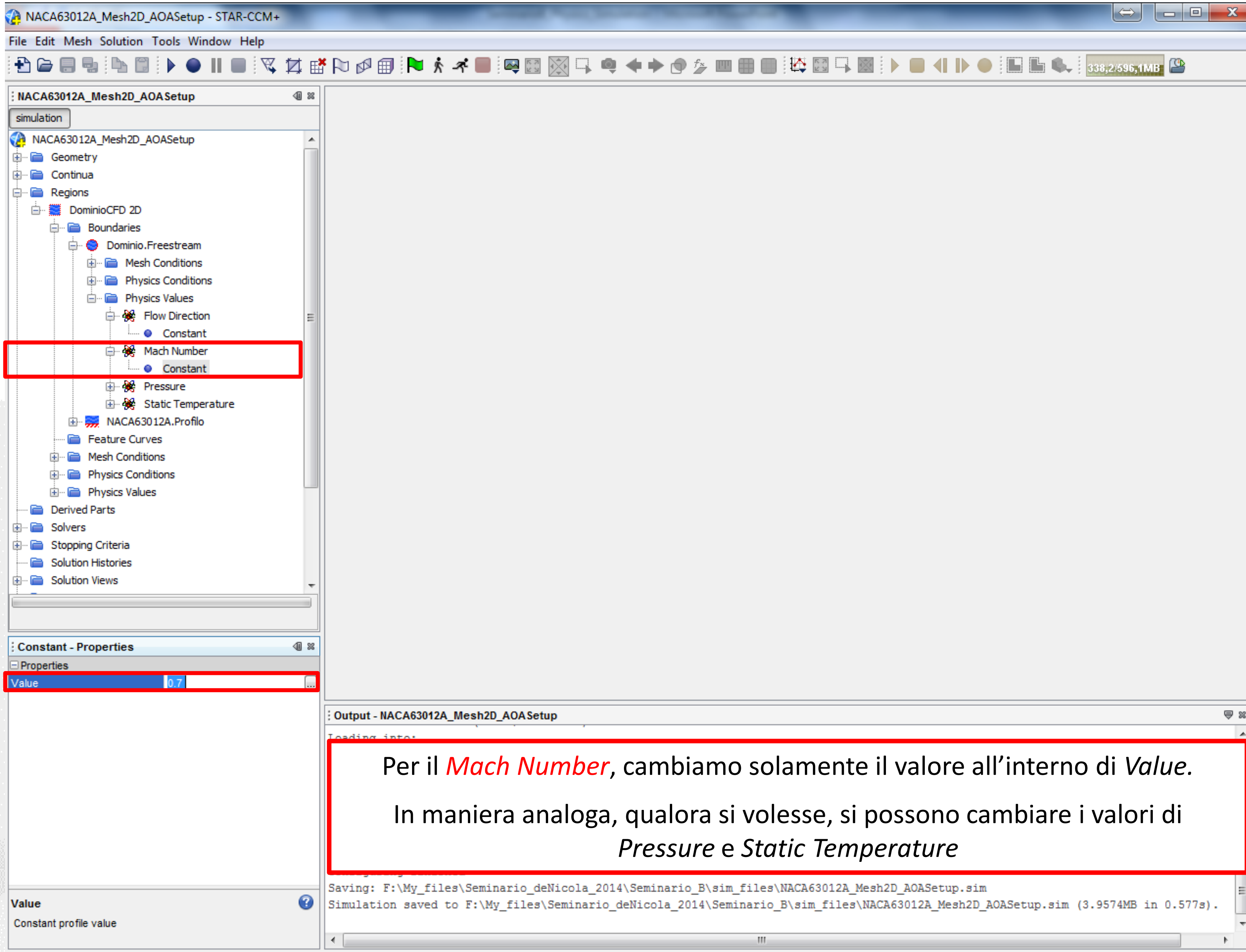
Per la definizione dell'angolo di attacco, specifichiamo le componenti del vettore velocità. Tali componenti verranno calcolate mediante dei *Report* all'interno del software.

Il *Report* a cui puntiamo è AOA; il collegamento ai *Report* si effettuata anteponendo il carattere **«$»** e postponendo la dicitura **«Report»** al nome del Report a cui ci si riferisce.

Quindi, come *Value* inseriamo il versore:

[cos($AOAReport),sin($AOAReport),0]

N.B.: l'argomento delle funzioni trigonometriche deve essere espresso in gradi



Per il *Mach Number*, cambiamo solamente il valore all'interno di *Value.*

In maniera analoga, qualora si volesse, si possono cambiare i valori di *Pressure* e *Static Temperature*

Boundary Types Reference (pp. 2438)

(STAR-CCM+ User Guide)

Free-stream: (pp. 3028)

For a free-stream boundary, there are three ways that you can specify the free-stream pressure, temperature, and Mach number. You select the appropriate option as a property of the Region Name > Boundary > Boundary Name > Physics Conditions > Free Stream Option node.

Free Stream Option Properties:

Mach Number + Pressure + Temperature

For this option, STAR-CCM+ adds Mach Number, Pressure, and Static Temperature nodes to the Physics Values for the boundary. This option is the default free-stream option.

Altitude + Length Scale + Reynolds Number

For this option, STAR-CCM+ adds the Altitude node to the Physics Values for the boundary. The free-stream pressure and temperature are computed internally using the specified altitude. Specify the appropriate Reynolds number and length scale values in the Mach Number node under the Physics Values node for the boundary. The free-stream Mach number is computed internally using the specified altitude, Reynolds number, and length scale.

Altitude + Mach Number

For this option, STAR-CCM+ adds the Altitude and Mach Number nodes to the Physics Values for the boundary. The free-stream pressure and temperature are computed internally using the specified altitude

When you use the Altitude + Length Scale + Reynolds Number or Altitude + Mach Number options, you also specify the appropriate atmosphere type as a property of the Physics Conditions > Atmosphere Type Option node.

Atmosphere Type Option Properties

Standard The US 1976

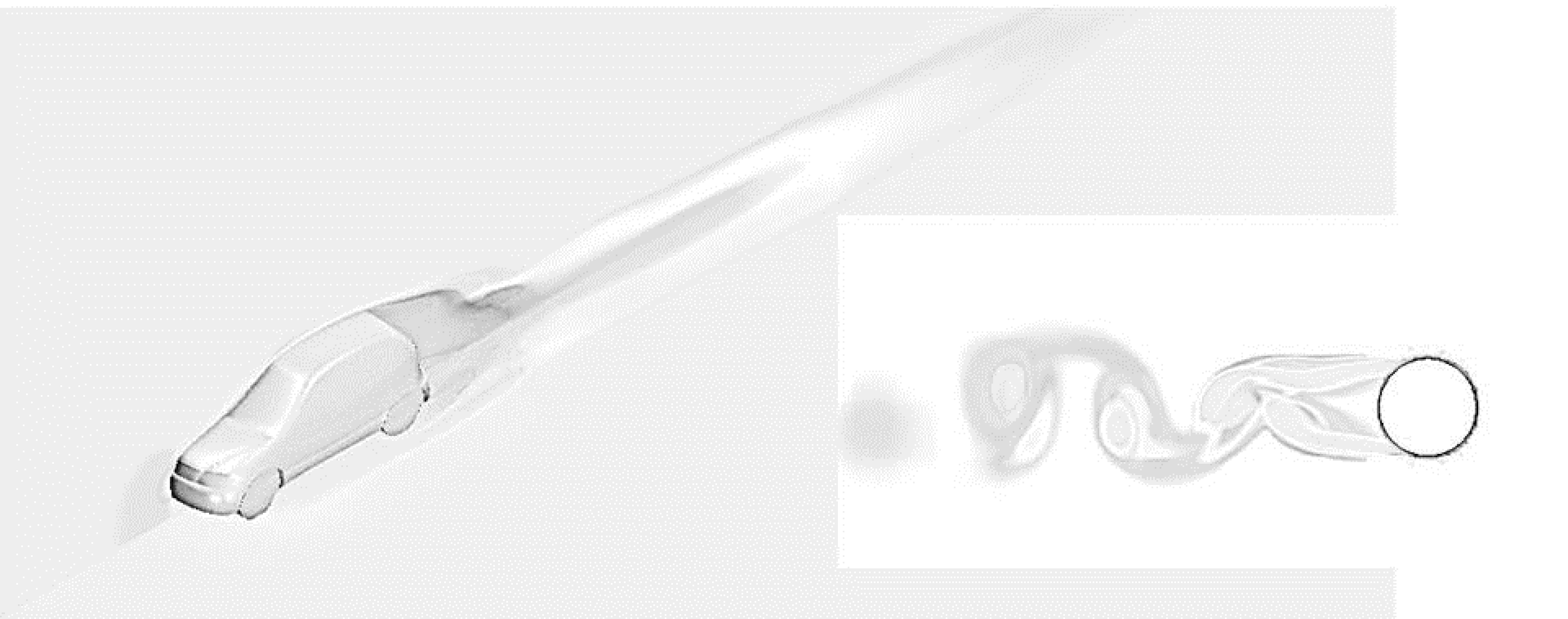
Standard Atmosphere is used to compute the free-stream pressure and temperature from the altitude.

User Table

A user-supplied table is used to compute the free-stream pressure and temperature from the altitude.

Note: When you use the Altitude + Length Scale + Reynolds Number or Altitude + Mach Number options, ensure that you set initial values that are consistent with the free-stream values. You set the initial values in the physics continuum, under the Initial Conditions node, in the Pressure and Static Temperature nodes.

Generazione di un report e di un grafico



Report Types Reference (pp. 7356)

(STAR-CCM+ User Guide)

This section details the available reports along with their formulations if necessary. Many reports share properties, and the following sections have been organized into like categories:

- System
- Statistical
- Specific types

An additional type, the expression report, exists to combine specific and/or statistical reports.

System Reports (pp. 7352)

This section provides a reference on the following types of reports, represented in the simulation tree by nodes, which share properties and are subject to the pop-up menu of the report node:

- Solver iteration CPU time
- Solver iteration elapsed time
- Solver CPU time per time-step
- Solver elapsed time per time-step
- Total solver CPU time
- Total solver elapsed time

No scalar is selected, and no input parts need be defined. When one of these reports is added to the report manager node, it is removed from the pop-up menu of that node.

Statistical Reports (pp. 7358)

This section describes the properties for the following types of reports, represented in the simulation tree by nodes, which share properties and are subject to the pop-up menu of the report node:

- Element count
- Frontal area
- Harmonic mass average
- Harmonic mass flow average
- Line integral
- Mass averaged scalar

Report Types Reference (cont.) (pp. 7356)

(STAR-CCM+ User Guide)

Statistical Reports (cont.)

- Mass flow averaged scalar
- Maximum
- Minimum
- Particle average
- Sum
- Surface area average
- Surface average scalar
- Surface integral
- Surface standard deviation
- Surface uniformity
- Volume average scalar
- Volume integral
- Volume standard deviation
- Volume uniformity

Specific Report Types (pp. 7372)

The following reports have their own unique sets of properties, except for heat transfer and mass flow, which have the same properties:

- Force
- Force (harmonic)
- Force coefficient
- Generalized vibratory force
- Harmonic mass flow
- Heat exchanger (single or dual stream)
- Heat transfer
- Li-Ion Cell
- Mass flow
- Mass flow (Phase)

Report Types Reference (cont.) (pp. 7356)

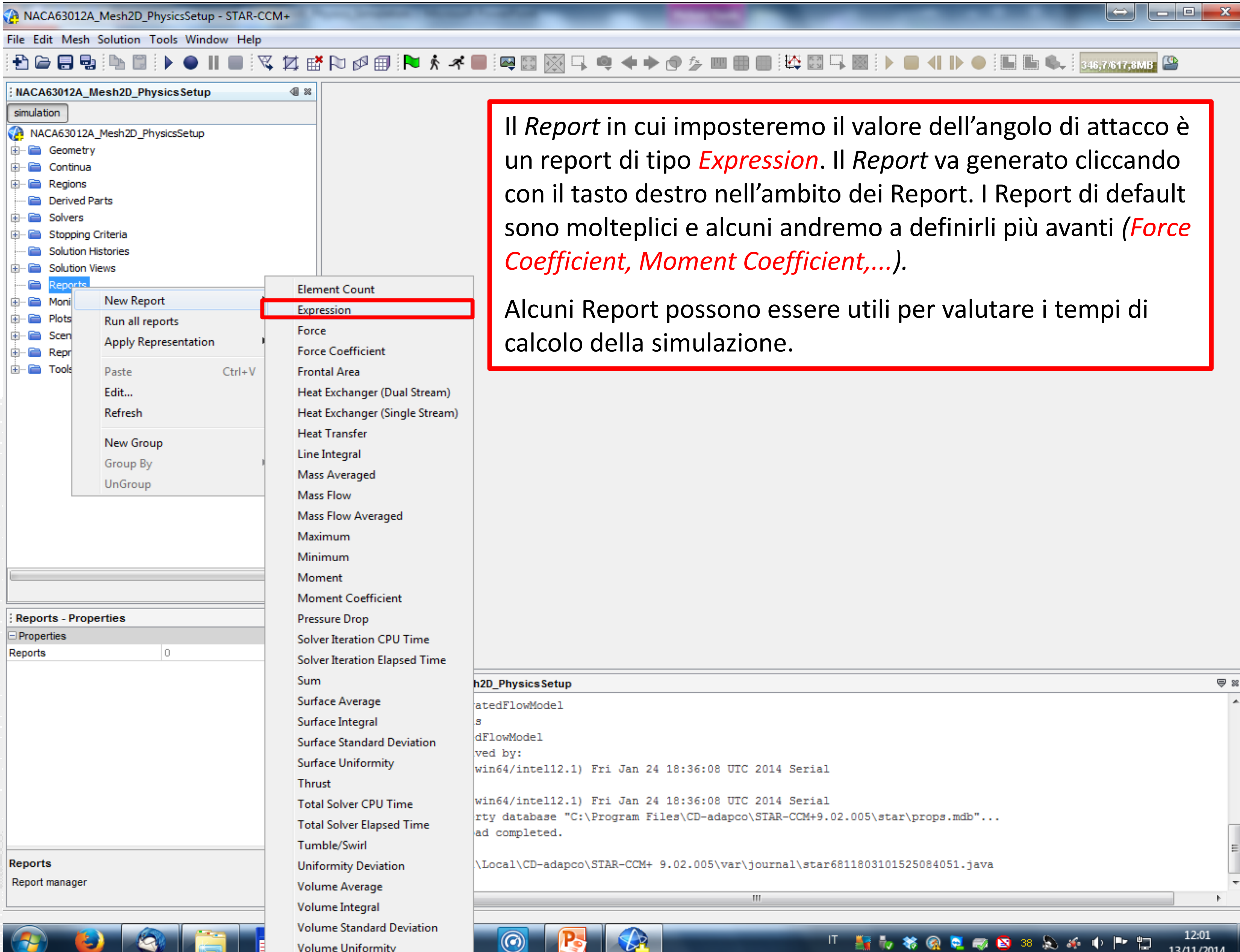
(STAR-CCM+ User Guide)

Specific Report Types (cont.)

- Moment
- Moment coefficient
- Pressure Drop
- Reaction force
- Reaction moment
- Rigid Body Angle
- Rigid Body 1-DOF Angle
- Rigid Body Component Distance
- Rigid Body Force
- Rigid Body Moment
- Rigid Body Rotational Energy
- Rigid Body Spring Elongation
- Rigid Body Catenary Length
- Rigid Body Total Distance
- Sauter mean diameter
- Tumble/Swirl Report
- Thrust
- Uniformity Deviation Report
- Work per cycle
- Coal Volatile Matter Yield
- Char Burn Out

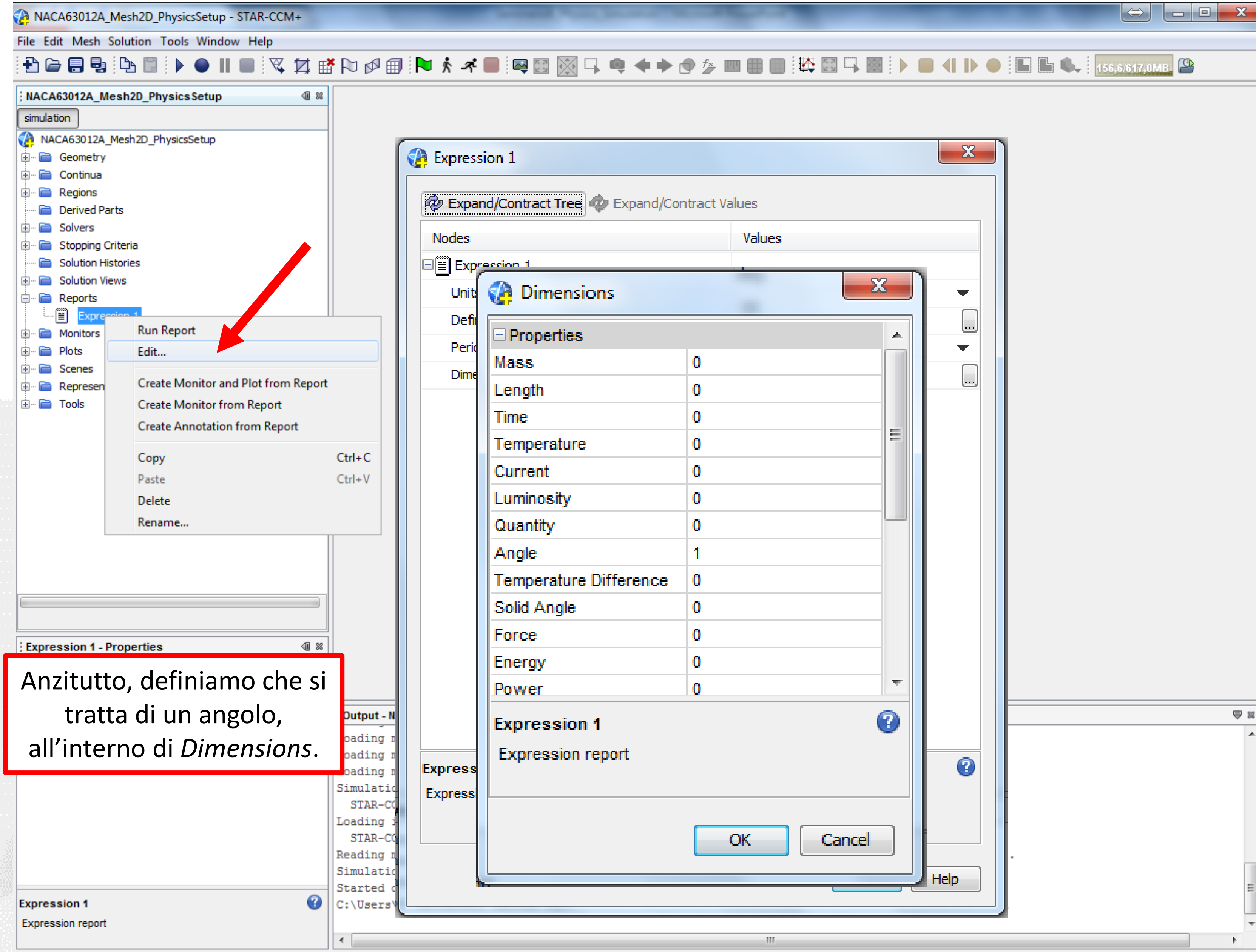
The following 6-DOF reports share properties and are subject to the pop-up menu of the report node:

- Rigid Body Acceleration
- Rigid Body Angular Acceleration
- Rigid Body Angular Momentum
- Rigid Body Angular Velocity
- Rigid Body Translation
- Rigid Body Velocity

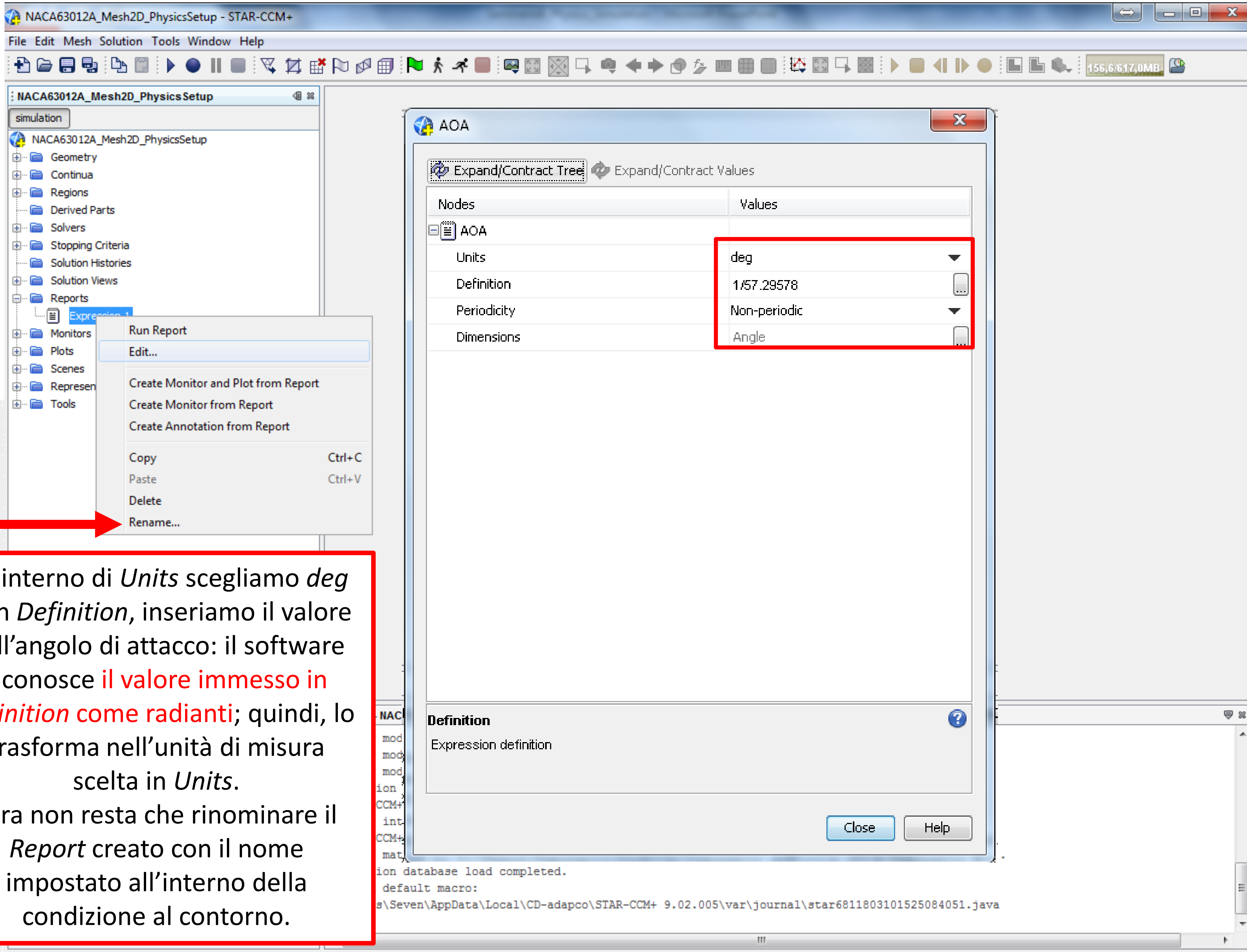


Il *Report* in cui imposteremo il valore dell’angolo di attacco è un report di tipo *Expression*. Il *Report* va generato cliccando con il tasto destro nell’ambito dei Report. I Report di default sono molteplici e alcuni andremo a definirli più avanti *(Force Coefficient, Moment Coefficient,...).*

Alcuni Report possono essere utili per valutare i tempi di calcolo della simulazione.

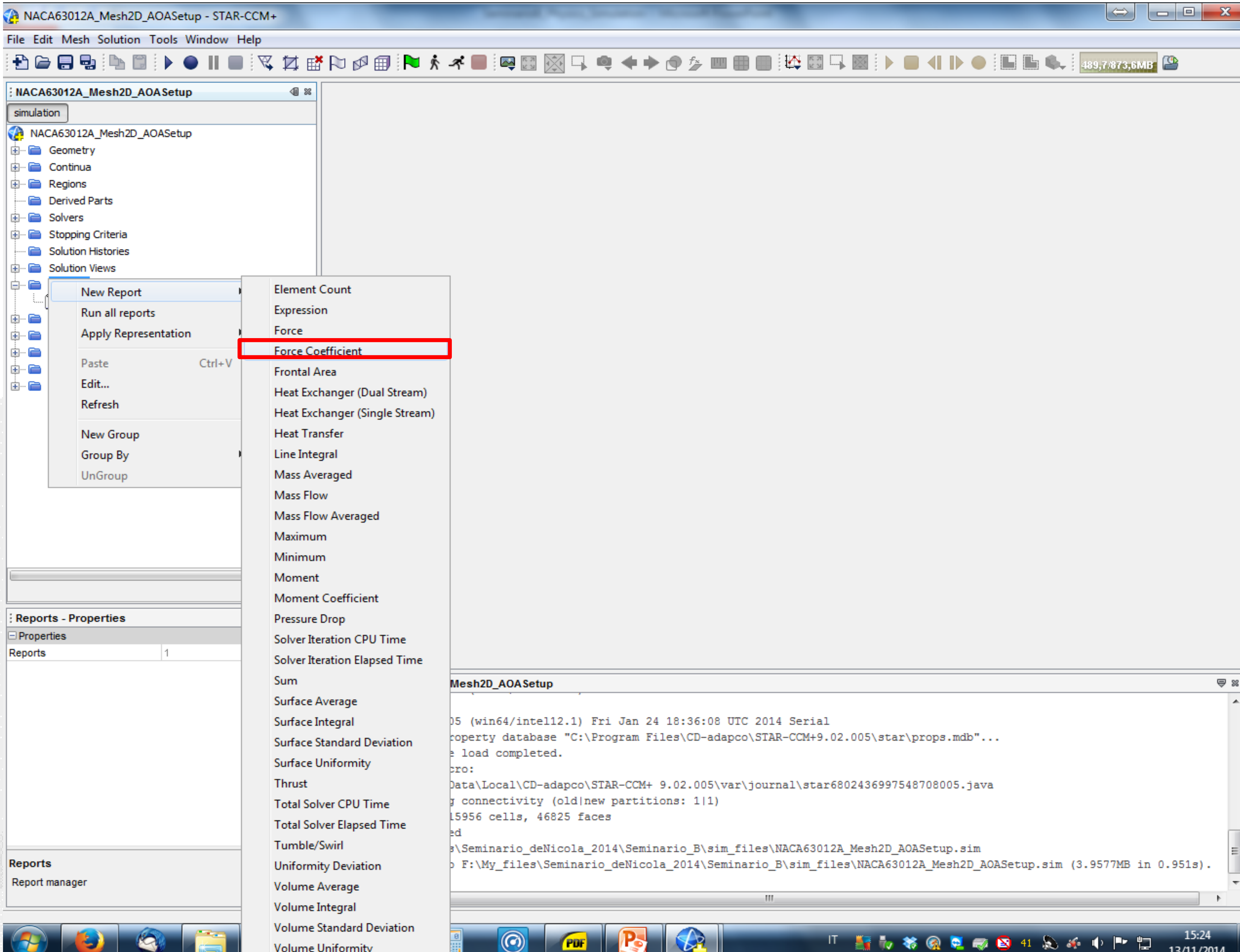


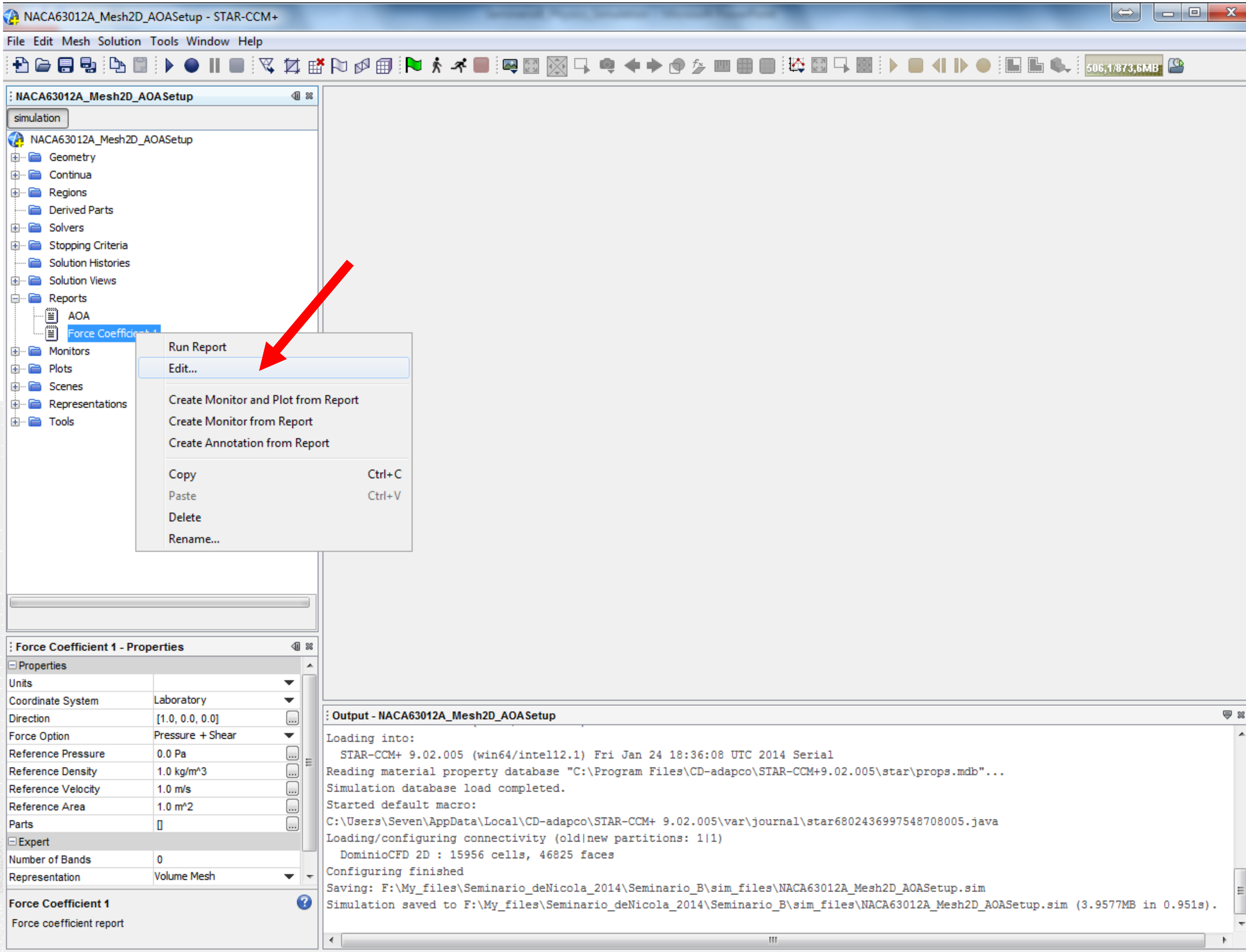
Anzitutto, definiamo che si tratta di un angolo, all'interno di *Dimensions*.

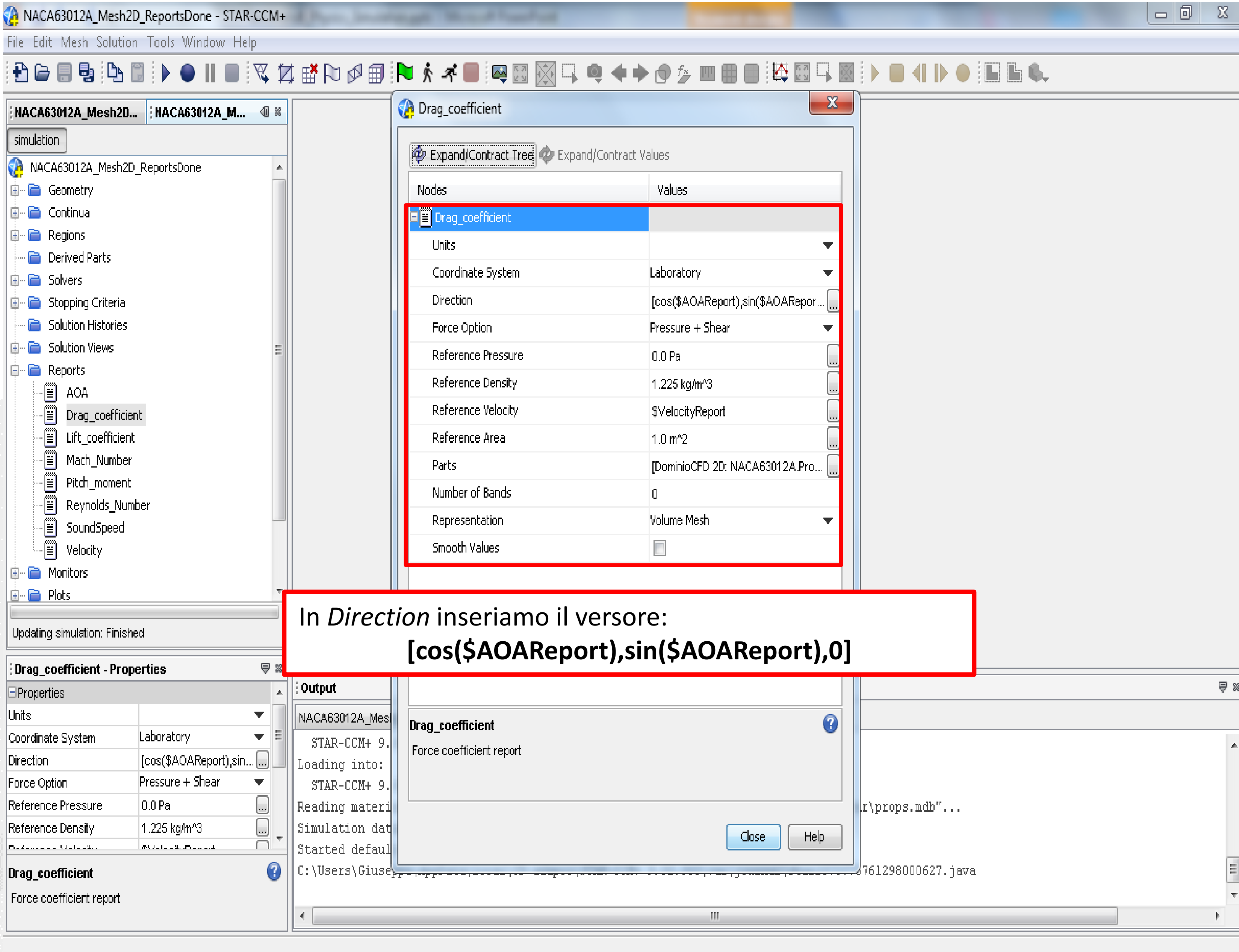


All'interno di *Units* scegliamo *deg* e, in *Definition*, inseriamo il valore dell'angolo di attacco: il software riconosce il valore immesso in *Definition* come radianti; quindi, lo trasforma nell'unità di misura scelta in *Units*.

Ora non resta che rinominare il *Report* creato con il nome impostato all'interno della condizione al contorno.

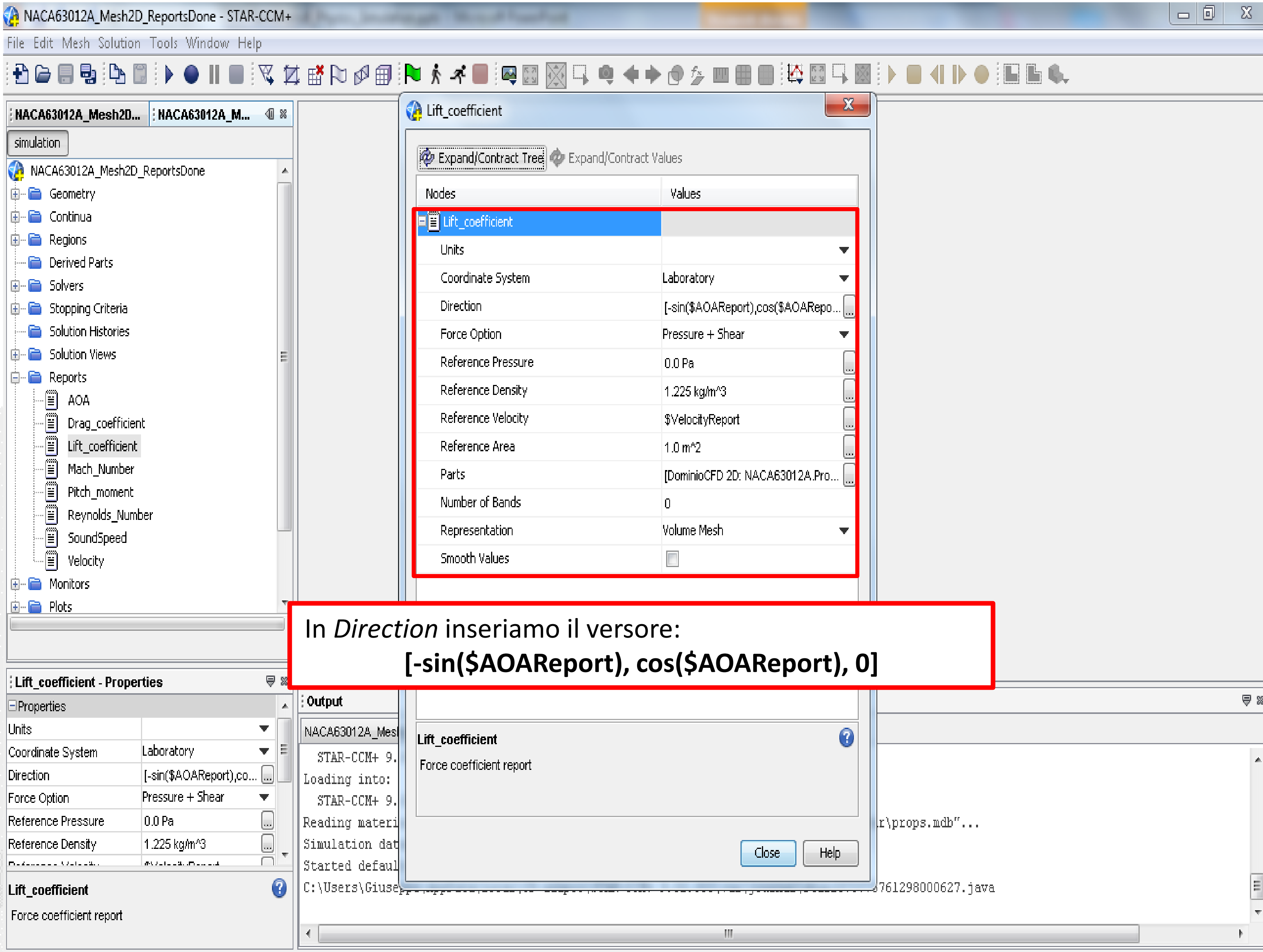

NACA63012A_Mesh2D_AOASetup - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_AOASetup
simulation
NACA63012A_Mesh2D_AOASetup
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
New Report
Run all reports
Apply Representation
Paste Ctrl+V
Edit...
Refresh
New Group
Group By
UnGroup
Element Count
Expression
Force
Force Coefficient
Frontal Area
Heat Exchanger (Dual Stream)
Heat Exchanger (Single Stream)
Heat Transfer
Line Integral
Mass Averaged
Mass Flow
Mass Flow Averaged
Maximum
Minimum
Moment
Moment Coefficient
Pressure Drop
Solver Iteration CPU Time
Solver Iteration Elapsed Time
Sum
Surface Average
Surface Integral
Surface Standard Deviation
Surface Uniformity
Thrust
Total Solver CPU Time
Total Solver Elapsed Time
Tumble/Swirl
Uniformity Deviation
Volume Average
Volume Integral
Volume Standard Deviation
Volume Uniformity
Reports - Properties
Properties
Reports 1
Reports
Report manager
Mesh2D_AOASetup
05 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Serial
roperty database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
e load completed.
cro:
Data\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star6802436997548708005.java
g connectivity (old|new partitions: 1|1)
15956 cells, 46825 faces
ed
s\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_AOASetup.sim
o F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_AOASetup.sim (3.9577MB in 0.951s).
15:24
13/11/2014


NACA63012A_Mesh2D_AOASetup - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_AOASetup
simulation
NACA63012A_Mesh2D_AOASetup
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Force Coefficient 1
Monitors
Plots
Scenes
Representations
Tools
Run Report
Edit...
Create Monitor and Plot from Report
Create Monitor from Report
Create Annotation from Report
Copy Ctrl+C
Paste Ctrl+V
Delete
Rename...
Force Coefficient 1 - Properties
Properties
Units
Coordinate System Laboratory
Direction [1.0, 0.0, 0.0]
Force Option Pressure + Shear
Reference Pressure 0.0 Pa
Reference Density 1.0 kg/m^3
Reference Velocity 1.0 m/s
Reference Area 1.0 m^2
Parts []
Expert
Number of Bands 0
Representation Volume Mesh
Force Coefficient 1
Force coefficient report
Output - NACA63012A_Mesh2D_AOASetup
Loading into:
STAR-CCM+ 9.02.005 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Serial
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Simulation database load completed.
Started default macro:
C:\Users\Seven\AppData\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star6802436997548708005.java
Loading/configuring connectivity (old|new partitions: 1|1)
DominioCFD 2D : 15956 cells, 46825 faces
Configuring finished
Saving: F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_AOASetup.sim
Simulation saved to F:\My_files\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_AOASetup.sim (3.9577MB in 0.951s).



In *Direction* inseriamo il versore:

[cos($AOAReport),sin($AOAReport),0]



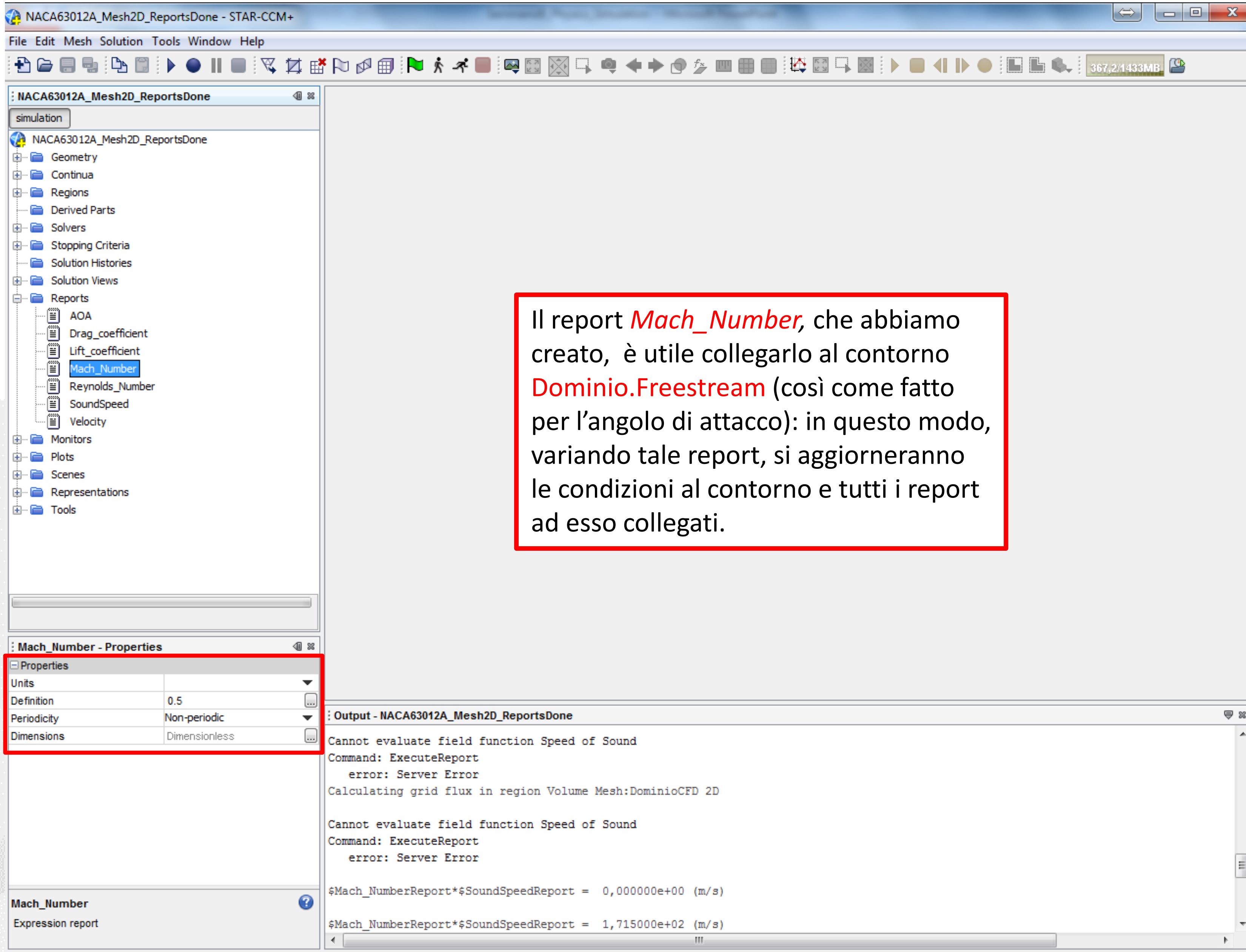
In *Direction* inseriamo il versore:

[-sin($AOAReport), cos($AOAReport), 0]

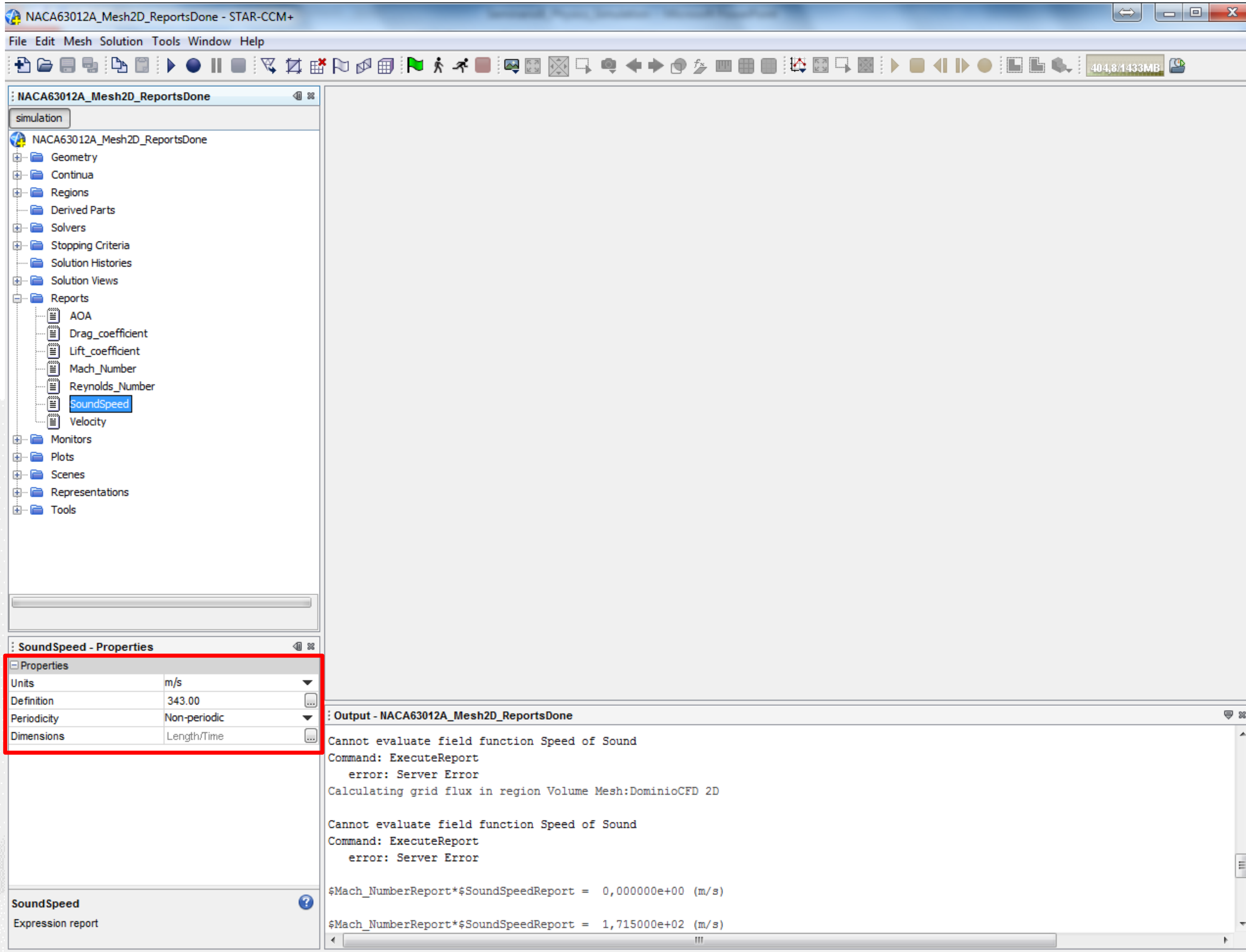
DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

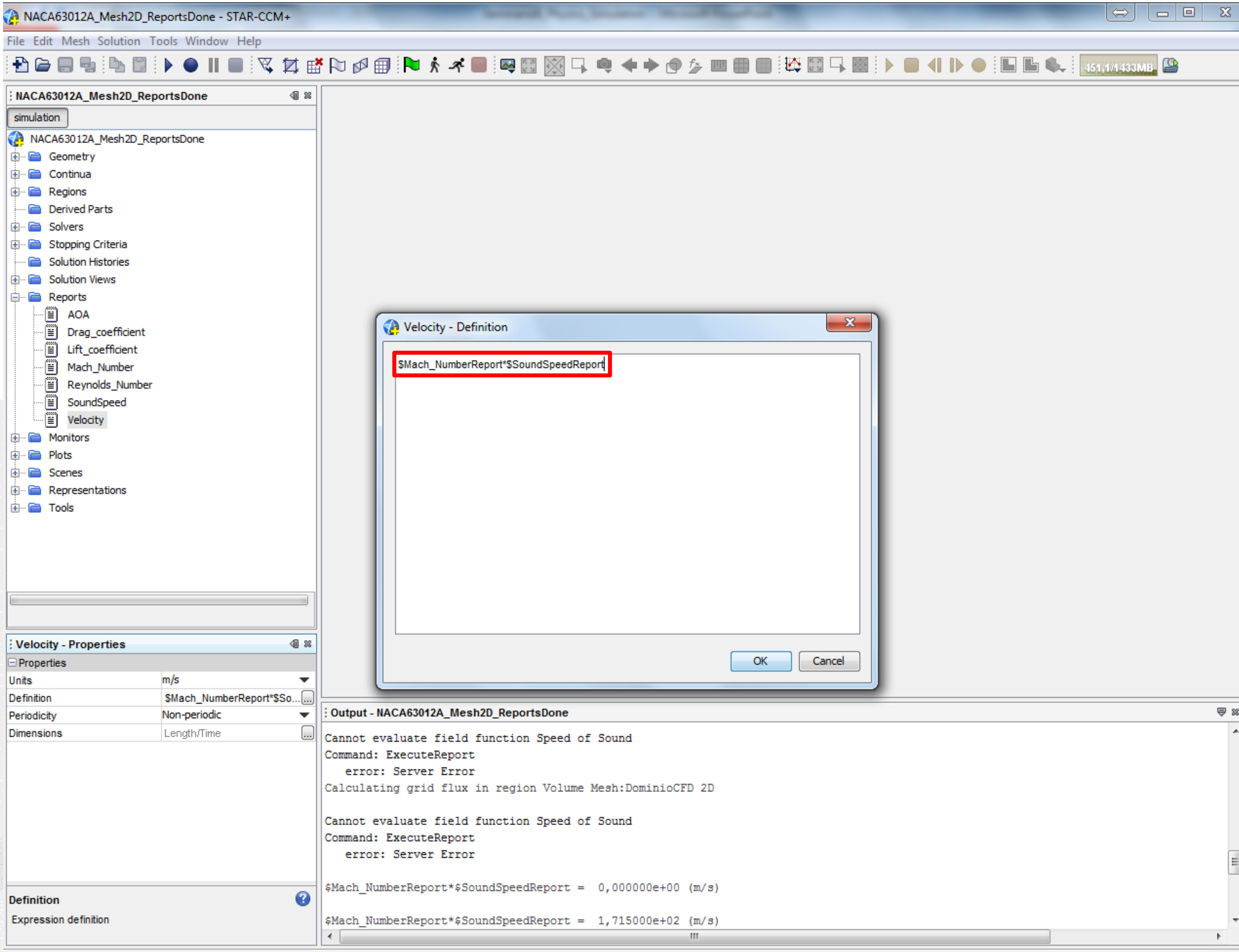
SEZIONE INGEGNERIA AEROSPAZIALE

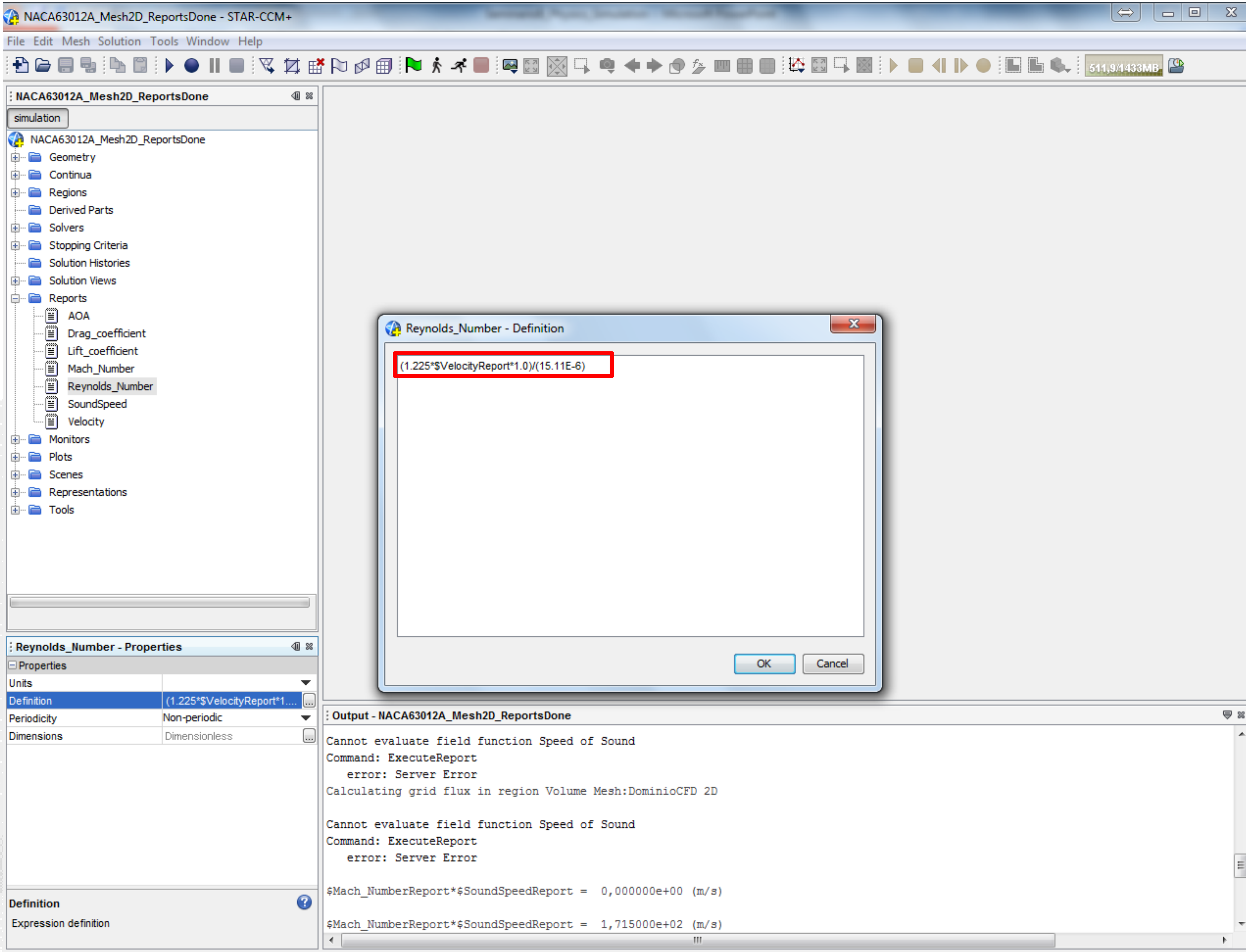
ADAG Aircraft Design & AeroFlightDynamics Group

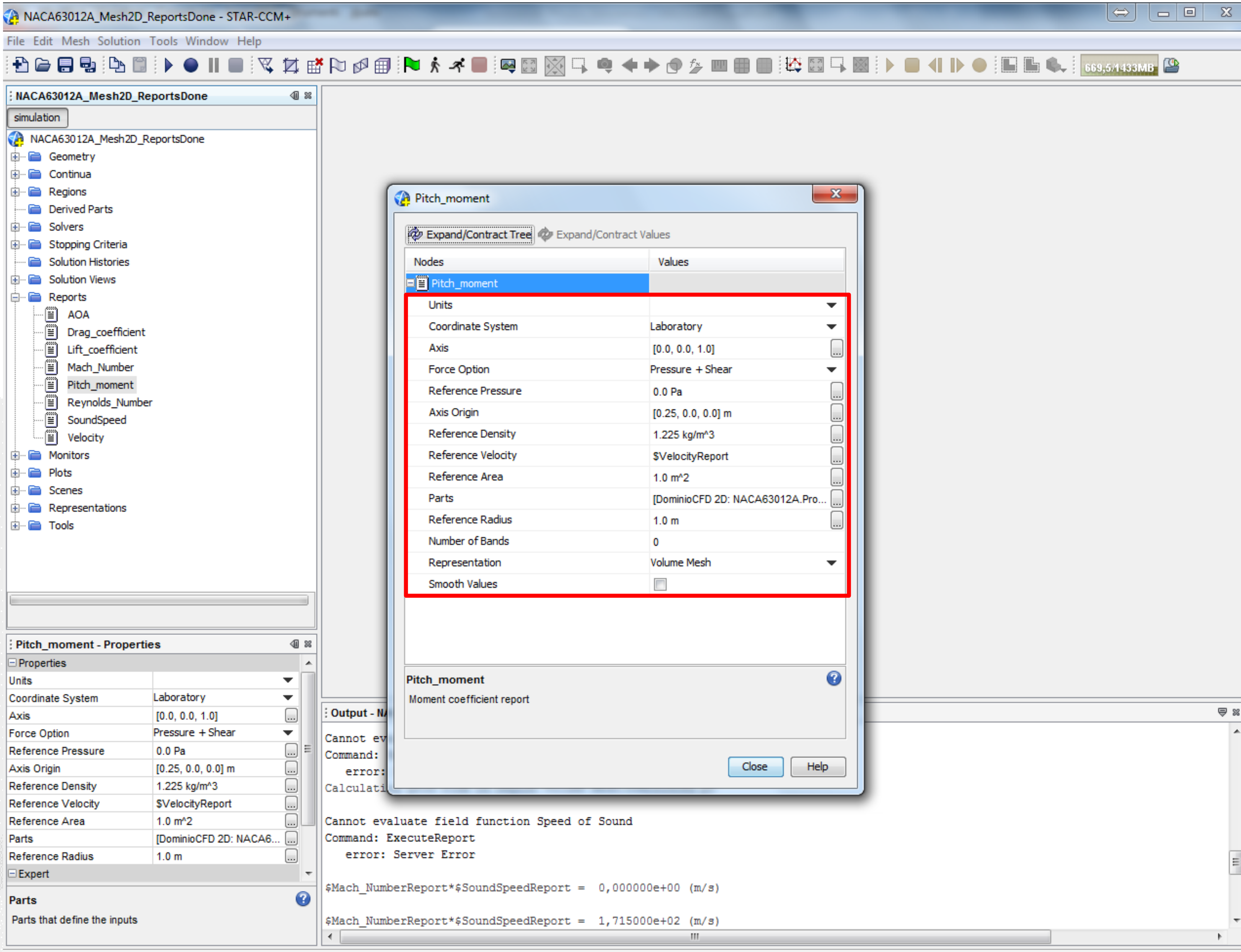


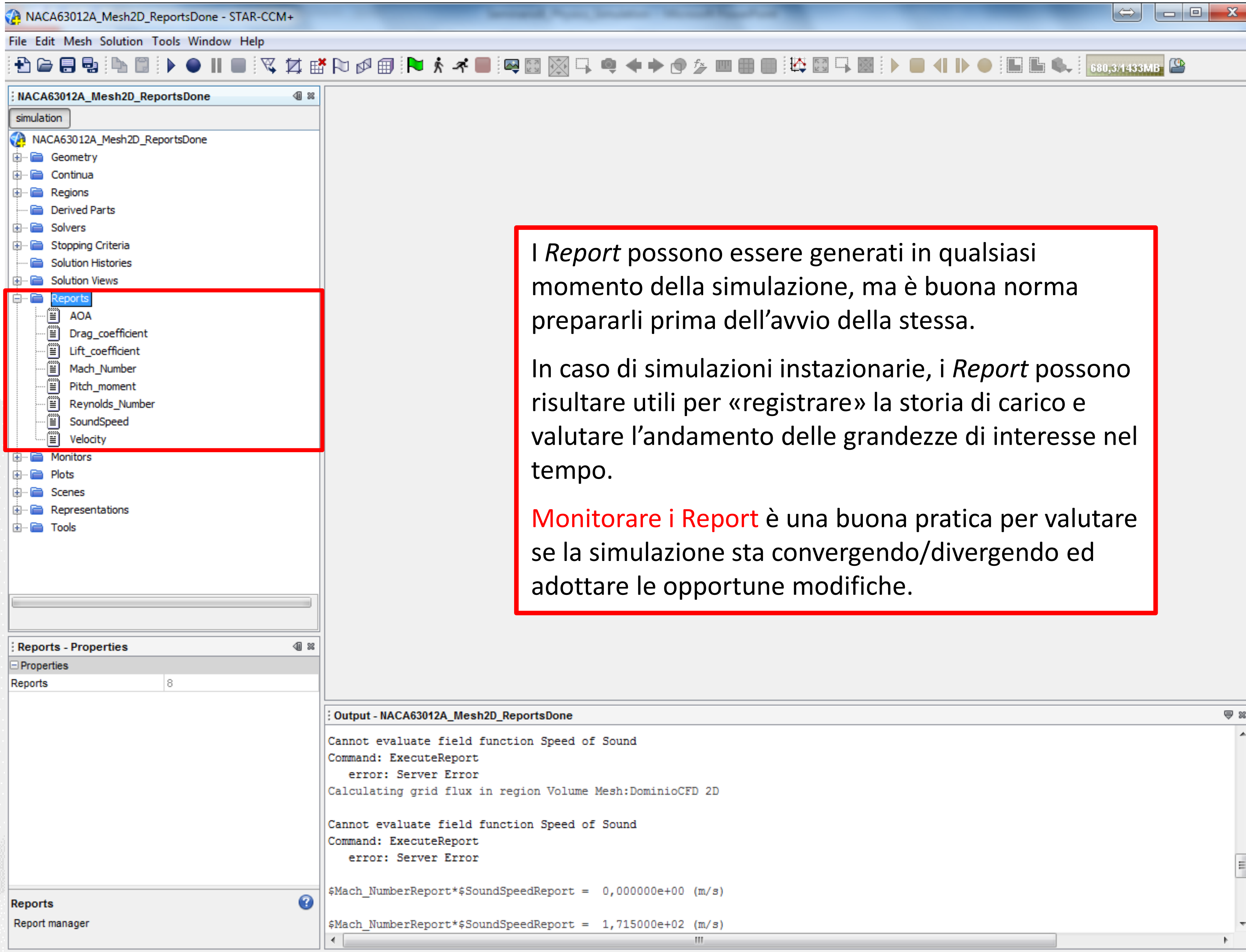
Il report *Mach_Number*, che abbiamo creato, è utile collegarlo al contorno Dominio.Freestream (così come fatto per l'angolo di attacco): in questo modo, variando tale report, si aggiorneranno le condizioni al contorno e tutti i report ad esso collegati.


NACA63012A_Mesh2D_ReportsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_ReportsDone
simulation
NACA63012A_Mesh2D_ReportsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Drag_coefficient
Lift_coefficient
Mach_Number
Reynolds_Number
SoundSpeed
Velocity
Monitors
Plots
Scenes
Representations
Tools
SoundSpeed - Properties
Properties
Units m/s
Definition 343.00
Periodicity Non-periodic
Dimensions Length/Time
SoundSpeed
Expression report
Output - NACA63012A_Mesh2D_ReportsDone
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
Calculating grid flux in region Volume Mesh:DominioCFD 2D
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
$Mach_NumberReport*$SoundSpeedReport = 0,000000e+00 (m/s)
$Mach_NumberReport*$SoundSpeedReport = 1,715000e+02 (m/s)


NACA63012A_Mesh2D_ReportsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_ReportsDone
simulation
NACA63012A_Mesh2D_ReportsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Drag_coefficient
Lift_coefficient
Mach_Number
Reynolds_Number
SoundSpeed
Velocity
Monitors
Plots
Scenes
Representations
Tools
Velocity - Properties
Properties
Units
m/s
Definition
$Mach_NumberReport*$So...
Periodicity
Non-periodic
Dimensions
Length/Time
Definition
Expression definition
Velocity - Definition
$Mach_NumberReport*$SoundSpeedReport
OK
Cancel
Output - NACA63012A_Mesh2D_ReportsDone
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
Calculating grid flux in region Volume Mesh:DominioCFD 2D
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
$Mach_NumberReport*$SoundSpeedReport = 0,000000e+00 (m/s)
$Mach_NumberReport*$SoundSpeedReport = 1,715000e+02 (m/s)


NACA63012A_Mesh2D_ReportsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_ReportsDone
simulation
NACA63012A_Mesh2D_ReportsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Drag_coefficient
Lift_coefficient
Mach_Number
Reynolds_Number
SoundSpeed
Velocity
Monitors
Plots
Scenes
Representations
Tools
Reynolds_Number - Definition
(1.225*$VelocityReport*1.0)/(15.11E-6)
OK
Cancel
Reynolds_Number - Properties
Properties
Units
Definition
(1.225*$VelocityReport*1....
Periodicity
Non-periodic
Dimensions
Dimensionless
Definition
Expression definition
Output - NACA63012A_Mesh2D_ReportsDone
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
Calculating grid flux in region Volume Mesh:DominioCFD 2D
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
$Mach_NumberReport*$SoundSpeedReport = 0,000000e+00 (m/s)
$Mach_NumberReport*$SoundSpeedReport = 1,715000e+02 (m/s)

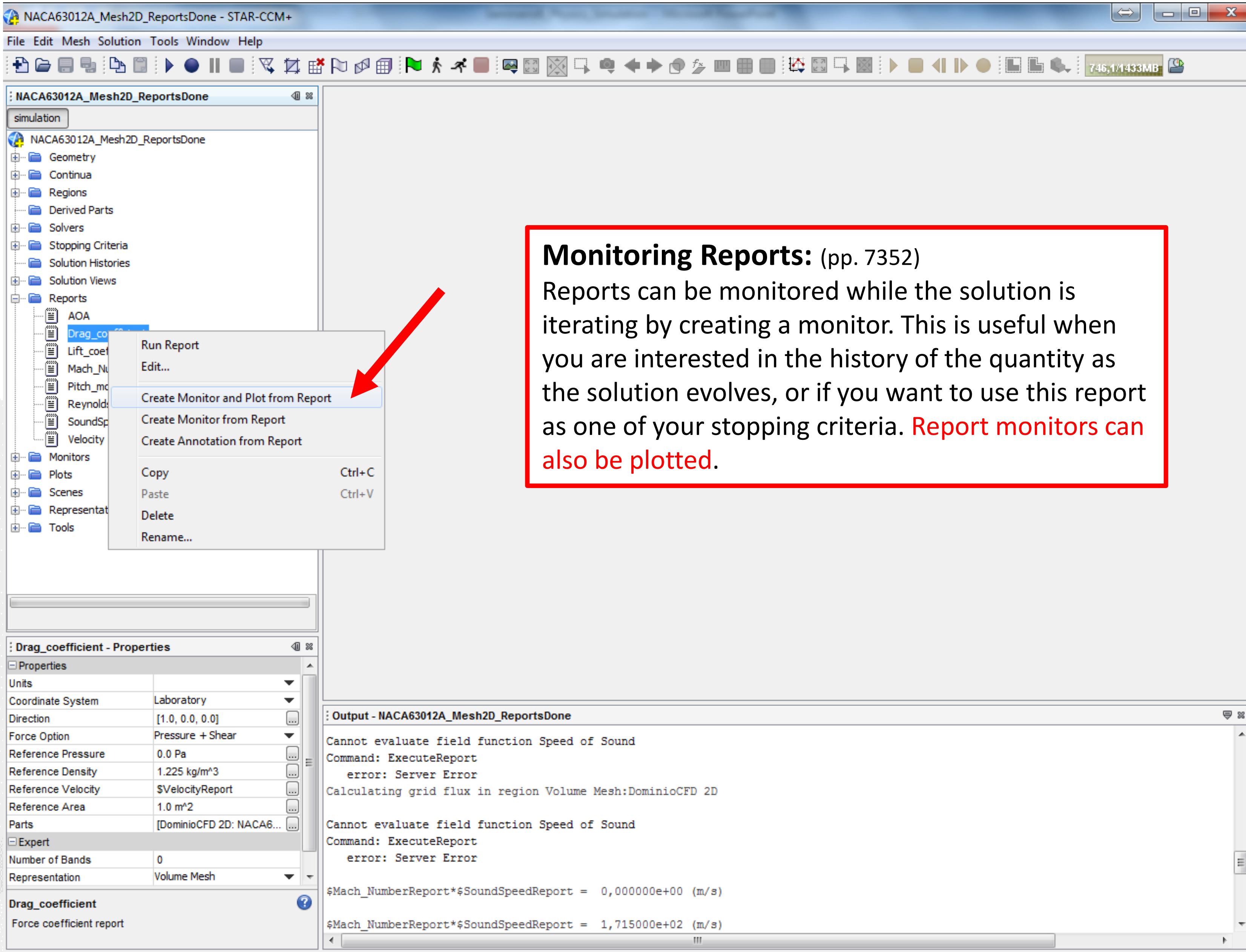

NACA63012A_Mesh2D_ReportsDone - STAR-CCM+
NACA63012A_Mesh2D_ReportsDone
simulation
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Drag_coefficient
Lift_coefficient
Mach_Number
Pitch_moment
Reynolds_Number
SoundSpeed
Velocity
Monitors
Plots
Scenes
Representations
Tools
Pitch_moment - Properties
Pitch_moment
Expand/Contract Tree
Expand/Contract Values
Nodes
Values
Units
Coordinate System
Laboratory
Axis
[0.0, 0.0, 1.0]
Force Option
Pressure + Shear
Reference Pressure
0.0 Pa
Axis Origin
[0.25, 0.0, 0.0] m
Reference Density
1.225 kg/m^3
Reference Velocity
$VelocityReport
Reference Area
1.0 m^2
Parts
[DominioCFD 2D: NACA63012A.Pro...
Reference Radius
1.0 m
Number of Bands
0
Representation
Volume Mesh
Smooth Values
Moment coefficient report
Close
Help
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
$Mach_NumberReport*$SoundSpeedReport = 0,000000e+00 (m/s)
$Mach_NumberReport*$SoundSpeedReport = 1,715000e+02 (m/s)



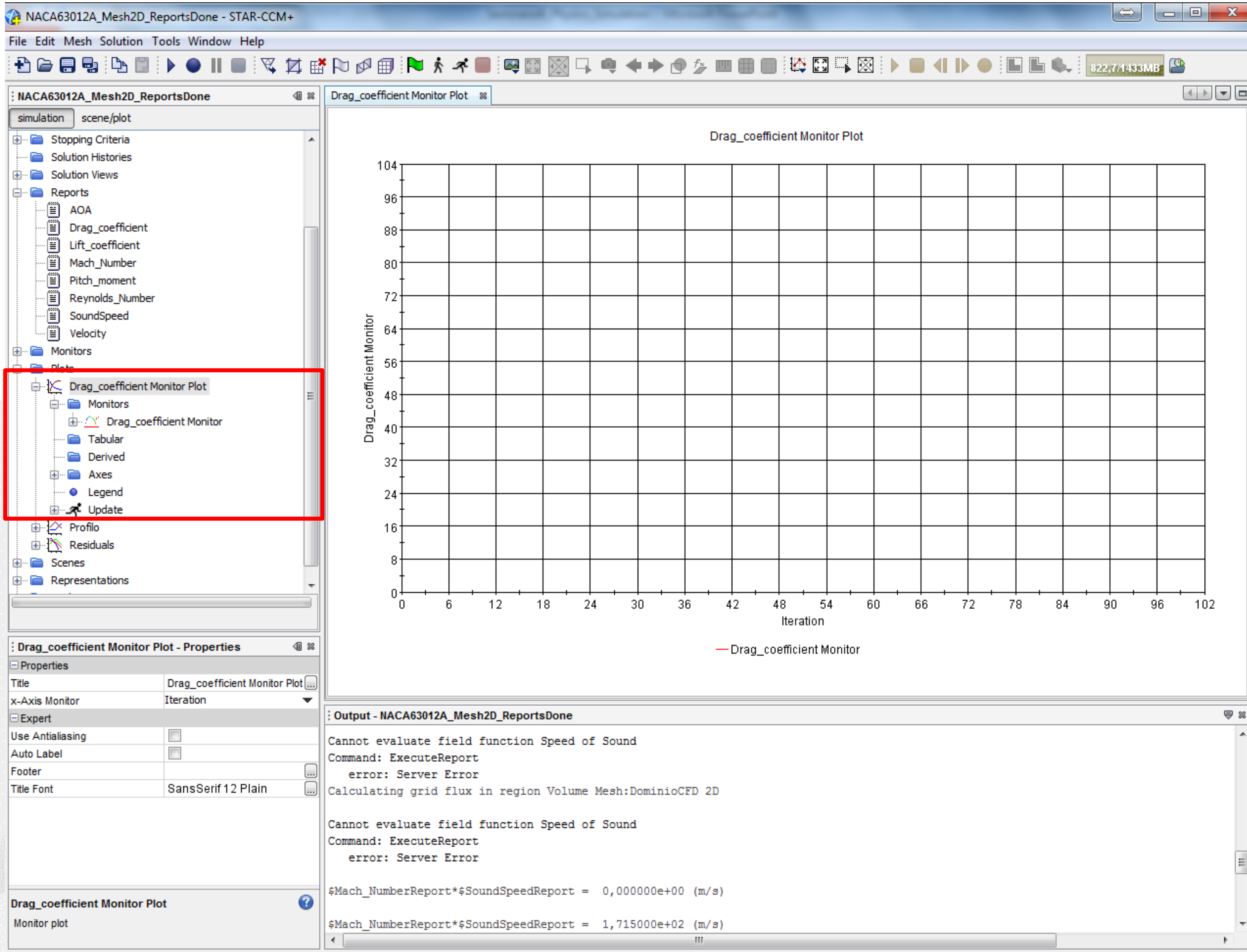
I *Report* possono essere generati in qualsiasi momento della simulazione, ma è buona norma prepararli prima dell’avvio della stessa.

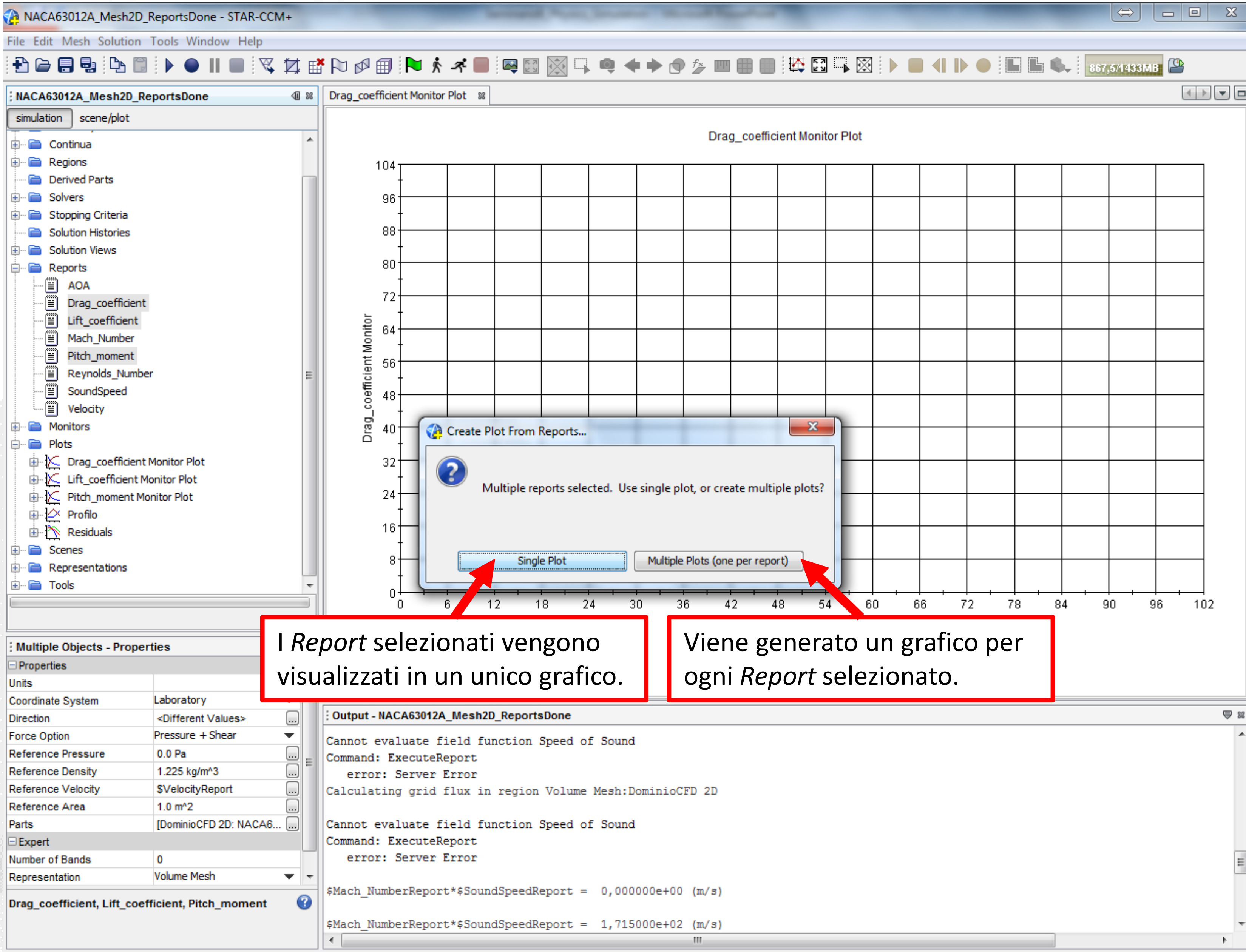
In caso di simulazioni instazionarie, i *Report* possono risultare utili per «registrare» la storia di carico e valutare l’andamento delle grandezze di interesse nel tempo.

Monitorare i Report è una buona pratica per valutare se la simulazione sta convergendo/divergendo ed adottare le opportune modifiche.



Monitoring Reports: (pp. 7352)
Reports can be monitored while the solution is iterating by creating a monitor. This is useful when you are interested in the history of the quantity as the solution evolves, or if you want to use this report as one of your stopping criteria. Report monitors can also be plotted.


NACA63012A_Mesh2D_ReportsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_ReportsDone
simulation
scene/plot
Stopping Criteria
Solution Histories
Solution Views
Reports
AOA
Drag_coefficient
Lift_coefficient
Mach_Number
Pitch_moment
Reynolds_Number
SoundSpeed
Velocity
Monitors
Drag_coefficient Monitor Plot
Monitors
Drag_coefficient Monitor
Tabular
Derived
Axes
Legend
Update
Profilo
Residuals
Scenes
Representations
Drag_coefficient Monitor Plot - Properties
Properties
Title
Drag_coefficient Monitor Plot
x-Axis Monitor
Iteration
Expert
Use Antialiasing
Auto Label
Footer
Title Font
SansSerif 12 Plain
Drag_coefficient Monitor Plot
Monitor plot
Drag_coefficient Monitor Plot
Drag_coefficient Monitor
Iteration
Drag_coefficient Monitor
Output - NACA63012A_Mesh2D_ReportsDone
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
Calculating grid flux in region Volume Mesh:DominioCFD 2D
Cannot evaluate field function Speed of Sound
Command: ExecuteReport
error: Server Error
$Mach_NumberReport*$SoundSpeedReport = 0,000000e+00 (m/s)
$Mach_NumberReport*$SoundSpeedReport = 1,715000e+02 (m/s)



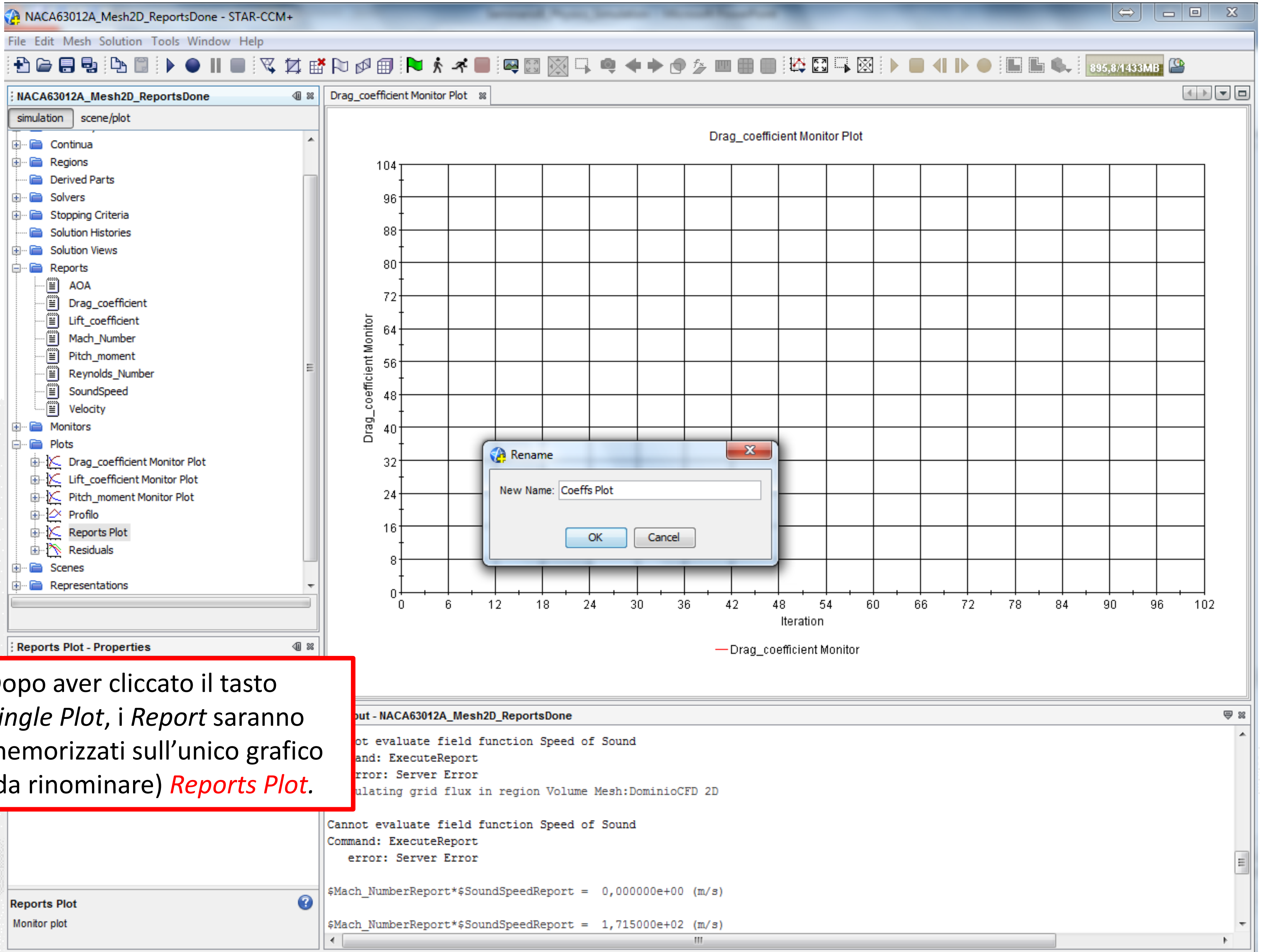
I *Report* selezionati vengono visualizzati in un unico grafico.

Viene generato un grafico per ogni *Report* selezionato.

DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

SEZIONE INGEGNERIA AEROSPAZIALE

ADAG Aircraft Design & AeroFlightDynamics Group



Dopo aver cliccato il tasto *Single Plot*, i *Report* saranno memorizzati sull'unico grafico (da rinominare) *Reports Plot.*

Plotting Results (pp. 7450)

(STAR-CCM+ User Guide)

The Plots node, which has a pop-up menu, is the manager object for all monitor plots and XY plots that have been created in the simulation. It exists even when empty to allow creation of the first plot, but will typically contain a Residuals plot, which is created automatically once you start iterating.

The plotting features in STAR-CCM+ allow you to create three kinds of two-dimensional plots:

- Monitor plots that display data from the simulation as it steps through the solution in two varieties:
  - Monitor plots based on monitored quantities
  - Residual plots (described elsewhere)
- XY plots that use solution data from the simulation and/or table data
- Histogram plots that display data, typically for particles or parcels

If the plot is open in the Graphics window while you are customizing properties, the relevant object on the plot display will change as you enter the new value. This is a useful way to become familiar with the various properties. All plots, along with any customization, are saved in the simulation file. You can also customize how an XY plot or histogram plot is updated as the simulation iterates. The plot display is highly interactive, having its own toolbar, zoom, pan and pop-up operations as well as being linked to the simulation tree through drag-and-drop operations. These operations include plotting data from a table and plotting data from a data set, which can also be done using the Properties window. The data being plotted can be viewed in a spreadsheet-like dialog or it can be exported to file for manipulation in other programs. The plot can also be printed to file or to a printer.

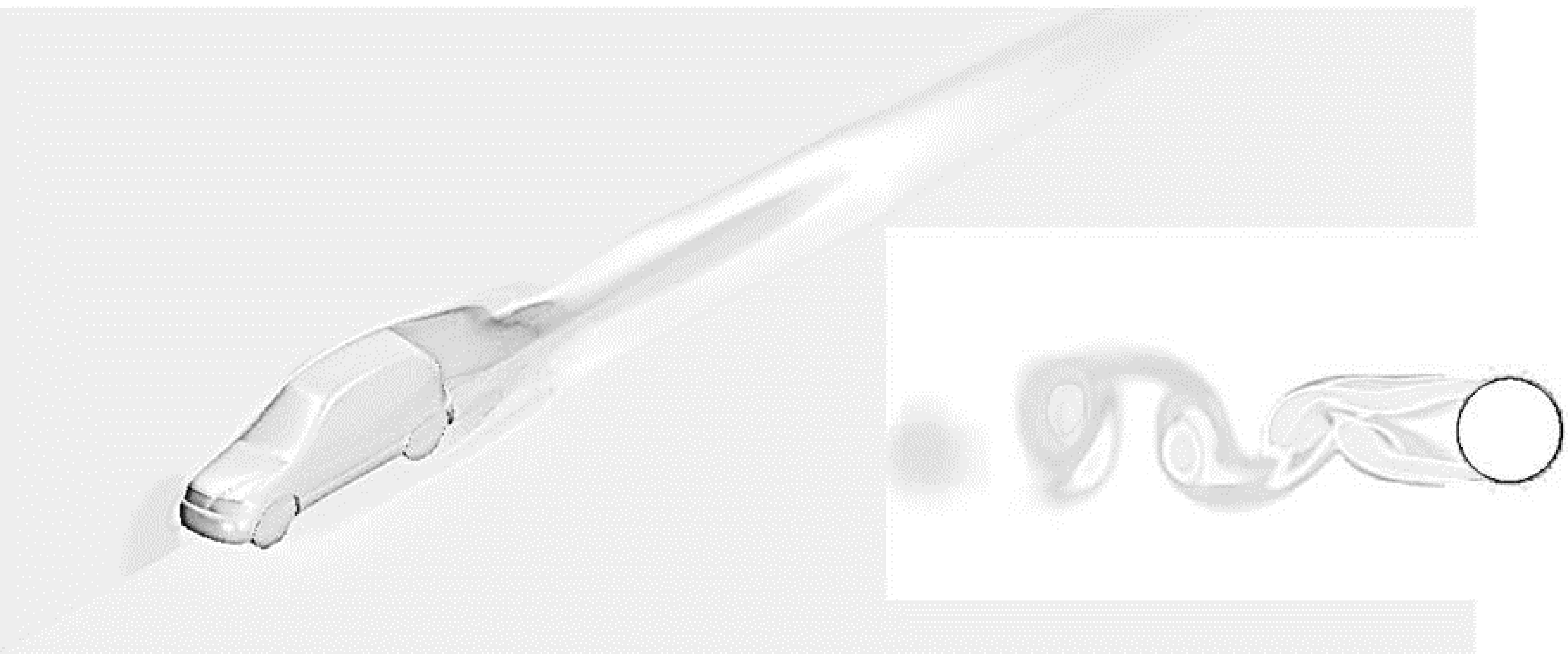
What Is a Residual Plot? (pp. 6654)

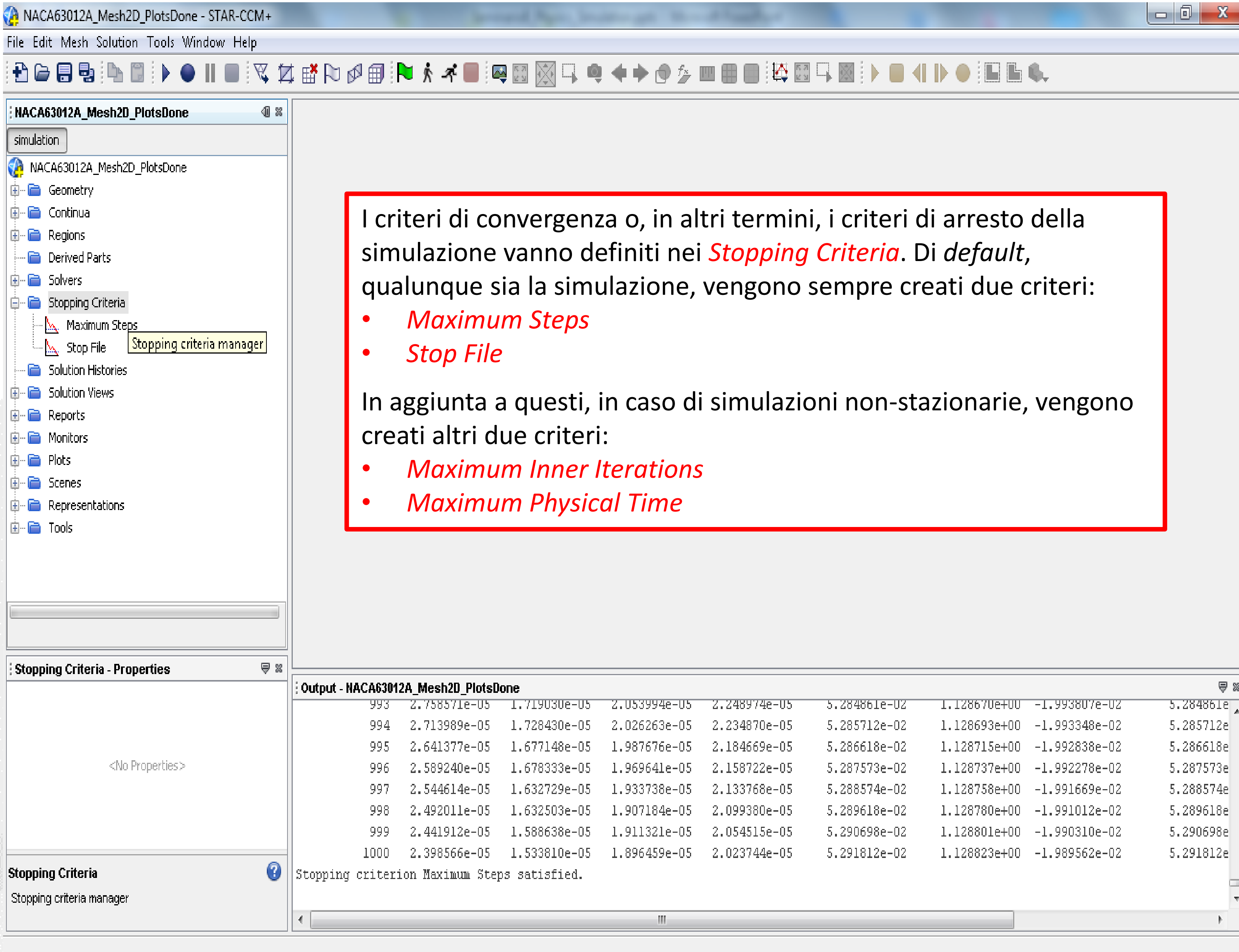
The residual plot is a monitor plot that is automatically created from the active residual monitors on semi-log axes when iterating starts. By default, all active residuals are displayed in the residual plot. The residual plot can be renamed but it cannot be deleted, though individual data series can be deleted from the plot.

What Is a Residual? (pp. 6650)

The residual in each cell represents the degree to which the discretized equation is satisfied. When the solver is run, a discretized version of the transport equations is solved for each cell in the mesh. Residual monitors keep a record of this global quantity for each of the transport equations solved in the continua within the simulation.

Impostazione dei criteri di convergenza e avvio del calcolo





I criteri di convergenza o, in altri termini, i criteri di arresto della simulazione vanno definiti nei *Stopping Criteria*. Di *default*, qualunque sia la simulazione, vengono sempre creati due criteri:

- *Maximum Steps*
- *Stop File*

In aggiunta a questi, in caso di simulazioni non-stazionarie, vengono creati altri due criteri:

- *Maximum Inner Iterations*
- *Maximum Physical Time*

Setting up Stopping Criteria (pp. 6617)

(STAR-CCM+ User Guide)

What Are Stopping Criteria?

Stopping criteria allow you to specify how long the solution runs for and under what conditions it stops iterating and/or marching in time. Each specified stopping criterion is evaluated at the completion of every simulation step and a logical rule is used to determine if the interaction of all of the criteria stops the solver.

Certain stopping criteria are generated automatically when a solver is chosen. For steady simulations, there are two stopping criteria:

- Maximum Steps: The Maximum Steps stopping criterion allows you to specify the maximum number of iterations in a steady solver or the maximum number of time-steps in an unsteady solver. The stopping decision is based on the number of steps that the solver executes, including any steps that are executed in a previous session. If you Clear the solution, the counter resets to zero, if you initialize the solution it does not. (pp. 6619)
- Stop File: The Stop File criterion allows you to specify the pathname of a file (named ABORT by default) that, once in place, causes the solver to stop. This action can be useful for stopping a batch job, for example. (pp. 6620)

For unsteady simulations, other two criteria are generated:

- Maximum Inner Iterations: The Maximum Inner Iterations stopping criterion is based on the number of inner iterations that the solver executes for transient analyses. The node of this criterion has its own properties, and appears when the implicit unsteady model is chosen. If the implicit unsteady solver is used, this stopping criterion can control the number of inner iterations to be run at each physical time-step. The Maximum Inner Iterations stopping criterion differs from other stopping criteria. This stopping criterion does not control when to stop the solver, only when to stop the inner iterations of the implicit solver and march the solution to the next time-step.(pp. 6623)
- Maximum Physical Time: The Maximum Physical Time stopping criterion is based on the simulation time that has elapsed in a transient analysis. The node of this criterion has its own properties, and appears when either the implicit unsteady model or the explicit unsteady model is chosen. When used with the implicit unsteady model, this stopping criterion is linked to the Time-Step property of the Implicit Unsteady node in the Solvers node. If the Time-Step is set to 1 second, and the Maximum Physical Time is set to 10 seconds, then the simulation runs for 10 time-steps. (pp. 6625)

Automatically generated stopping criteria cannot be deleted, but the Enabled property can be activated or deactivated.

Setting up Stopping Criteria (cont.)

(STAR-CCM+ User Guide)

Using Monitors as Stopping Criteria (pp. 6630)

It is possible to create stopping criteria that are based on existing monitors. This function lets you use more meaningful criteria to judge convergence. For example, if you are simulating the flow over an airfoil you can stop iterating when the drag and lift coefficients of the airfoil have reached steady values.

You can create stopping criteria that are based on any existing monitor. These criteria check the value of the associated monitor and return a satisfied condition when that value reaches some user-specified minimum, maximum, asymptotic limit or standard deviation.

Criterion Option — The method to use for evaluating this criterion.

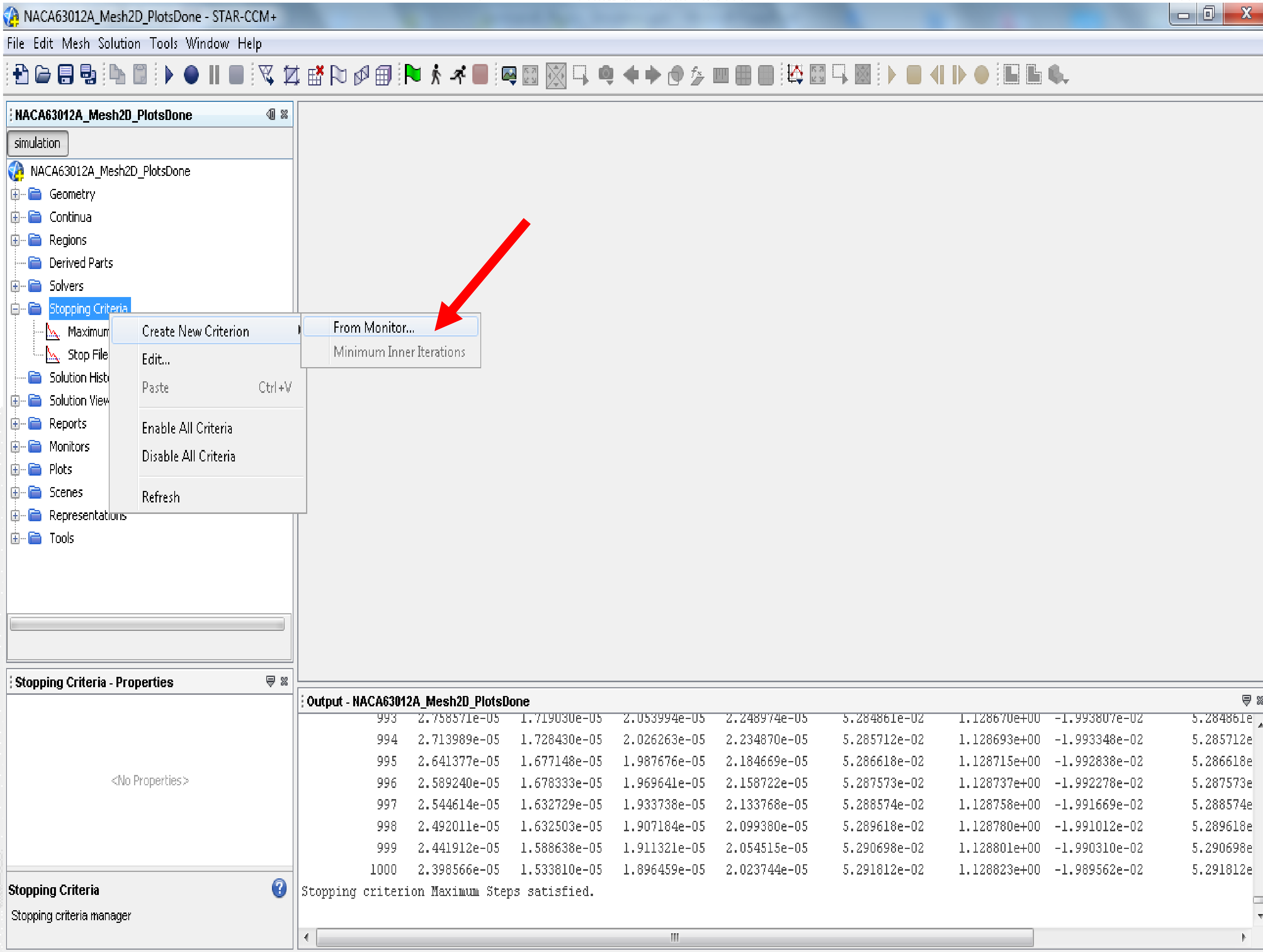
- Minimum: Specifies that this criterion is satisfied when the monitor reaches a user-specified minimum value. A Minimum Limit node is added as a sub-node.
- Maximum: Specifies that this criterion is satisfied when the monitor reaches a user-specified maximum value. A Maximum Limit node is added as a sub-node.
- Asymptotic: Specifies that this criterion is satisfied when the monitor has stabilized to a particular range during a number of iterations. The range and iteration window are specified in the Asymptotic Limit sub-node.
- Standard Deviation: Specifies that this criterion is satisfied when the monitor reaches a user-specified standard deviation. A Standard Deviation node is added as a sub-node.

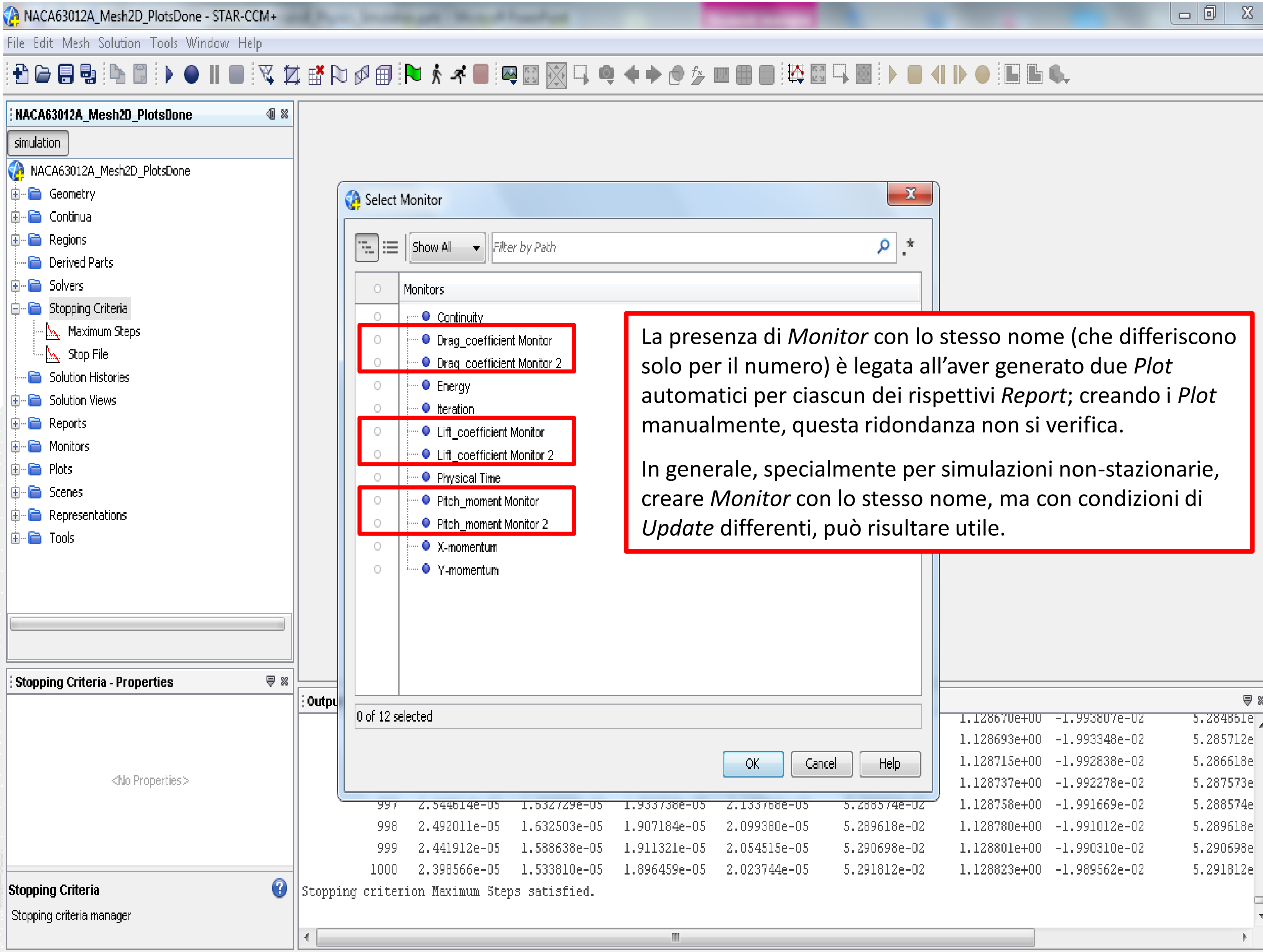
Using Multiple Stopping Criteria (pp. 6637)

If more than one stopping criterion is activated, it is necessary to construct a logical rule to determine when the criteria stops the solver. Logical rules (AND, OR) are assigned to each individual criterion and are used to determine how they interact as a group. If a criterion is assigned an OR logical rule, the solver stops when it is satisfied. The solver also stops when all the criteria assigned the AND logical rule are satisfied. By default. all criteria are assigned the OR logical rule.

Logical Rule — Defines how this criterion interacts with other stopping criteria that have been enabled.

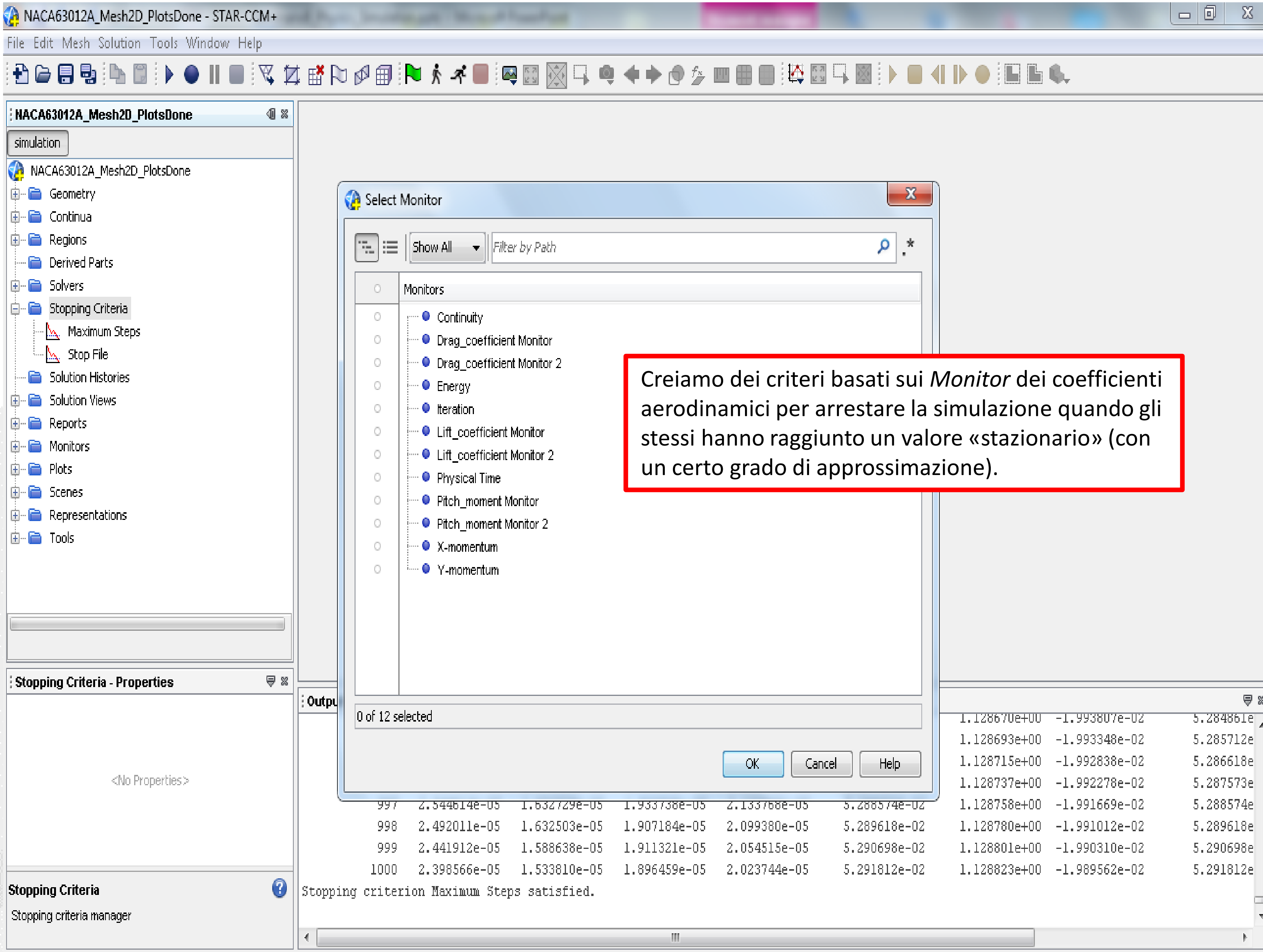
- AND: Requires that this criterion, along with one or more other criteria with the Logical Rule set to AND , all be satisfied before stopping the solver.
- OR: Indicates that satisfying only this criterion is sufficient to stop the solver


NACA63012A_Mesh2D_PlotsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PlotsDone
simulation
NACA63012A_Mesh2D_PlotsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Maximum
Stop File
Solution Hist
Solution View
Reports
Monitors
Plots
Scenes
Representations
Tools
Create New Criterion
Edit...
Paste Ctrl+V
Enable All Criteria
Disable All Criteria
Refresh
From Monitor...
Minimum Inner Iterations
Stopping Criteria - Properties
<No Properties>
Stopping Criteria
Stopping criteria manager
Output - NACA63012A_Mesh2D_PlotsDone
994 2.713989e-05 1.728430e-05 2.026263e-05 2.234870e-05 5.285712e-02 1.128693e+00 -1.993348e-02 5.285712e
995 2.641377e-05 1.677148e-05 1.987676e-05 2.184669e-05 5.286618e-02 1.128715e+00 -1.992838e-02 5.286618e
996 2.589240e-05 1.678333e-05 1.969641e-05 2.158722e-05 5.287573e-02 1.128737e+00 -1.992278e-02 5.287573e
997 2.544614e-05 1.632729e-05 1.933738e-05 2.133768e-05 5.288574e-02 1.128758e+00 -1.991669e-02 5.288574e
998 2.492011e-05 1.632503e-05 1.907184e-05 2.099380e-05 5.289618e-02 1.128780e+00 -1.991012e-02 5.289618e
999 2.441912e-05 1.588638e-05 1.911321e-05 2.054515e-05 5.290698e-02 1.128801e+00 -1.990310e-02 5.290698e
1000 2.398566e-05 1.533810e-05 1.896459e-05 2.023744e-05 5.291812e-02 1.128823e+00 -1.989562e-02 5.291812e
Stopping criterion Maximum Steps satisfied.

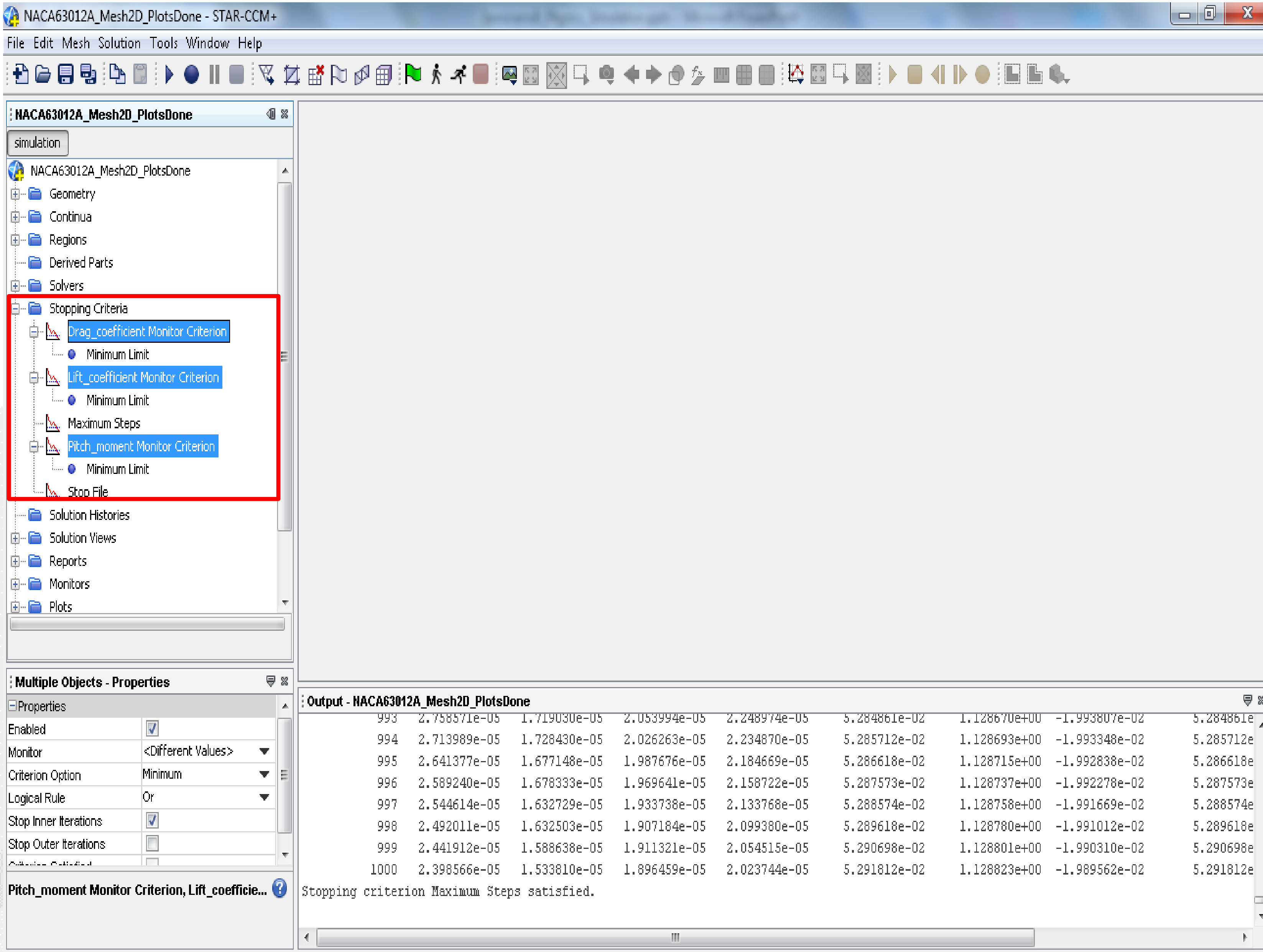


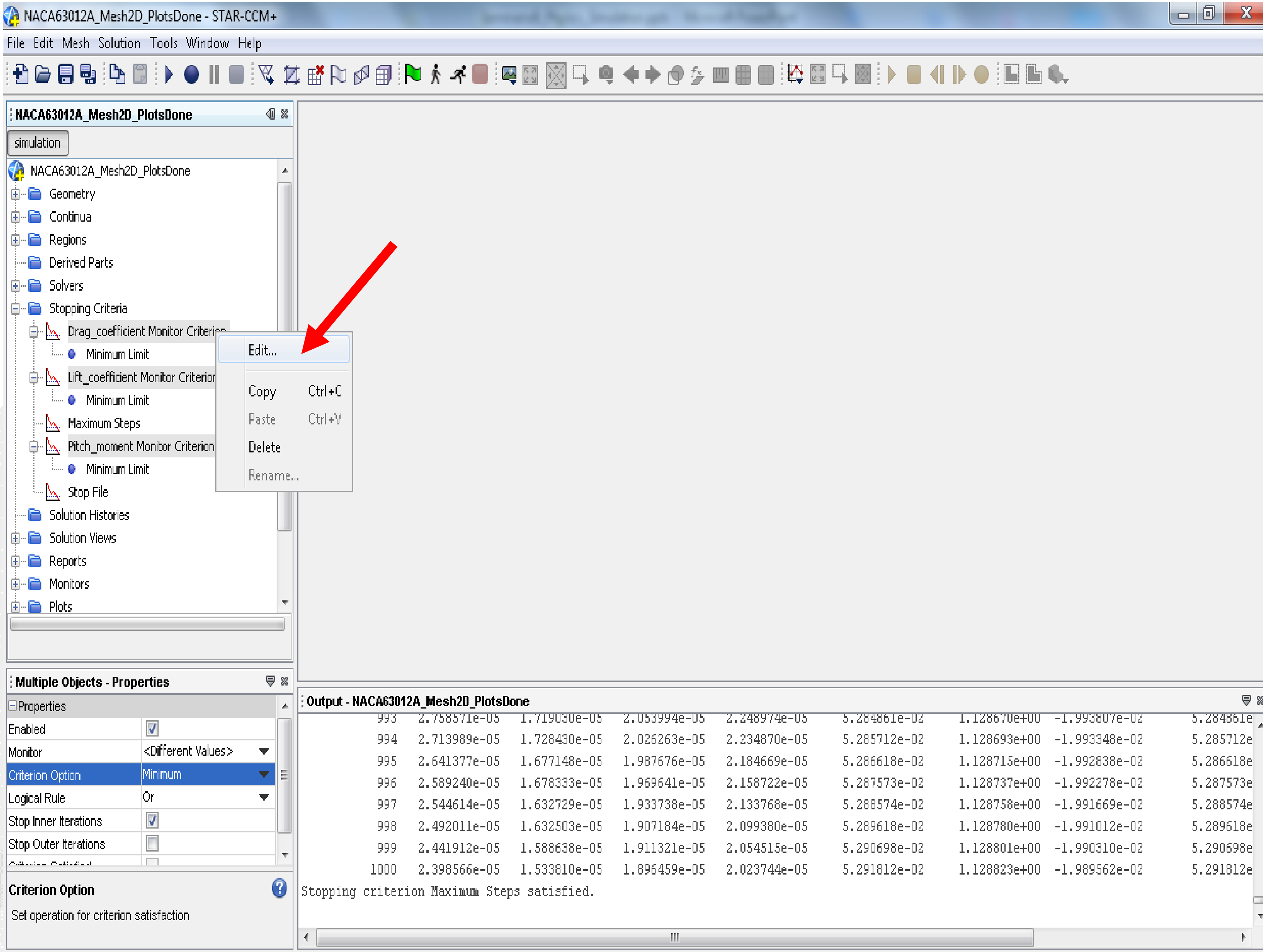
La presenza di *Monitor* con lo stesso nome (che differiscono solo per il numero) è legata all'aver generato due *Plot* automatici per ciascun dei rispettivi *Report*; creando i *Plot* manualmente, questa ridondanza non si verifica.

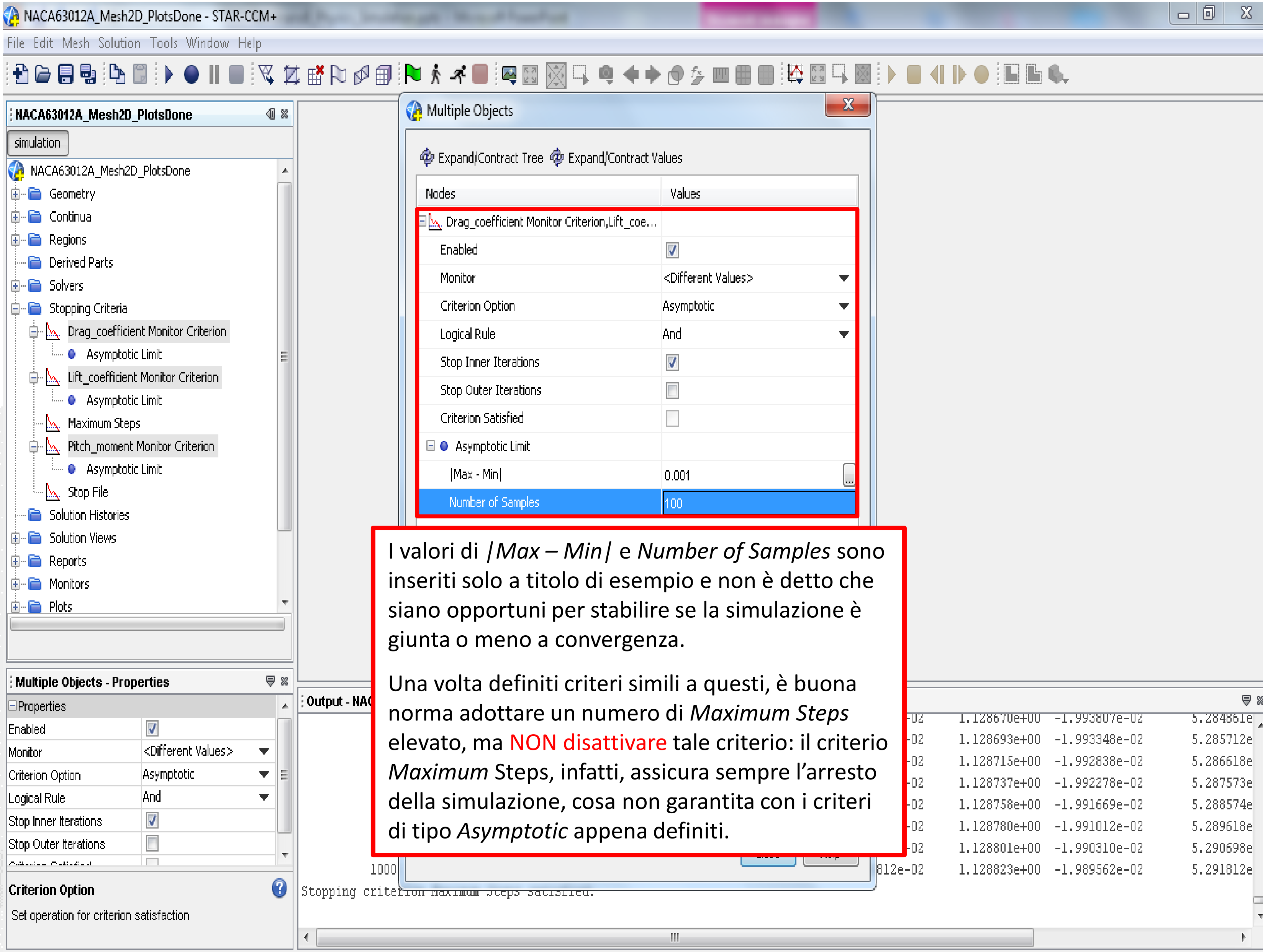
In generale, specialmente per simulazioni non-stazionarie, creare *Monitor* con lo stesso nome, ma con condizioni di *Update* differenti, può risultare utile.



Creiamo dei criteri basati sui *Monitor* dei coefficienti aerodinamici per arrestare la simulazione quando gli stessi hanno raggiunto un valore «stazionario» (con un certo grado di approssimazione).

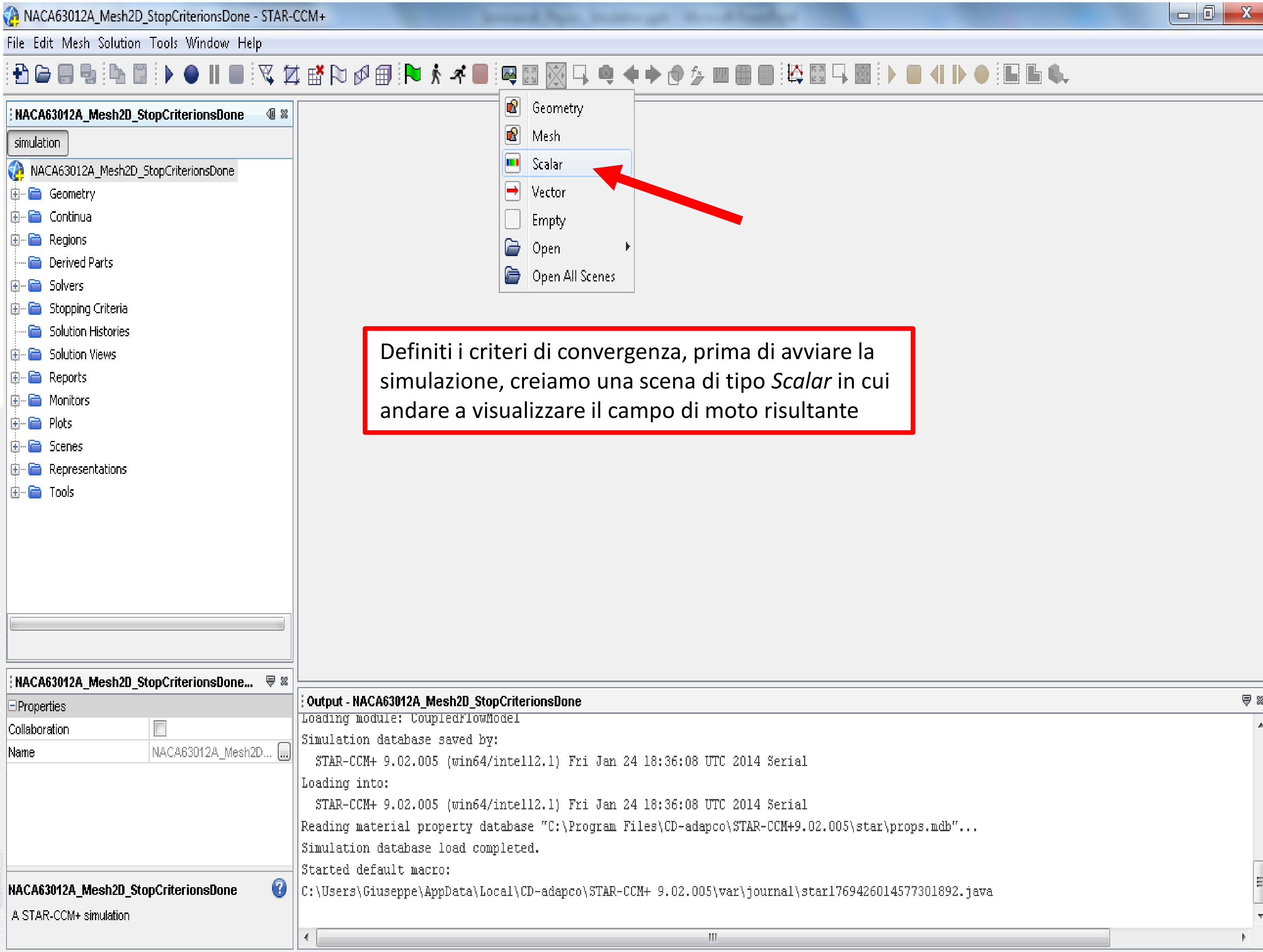

NACA63012A_Mesh2D_PlotsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PlotsDone
simulation
NACA63012A_Mesh2D_PlotsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Drag_coefficient Monitor Criterion
Minimum Limit
Lift_coefficient Monitor Criterion
Minimum Limit
Maximum Steps
Pitch_moment Monitor Criterion
Minimum Limit
Stop File
Solution Histories
Solution Views
Reports
Monitors
Plots
Multiple Objects - Properties
Properties
Enabled
Monitor
<Different Values>
Criterion Option
Minimum
Logical Rule
Or
Stop Inner Iterations
Stop Outer Iterations
Pitch_moment Monitor Criterion, Lift_coefficie...
Output - NACA63012A_Mesh2D_PlotsDone
994 2.713989e-05 1.728430e-05 2.026263e-05 2.234870e-05 5.285712e-02 1.128693e+00 -1.993348e-02 5.285712e
995 2.641377e-05 1.677148e-05 1.987676e-05 2.184669e-05 5.286618e-02 1.128715e+00 -1.992838e-02 5.286618e
996 2.589240e-05 1.678333e-05 1.969641e-05 2.158722e-05 5.287573e-02 1.128737e+00 -1.992278e-02 5.287573e
997 2.544614e-05 1.632729e-05 1.933738e-05 2.133768e-05 5.288574e-02 1.128758e+00 -1.991669e-02 5.288574e
998 2.492011e-05 1.632503e-05 1.907184e-05 2.099380e-05 5.289618e-02 1.128780e+00 -1.991012e-02 5.289618e
999 2.441912e-05 1.588638e-05 1.911321e-05 2.054515e-05 5.290698e-02 1.128801e+00 -1.990310e-02 5.290698e
1000 2.398566e-05 1.533810e-05 1.896459e-05 2.023744e-05 5.291812e-02 1.128823e+00 -1.989562e-02 5.291812e
Stopping criterion Maximum Steps satisfied.


NACA63012A_Mesh2D_PlotsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PlotsDone
simulation
NACA63012A_Mesh2D_PlotsDone
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Drag_coefficient Monitor Criterion
Minimum Limit
Lift_coefficient Monitor Criterion
Minimum Limit
Maximum Steps
Pitch_moment Monitor Criterion
Minimum Limit
Stop File
Solution Histories
Solution Views
Reports
Monitors
Plots
Edit...
Copy Ctrl+C
Paste Ctrl+V
Delete
Rename...
Multiple Objects - Properties
Properties
Enabled
Monitor <Different Values>
Criterion Option Minimum
Logical Rule Or
Stop Inner Iterations
Stop Outer Iterations
Criterion Option
Set operation for criterion satisfaction
Output - NACA63012A_Mesh2D_PlotsDone
994 2.713989e-05 1.728430e-05 2.026263e-05 2.234870e-05 5.285712e-02 1.128693e+00 -1.993348e-02 5.285712e
995 2.641377e-05 1.677148e-05 1.987676e-05 2.184669e-05 5.286618e-02 1.128715e+00 -1.992838e-02 5.286618e
996 2.589240e-05 1.678333e-05 1.969641e-05 2.158722e-05 5.287573e-02 1.128737e+00 -1.992278e-02 5.287573e
997 2.544614e-05 1.632729e-05 1.933738e-05 2.133768e-05 5.288574e-02 1.128758e+00 -1.991669e-02 5.288574e
998 2.492011e-05 1.632503e-05 1.907184e-05 2.099380e-05 5.289618e-02 1.128780e+00 -1.991012e-02 5.289618e
999 2.441912e-05 1.588638e-05 1.911321e-05 2.054515e-05 5.290698e-02 1.128801e+00 -1.990310e-02 5.290698e
1000 2.398566e-05 1.533810e-05 1.896459e-05 2.023744e-05 5.291812e-02 1.128823e+00 -1.989562e-02 5.291812e
Stopping criterion Maximum Steps satisfied.



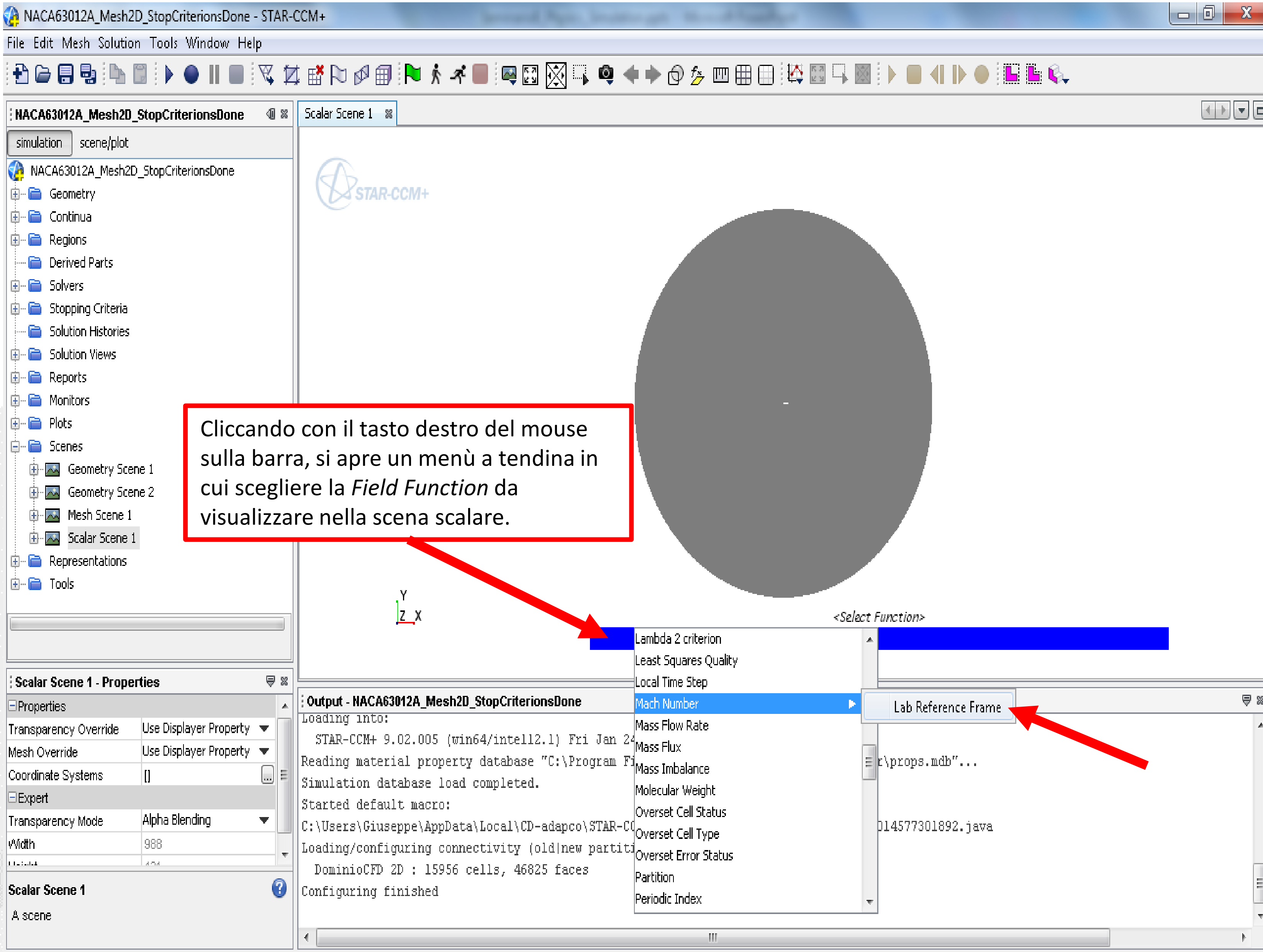
I valori di *|Max – Min|* e *Number of Samples* sono inseriti solo a titolo di esempio e non è detto che siano opportuni per stabilire se la simulazione è giunta o meno a convergenza.

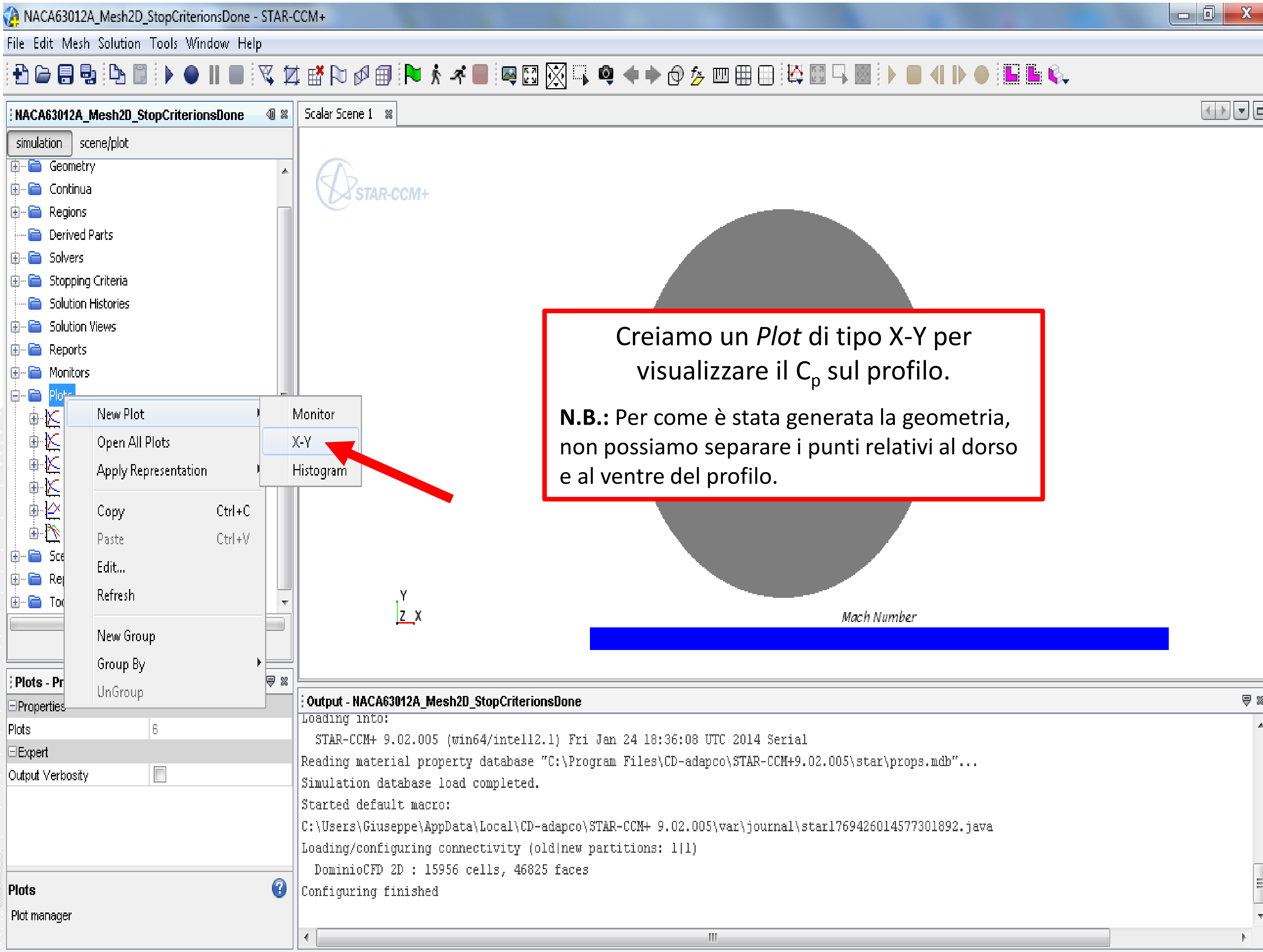
Una volta definiti criteri simili a questi, è buona norma adottare un numero di *Maximum Steps* elevato, ma NON disattivare tale criterio: il criterio *Maximum* Steps, infatti, assicura sempre l'arresto della simulazione, cosa non garantita con i criteri di tipo *Asymptotic* appena definiti.



Definiti i criteri di convergenza, prima di avviare la simulazione, creiamo una scena di tipo *Scalar* in cui andare a visualizzare il campo di moto risultante

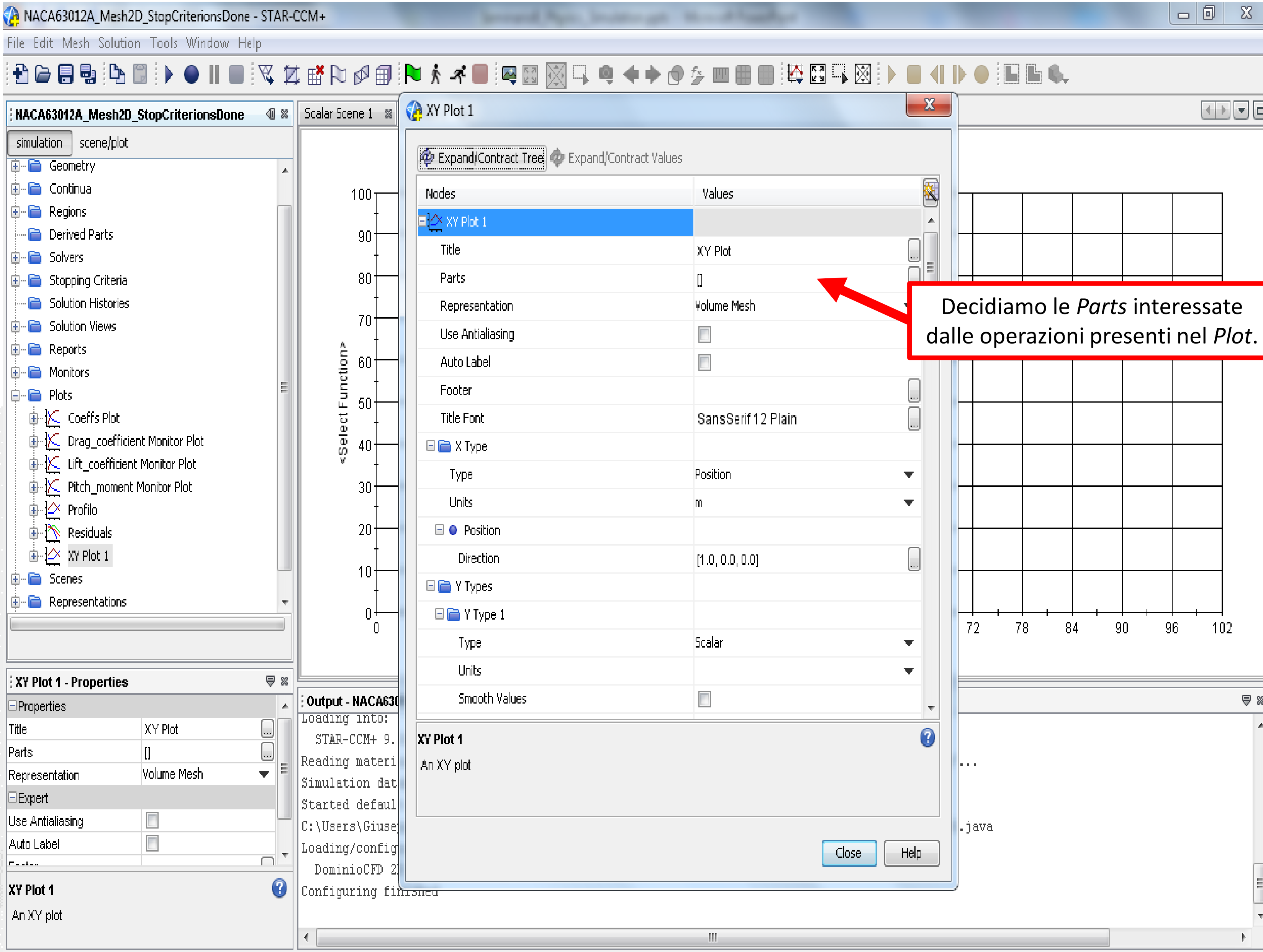
Cliccando con il tasto destro del mouse sulla barra, si apre un menù a tendina in cui scegliere la *Field Function* da visualizzare nella scena scalare.



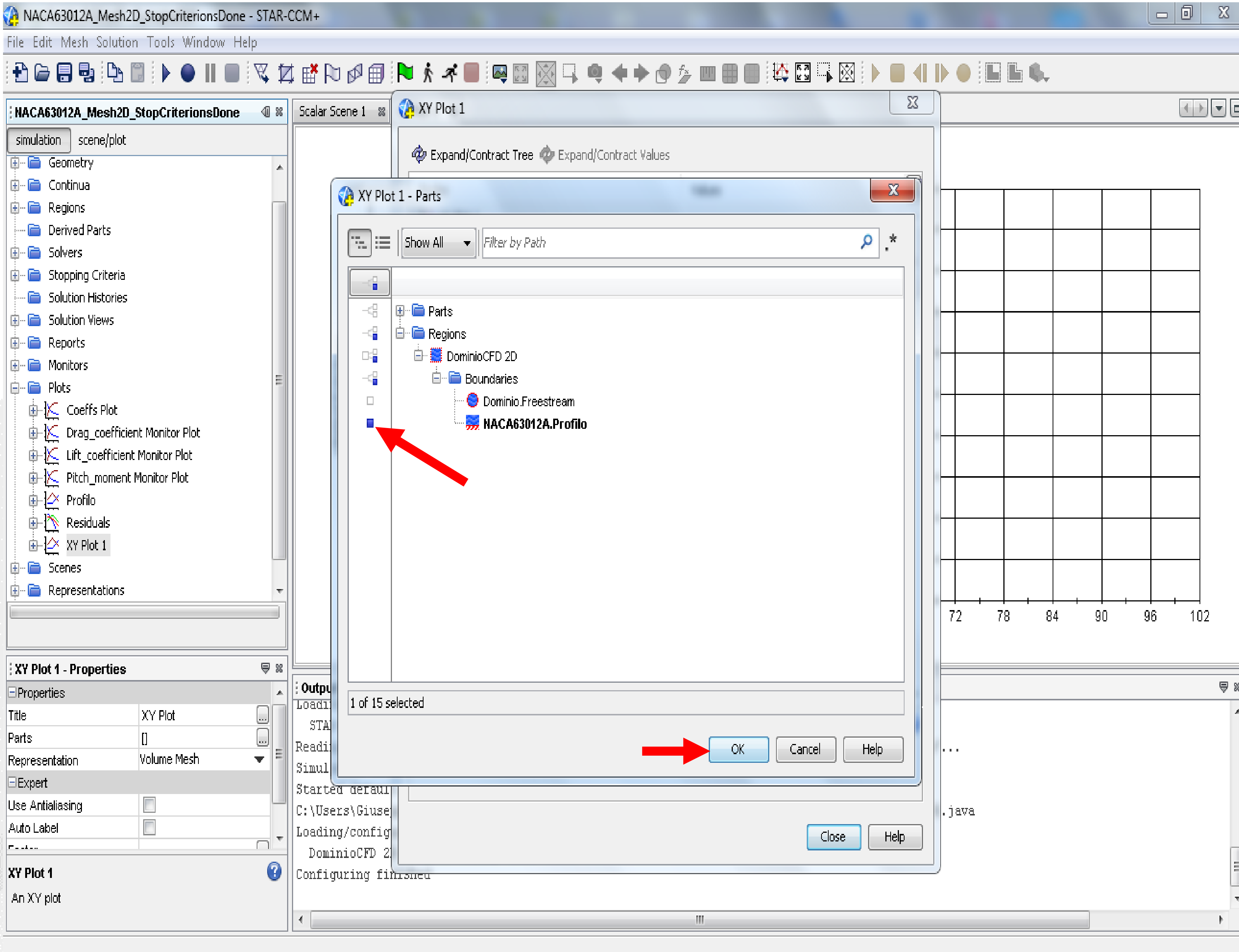


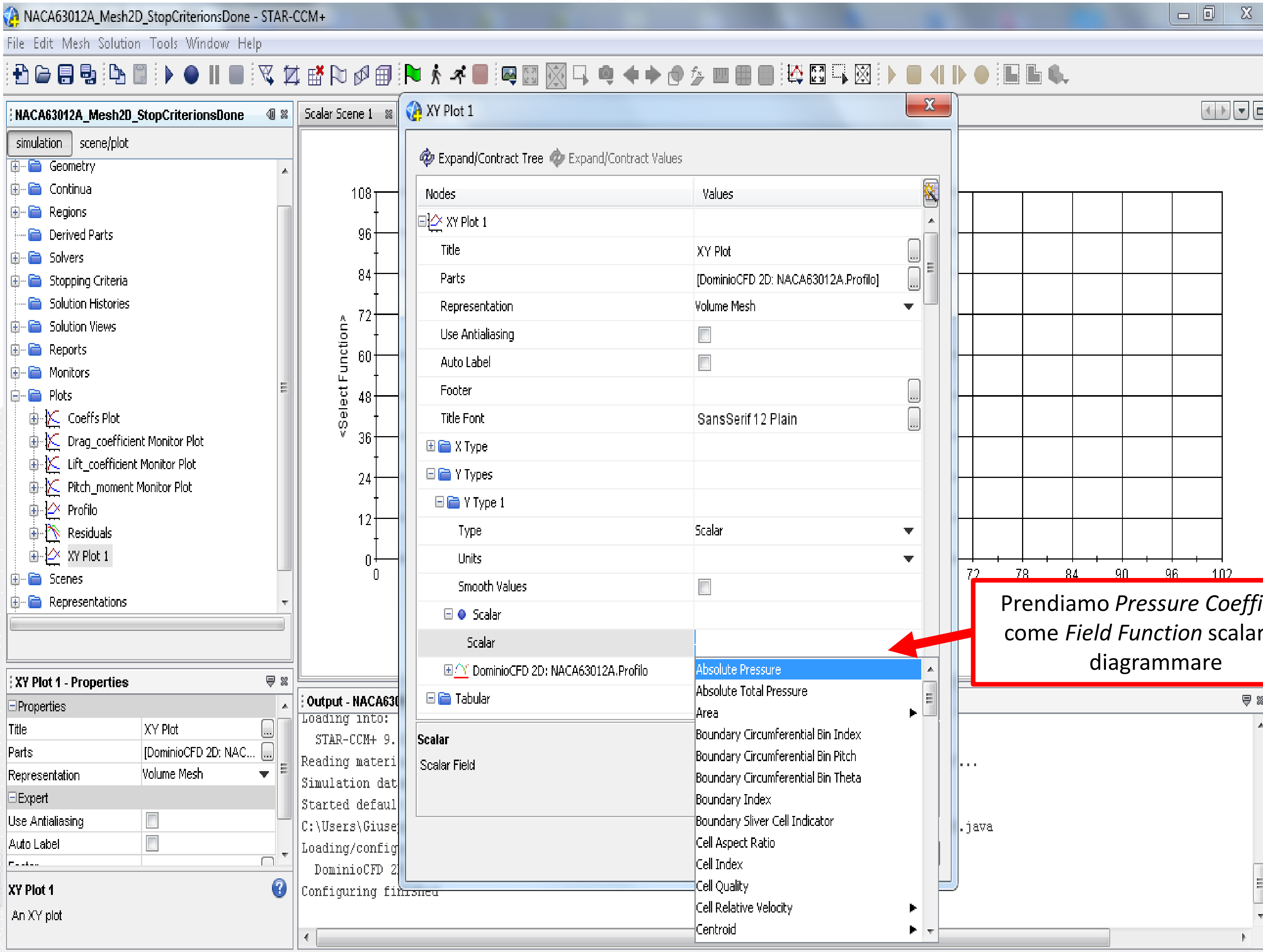
Creiamo un *Plot* di tipo X-Y per visualizzare il C_p sul profilo.

N.B.: Per come è stata generata la geometria, non possiamo separare i punti relativi al dorso e al ventre del profilo.

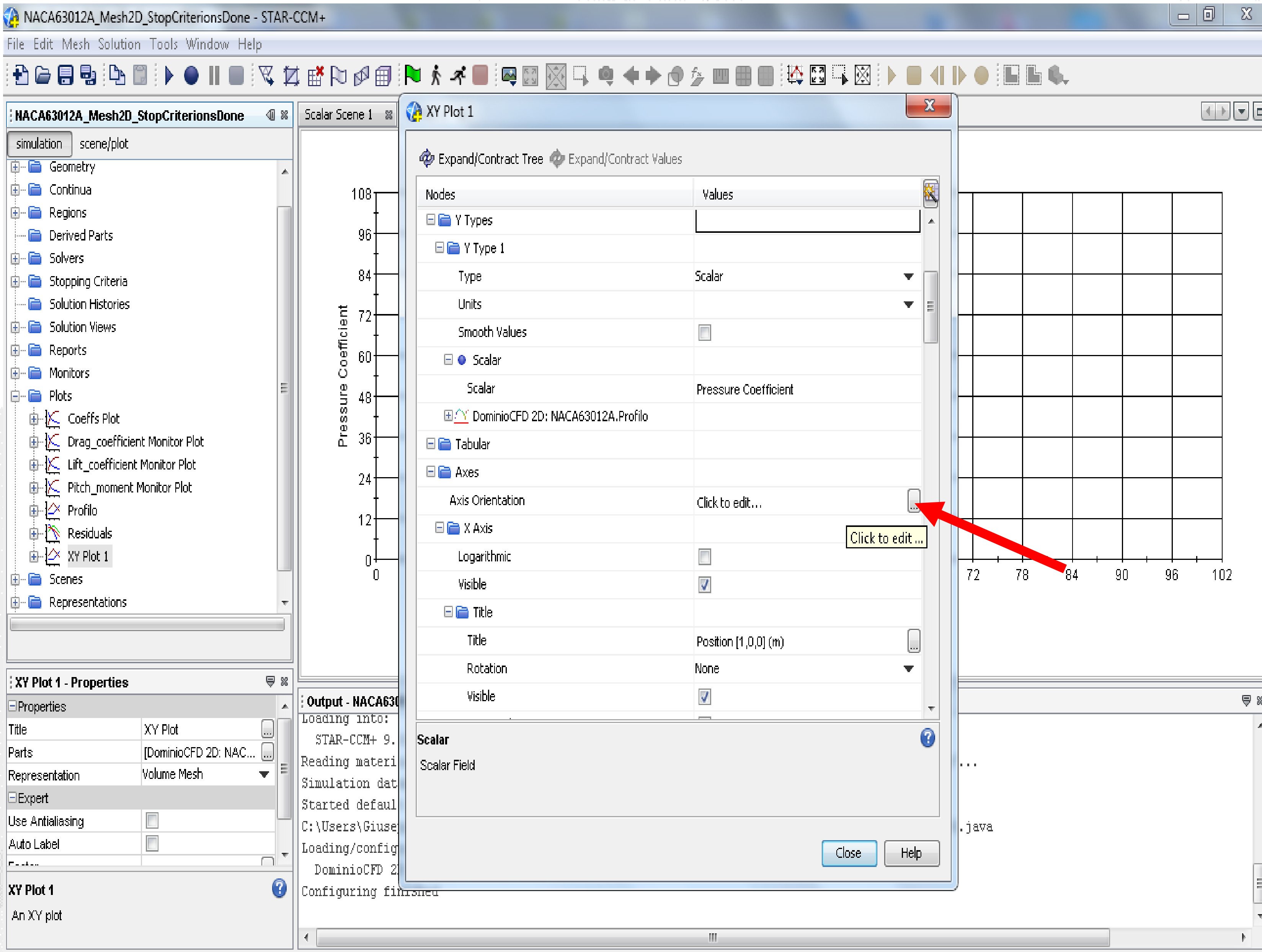


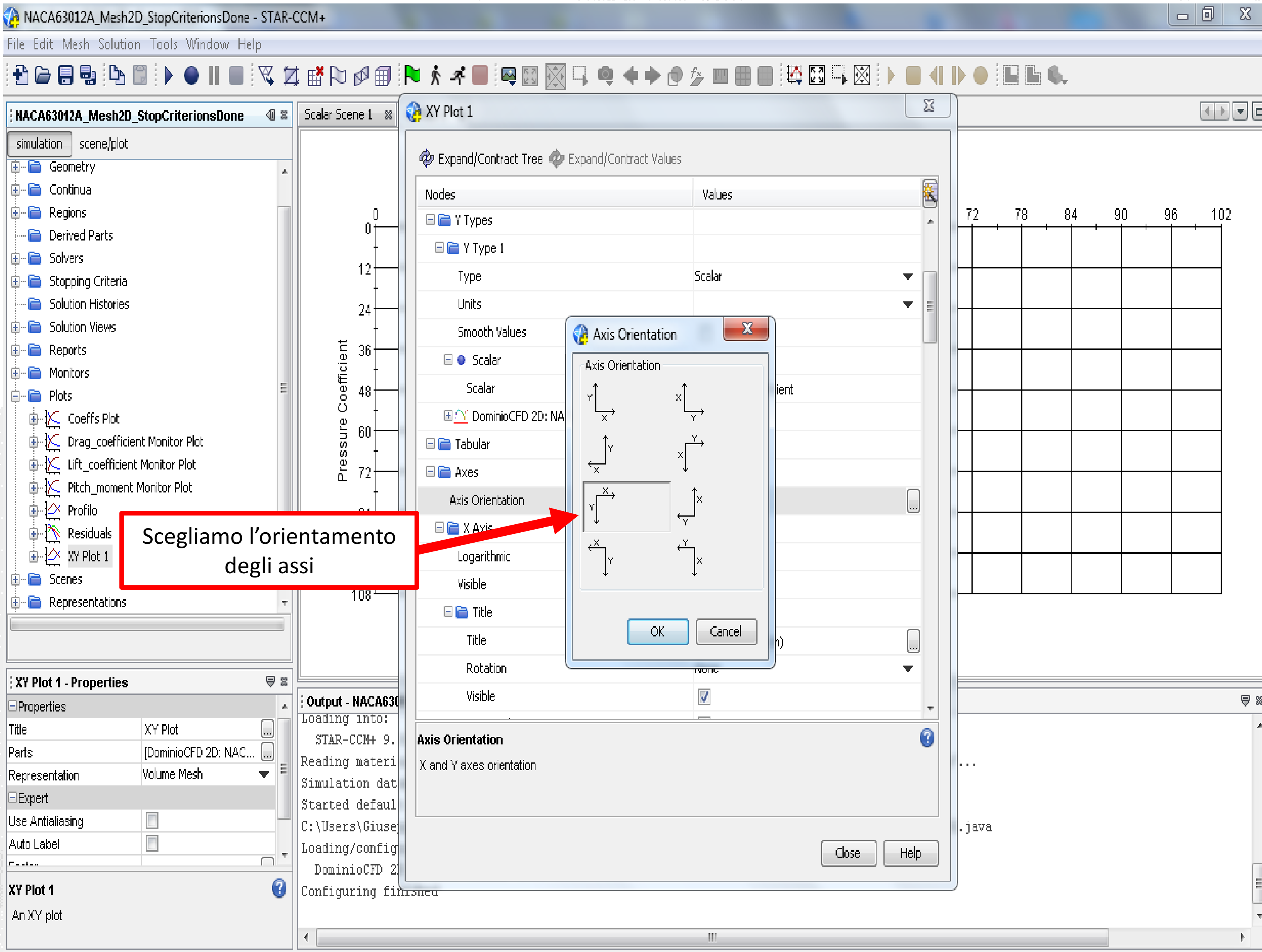
Decidiamo le *Parts* interessate dalle operazioni presenti nel *Plot*.

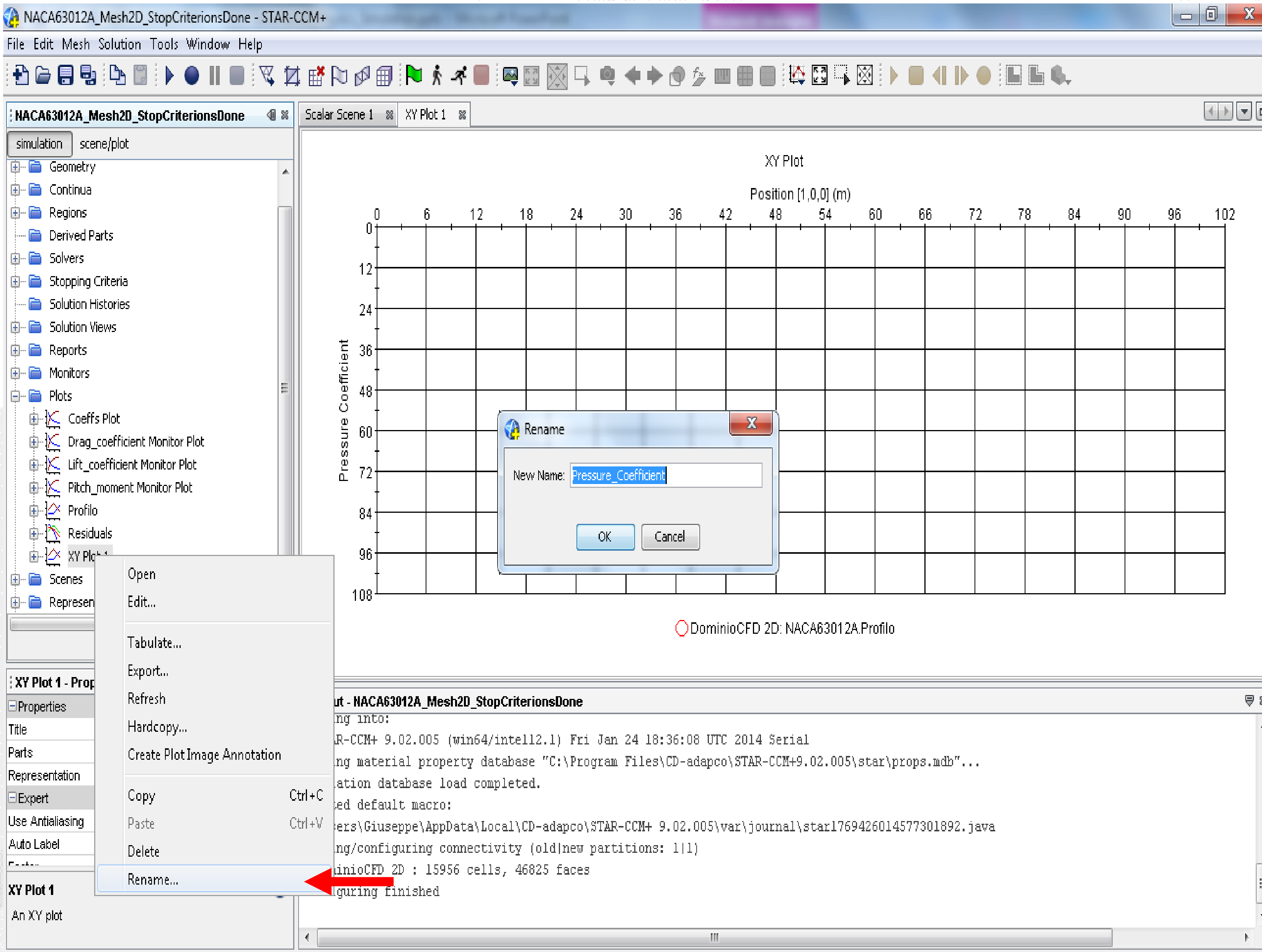

NACA63012A_Mesh2D_StopCriterionsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_StopCriterionsDone
simulation
scene/plot
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Profilo
Residuals
XY Plot 1
Scenes
Representations
XY Plot 1 - Properties
Properties
Title
XY Plot
Parts
[]
Representation
Volume Mesh
Expert
Use Antialiasing
Auto Label
XY Plot 1
An XY plot
Scalar Scene 1
XY Plot 1
Expand/Contract Tree
Expand/Contract Values
XY Plot 1 - Parts
Show All
Filter by Path
Parts
Regions
DominioCFD 2D
Boundaries
Dominio.Freestream
NACA63012A.Profilo
1 of 15 selected
OK
Cancel
Help
Close
Help
72
78
84
90
96
102
Outpu
Configuring finished

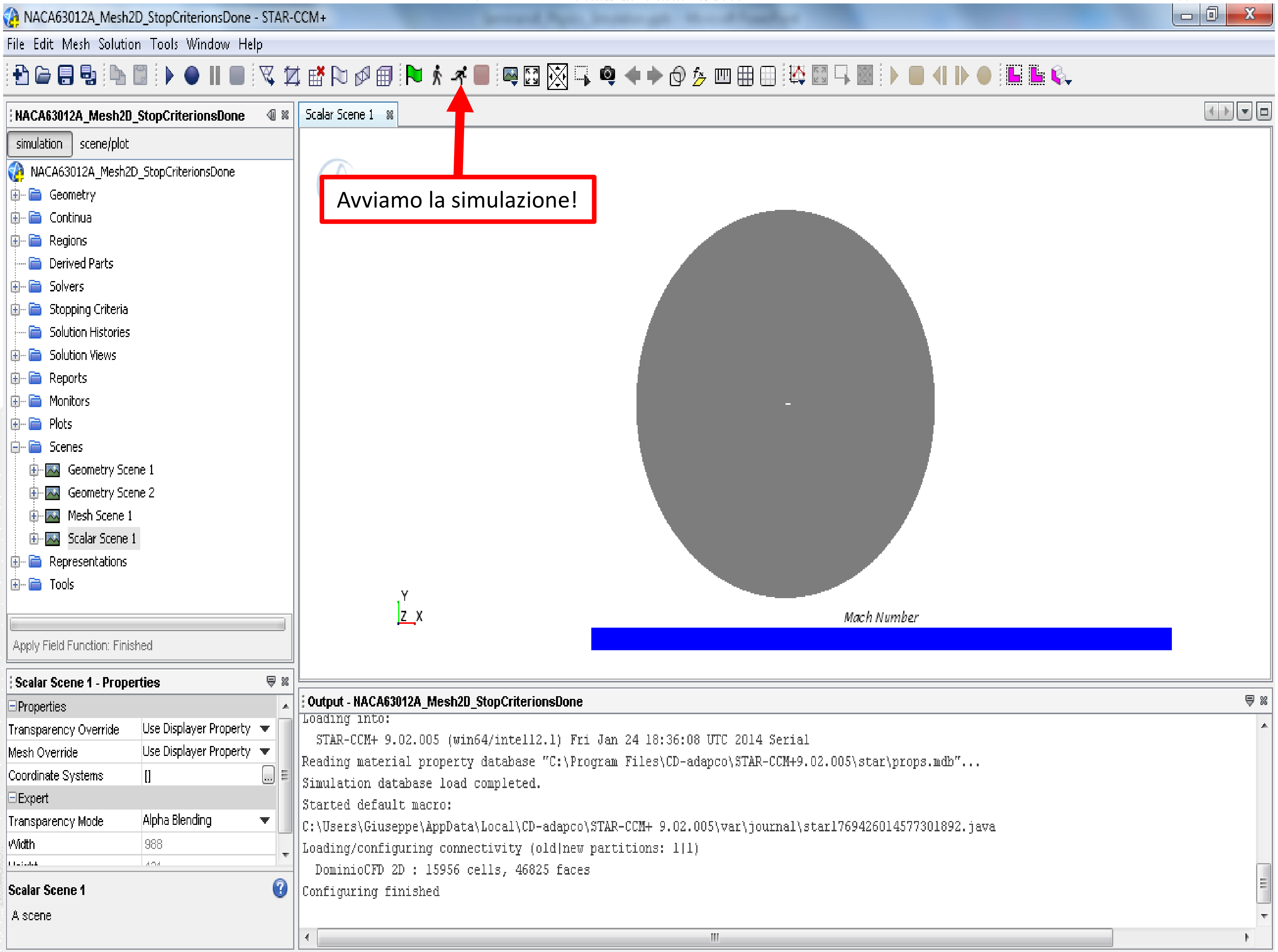


Prendiamo *Pressure Coefficient* come *Field Function* scalare da diagrammare


NACA63012A_Mesh2D_StopCriterionsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_StopCriterionsDone
simulation
scene/plot
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Profilo
Residuals
XY Plot 1
Scenes
Representations
XY Plot 1 - Properties
Properties
Title
XY Plot
Parts
[DominioCFD 2D: NAC...
Representation
Volume Mesh
Expert
Use Antialiasing
Auto Label
XY Plot 1
An XY plot
Scalar Scene 1
Pressure Coefficient
XY Plot 1
Expand/Contract Tree
Expand/Contract Values
Nodes
Values
Y Types
Y Type 1
Type
Scalar
Units
Smooth Values
Scalar
Scalar
Pressure Coefficient
DominioCFD 2D: NACA63012A.Profilo
Tabular
Axes
Axis Orientation
Click to edit...
X Axis
Logarithmic
Visible
Title
Title
Position [1,0,0] (m)
Rotation
None
Visible
Click to edit ...
Scalar
Scalar Field
Close
Help
Output - NACA630
Loading into:
STAR-CCM+ 9.
Reading materi
Simulation dat
Started defaul
C:\Users\Giuse
Loading/config
DominioCFD 2
Configuring finished
.java


NACA63012A_Mesh2D_StopCriterionsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_StopCriterionsDone
simulation
scene/plot
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Profilo
Residuals
XY Plot 1
Scenes
Representations
XY Plot 1 - Properties
Properties
Title
XY Plot
Parts
[DominioCFD 2D: NAC...
Representation
Volume Mesh
Expert
Use Antialiasing
Auto Label
XY Plot 1
An XY plot
Scalar Scene 1
Pressure Coefficient
XY Plot 1
Expand/Contract Tree
Expand/Contract Values
Nodes
Values
Y Types
Y Type 1
Type
Scalar
Units
Smooth Values
Scalar
Tabular
Axes
Axis Orientation
X Axis
Logarithmic
Visible
Title
Rotation
Axis Orientation
X and Y axes orientation
Close
Help
Axis Orientation
OK
Cancel
Output - NACA630
Loading into:
Reading materi
Simulation dat
Started defaul
Loading/config
Configuring finished
Scegliamo l'orientamento degli assi


NACA63012A_Mesh2D_StopCriterionsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_StopCriterionsDone
simulation
scene/plot
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Profilo
Residuals
Scenes
Scalar Scene 1
XY Plot 1
XY Plot
Position [1,0,0] (m)
Pressure Coefficient
DominioCFD 2D: NACA63012A.Profilo
Rename
New Name:
Pressure_Coefficient
OK
Cancel
Open
Edit...
Tabulate...
Export...
Refresh
Hardcopy...
Create Plot Image Annotation
Copy
Ctrl+C
Paste
Ctrl+V
Delete
Rename...
Properties
Title
Parts
Representation
Expert
Use Antialiasing
Auto Label
XY Plot 1
An XY plot
R-CCM+ 9.02.005 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Serial
ation database load completed.
iinioCFD 2D : 15956 cells, 46825 faces
iguring finished



NACA63012A_Mesh2D_StopCriterionsDone - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Avviamo la simulazione!
NACA63012A_Mesh2D_StopCriterionsDone
simulation
scene/plot
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Representations
Tools
Apply Field Function: Finished
Scalar Scene 1 - Properties
Properties
Transparency Override
Use Displayer Property
Mesh Override
Use Displayer Property
Coordinate Systems
[]
Expert
Transparency Mode
Alpha Blending
Width
988
Scalar Scene 1
A scene
Scalar Scene 1
Y
Z X
Mach Number
Output - NACA63012A_Mesh2D_StopCriterionsDone
Loading into:
STAR-CCM+ 9.02.005 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Serial
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Simulation database load completed.
Started default macro:
C:\Users\Giuseppe\AppData\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star1769426014577301892.java
Loading/configuring connectivity (old|new partitions: 1|1)
DominioCFD 2D : 15956 cells, 46825 faces
Configuring finished

Judging Convergence (pp. 6660)

(STAR-CCM+ User Guide)

Residual monitor plots are useful for judging the convergence (or divergence) of a solution, and they are created automatically within every simulation. However, it is important to understand both the significance of residuals and their limitations. While it is true that the residual quantity tends toward a small number when the solution is converged, the residual monitors cannot be relied on as the only measure of convergence.

The limitations of residuals are as follows:

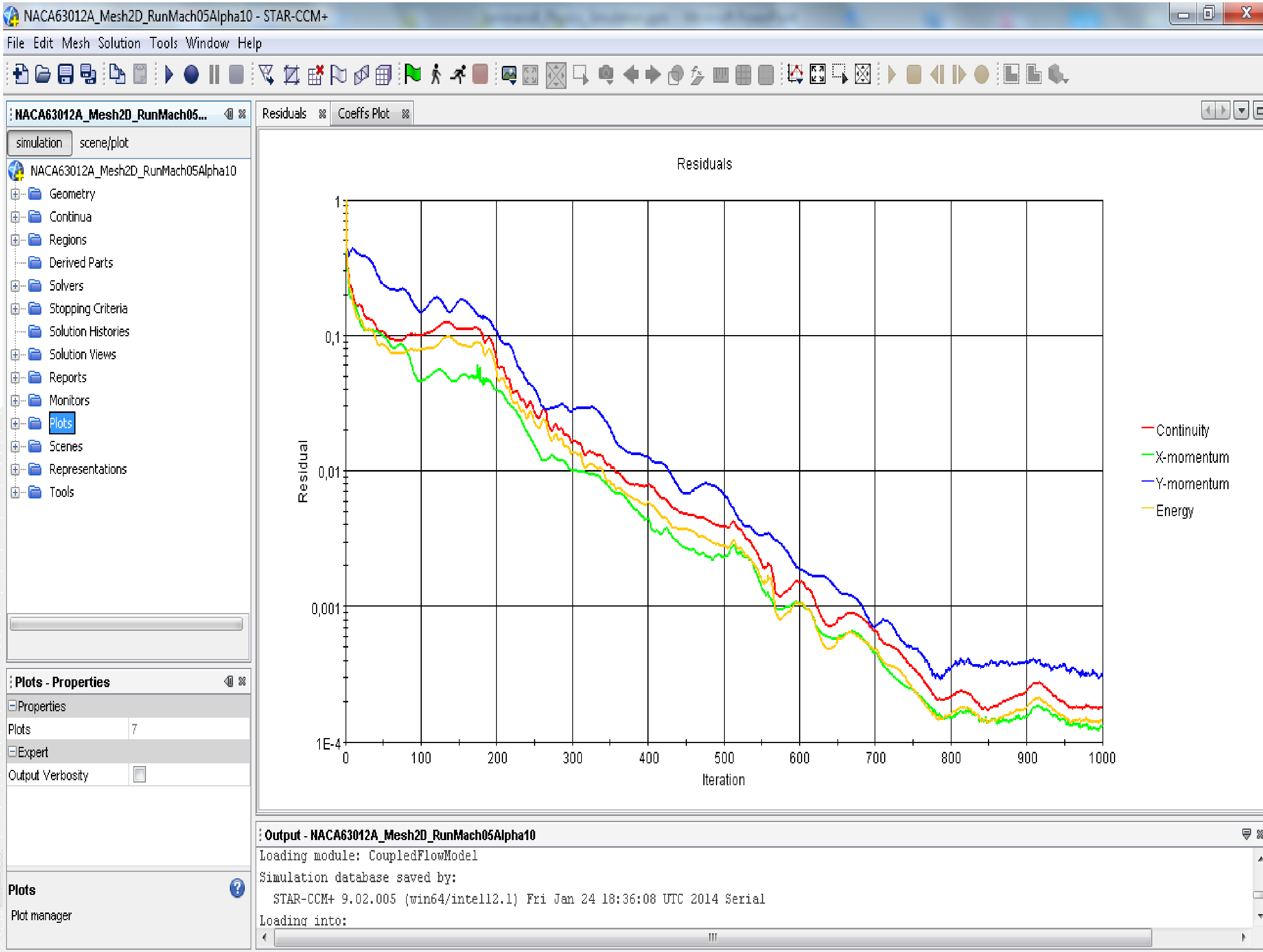
- The amount that a residual decreases by depends on the particulars of the simulation. Therefore, a three-order-of-magnitude drop in residuals is possibly acceptable for one simulation, but not another. The initial guess also strongly influences the amount that residuals are reduced. If the initial solution satisfies the discretized equations perfectly, the residuals do not drop at all.
- There are two types of discretization errors: dissipative errors and dispersive errors. Dissipative errors are characteristic of first-order upwind schemes; they are inherently stabilizing and produce residual plots that tend to decrease monotonically. Dispersive errors are characteristic of second-order upwind schemes which tend to “smear” solutions less than first-order schemes. While dispersive errors tend to produce residual plots that are not monotonic. This outcome is generally an acceptable price to pay for the enhanced accuracy. In some cases, often because of poor mesh quality, dispersive errors result in oscillating solutions (that is, changing from one iteration to the next) within a few cells. The result is that the residual plots can indicate that the solution is not “converged”. You have a choice to either accept the solution, or to try to stabilize it by choosing a lower-order numerical scheme. Frequently, it is better to accept the solution.
- Residuals do not necessarily relate to quantities of engineering interest in the simulation such as integrated forces, pressure losses, or mass flow rates.

With the issues above in mind, it is advisable to monitor quantities of engineering interest, such as integrated forces, pressure changes, or mass flow rates as well as the residuals. STAR-CCM+ features such as scenes and plots can help you examine these quantities while the solution progresses. The choice of the engineering quantity, as well as the convergence criterion, is your judgment call. Use your judgment and decide which coefficient is the most critical.

DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

SEZIONE INGEGNERIA AEROSPAZIALE

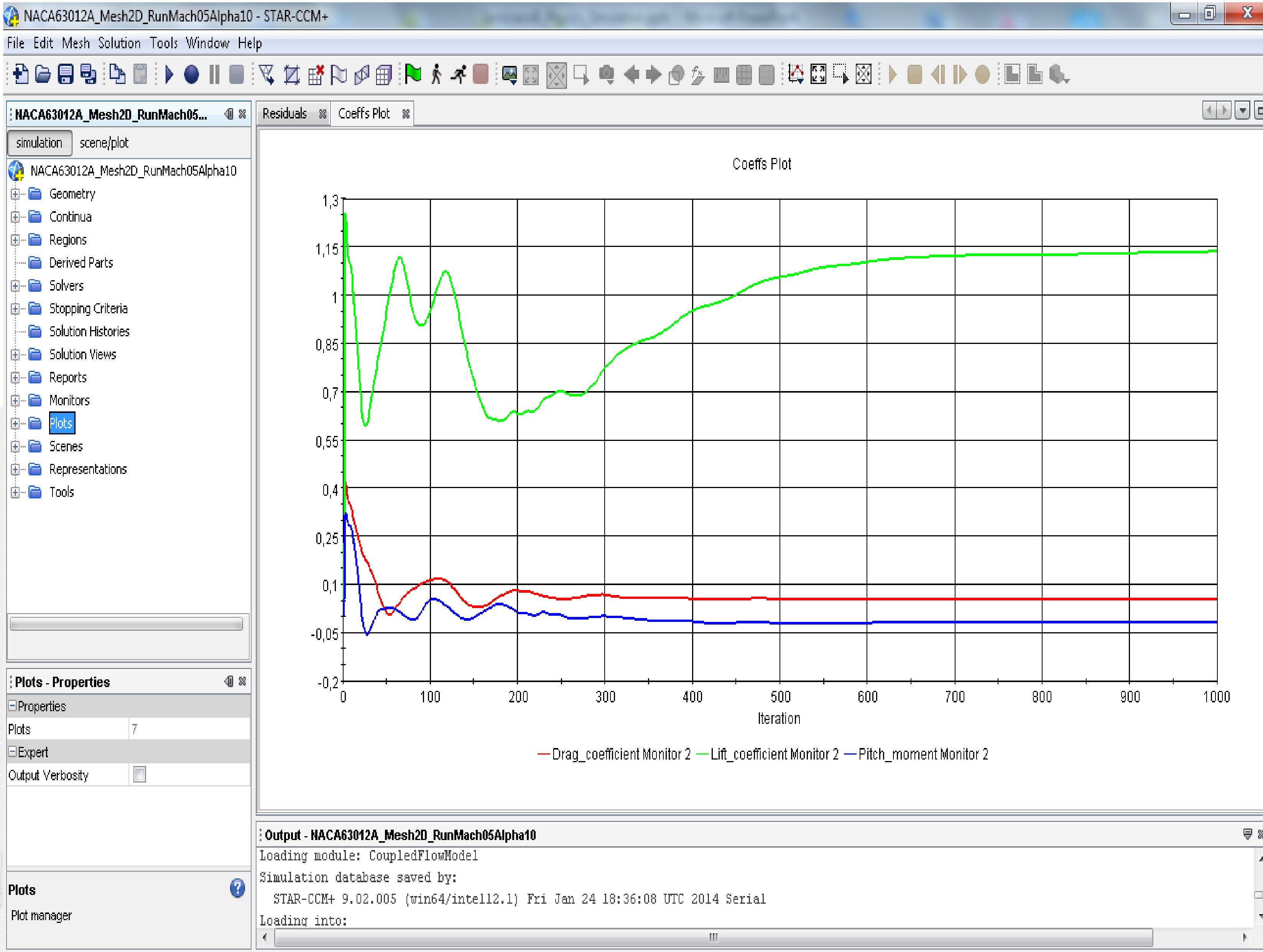
ADAG
Aircraft Design & AeroFlightDynamics Group

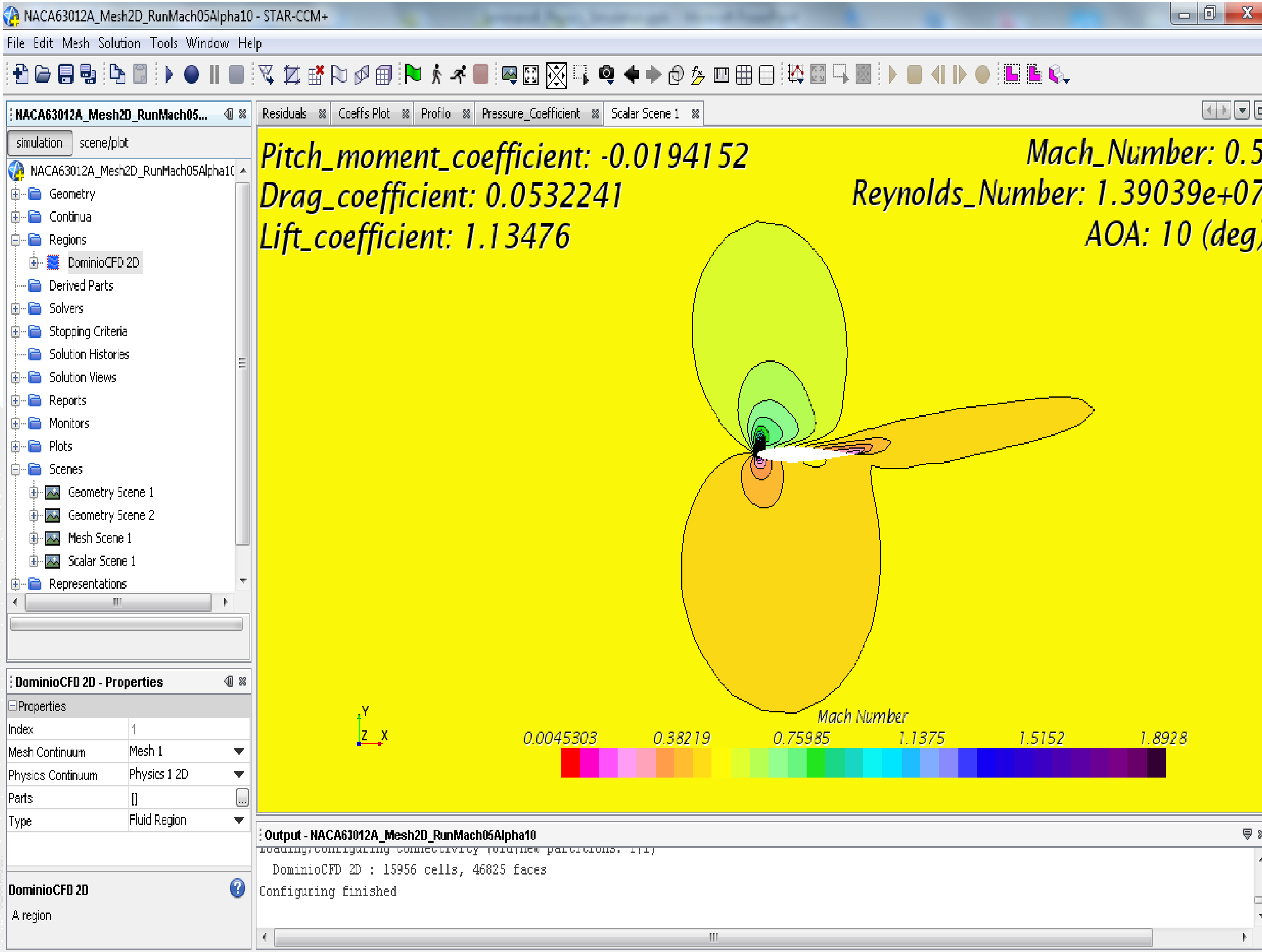


DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

SEZIONE INGEGNERIA AEROSPAZIALE

ADAG
Aircraft Design & AeroFlightDynamics Group

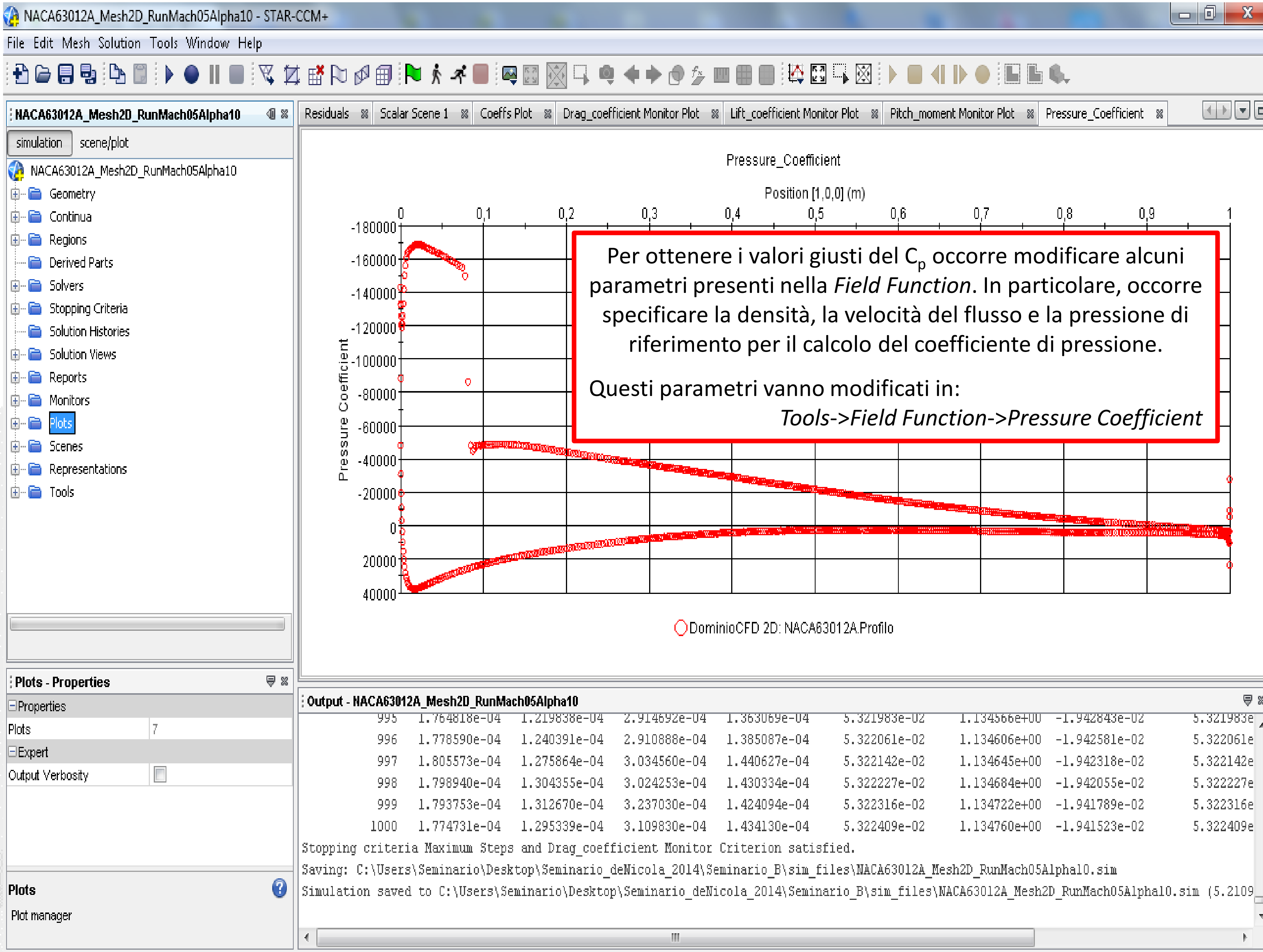



NACA63012A_Mesh2D_RunMach05Alpha10 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_RunMach05...
simulation
scene/plot
NACA63012A_Mesh2D_RunMach05Alpha10
Geometry
Continua
Regions
DominioCFD 2D
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Representations
DominioCFD 2D - Properties
Properties
Index 1
Mesh Continuum Mesh 1
Physics Continuum Physics 1 2D
Parts []
Type Fluid Region
DominioCFD 2D
A region
Residuals
Coeffs Plot
Profilo
Pressure_Coefficient
Scalar Scene 1
Pitch_moment_coefficient: -0.019452
Drag_coefficient: 0.0532241
Lift_coefficient: 1.13476
Mach_Number: 0.5
Reynolds_Number: 1.39039e+07
AOA: 10 (deg)
Mach Number
0.0045303
0.38219
0.75985
1.1375
1.5152
1.8928
Output - NACA63012A_Mesh2D_RunMach05Alpha10
DominioCFD 2D : 15956 cells, 46825 faces
Configuring finished

DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

SEZIONE INGEGNERIA AEROSPAZIALE

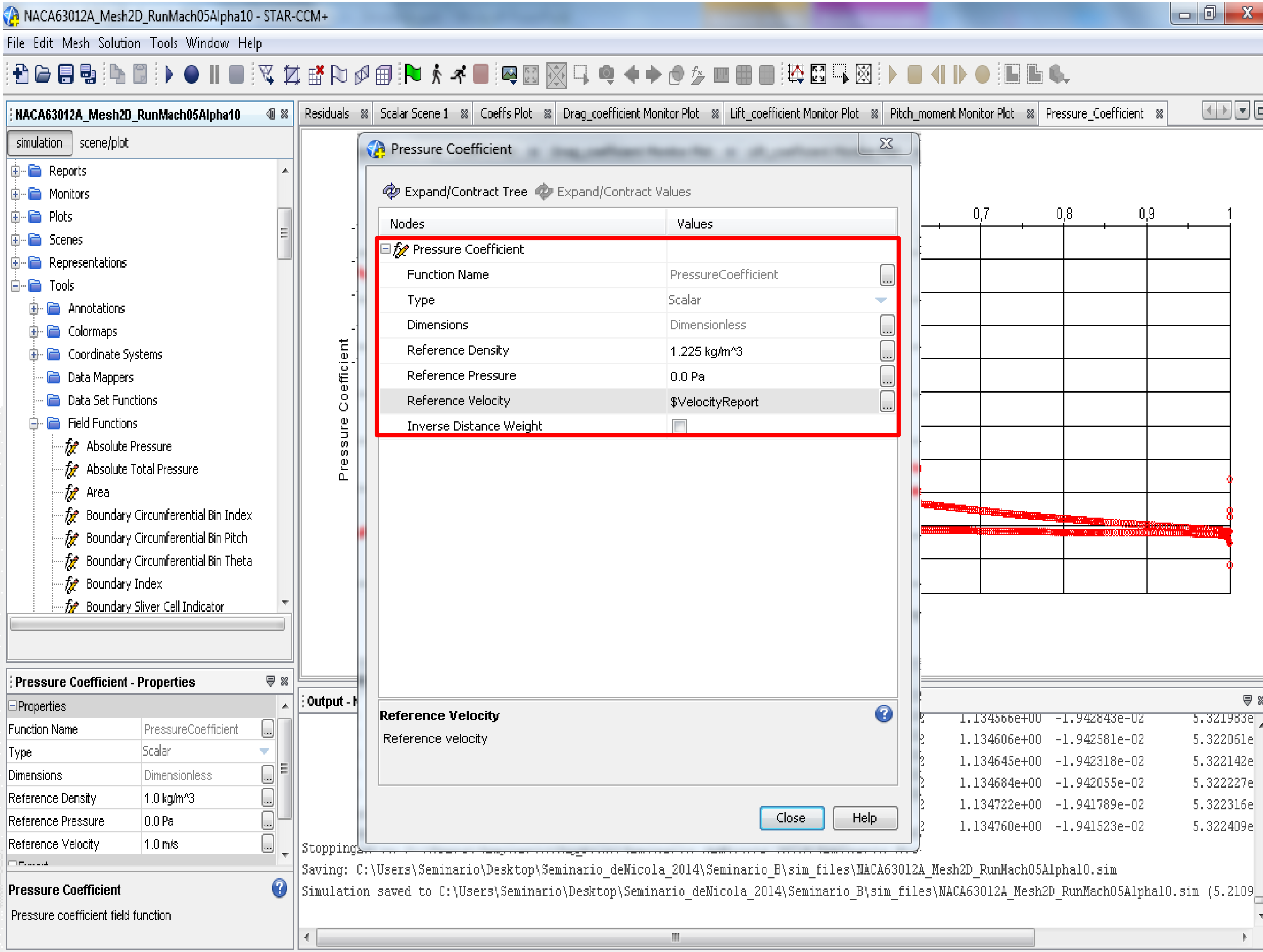
ADAG Aircraft Design & AeroFlightDynamics Group

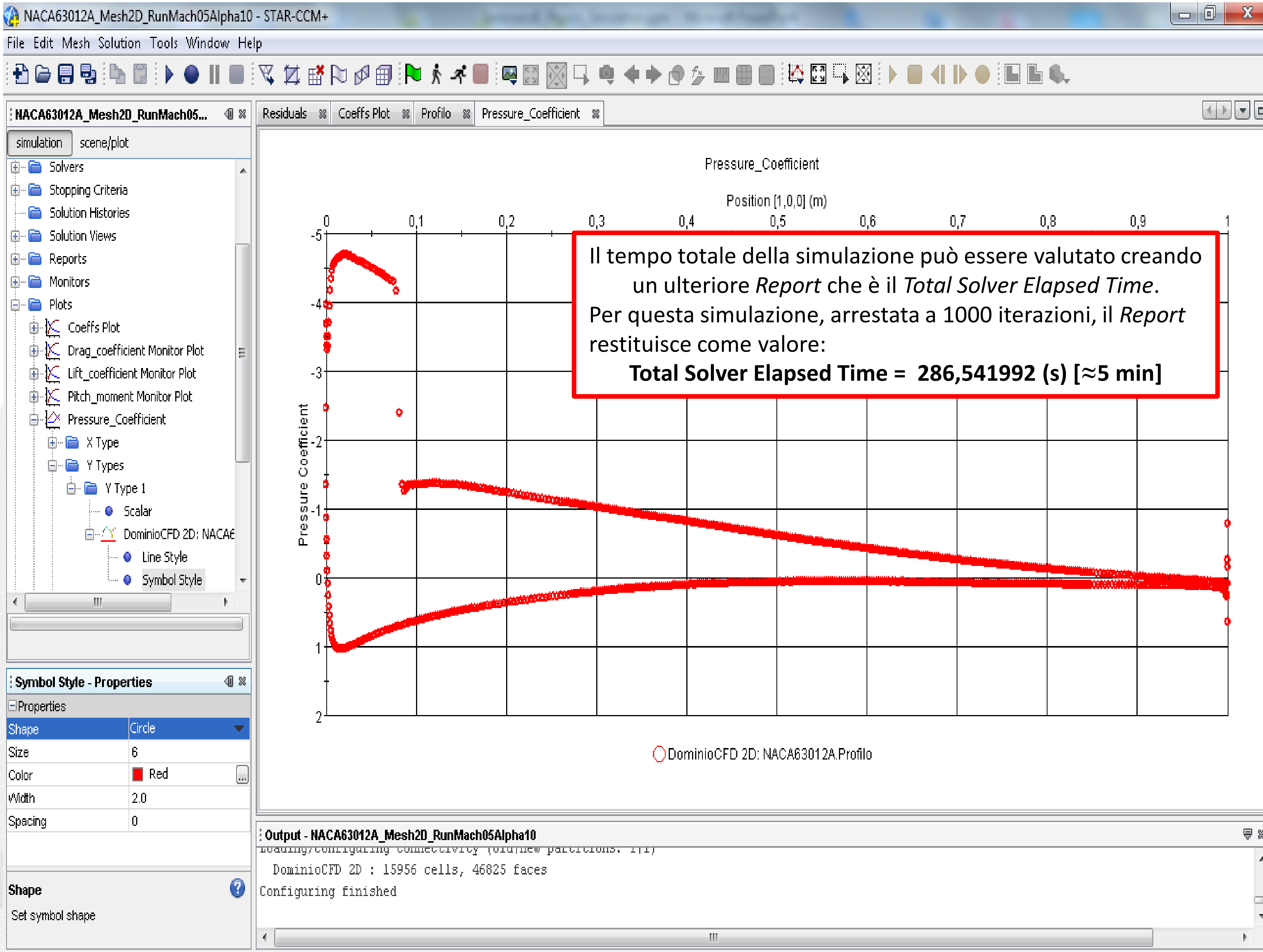


Per ottenere i valori giusti del C_p occorre modificare alcuni parametri presenti nella *Field Function*. In particolare, occorre specificare la densità, la velocità del flusso e la pressione di riferimento per il calcolo del coefficiente di pressione.

Questi parametri vanno modificati in:

Tools->Field Function->Pressure Coefficient

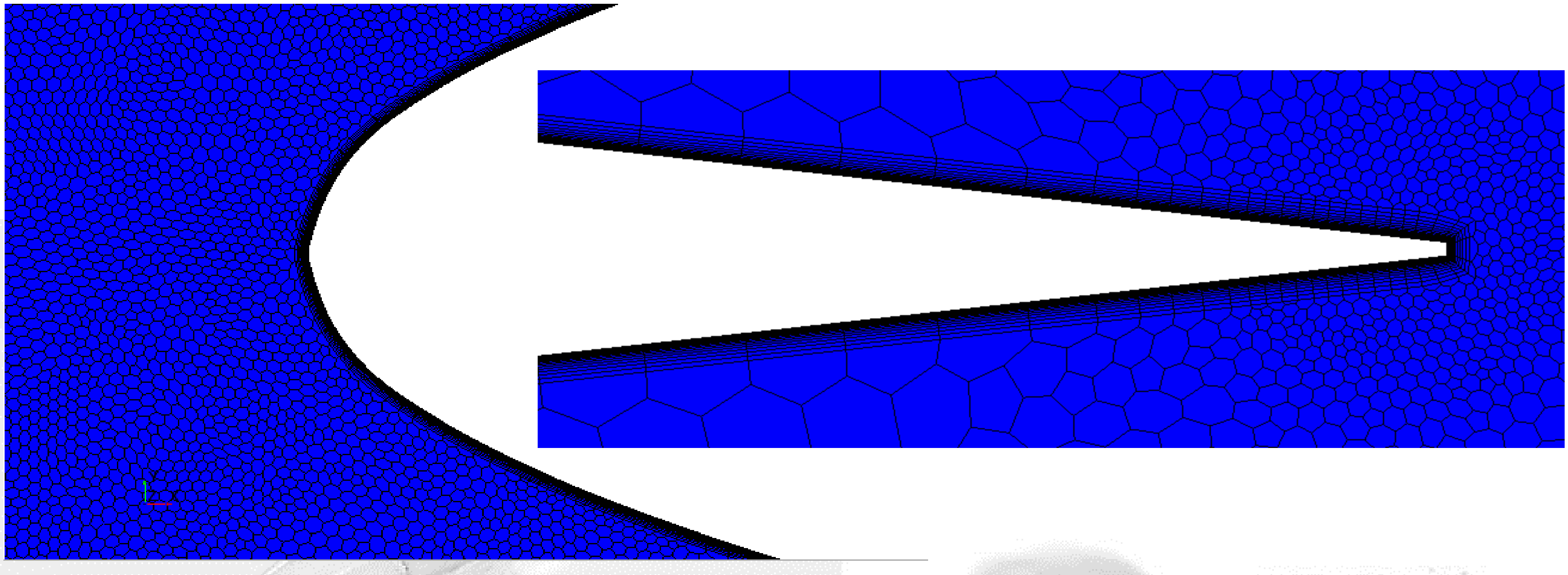

NACA63012A_Mesh2D_RunMach05Alpha10 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_RunMach05Alpha10
simulation
scene/plot
Reports
Monitors
Plots
Scenes
Representations
Tools
Annotations
Colormaps
Coordinate Systems
Data Mappers
Data Set Functions
Field Functions
Absolute Pressure
Absolute Total Pressure
Area
Boundary Circumferential Bin Index
Boundary Circumferential Bin Pitch
Boundary Circumferential Bin Theta
Boundary Index
Boundary Sliver Cell Indicator
Residuals
Scalar Scene 1
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Pressure_Coefficient
Pressure Coefficient
Expand/Contract Tree
Expand/Contract Values
Nodes
Values
Pressure Coefficient
Function Name
PressureCoefficient
Type
Scalar
Dimensions
Dimensionless
Reference Density
1.225 kg/m^3
Reference Pressure
0.0 Pa
Reference Velocity
$VelocityReport
Inverse Distance Weight
Reference Velocity
Reference velocity
Close
Help
Pressure Coefficient
0,7
0,8
0,9
1
Pressure Coefficient - Properties
Properties
Function Name
PressureCoefficient
Type
Scalar
Dimensions
Dimensionless
Reference Density
1.0 kg/m^3
Reference Pressure
0.0 Pa
Reference Velocity
1.0 m/s
Pressure Coefficient
Pressure coefficient field function
Output
1.134566e+00 -1.942843e-02 5.321983e
1.134606e+00 -1.942581e-02 5.322061e
1.134645e+00 -1.942318e-02 5.322142e
1.134684e+00 -1.942055e-02 5.322227e
1.134722e+00 -1.941789e-02 5.322316e
1.134760e+00 -1.941523e-02 5.322409e
Saving: C:\Users\Seminario\Desktop\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_RunMach05Alpha10.sim
Simulation saved to C:\Users\Seminario\Desktop\Seminario_deNicola_2014\Seminario_B\sim_files\NACA63012A_Mesh2D_RunMach05Alpha10.sim (5.2109



Il tempo totale della simulazione può essere valutato creando un ulteriore *Report* che è il *Total Solver Elapsed Time*.
Per questa simulazione, arrestata a 1000 iterazioni, il *Report* restituisce come valore:

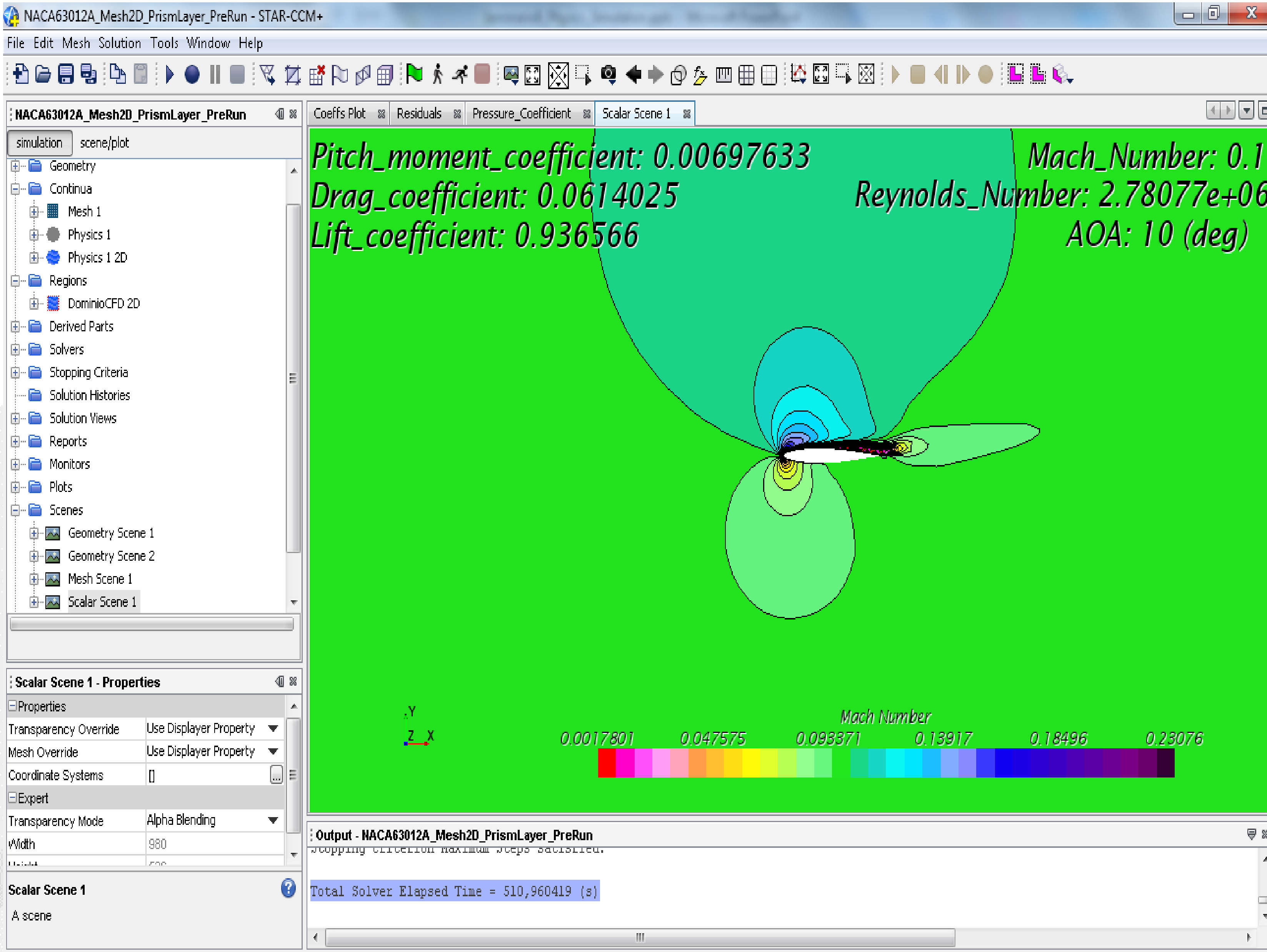
Total Solver Elapsed Time = 286,541992 (s) [≈5 min]

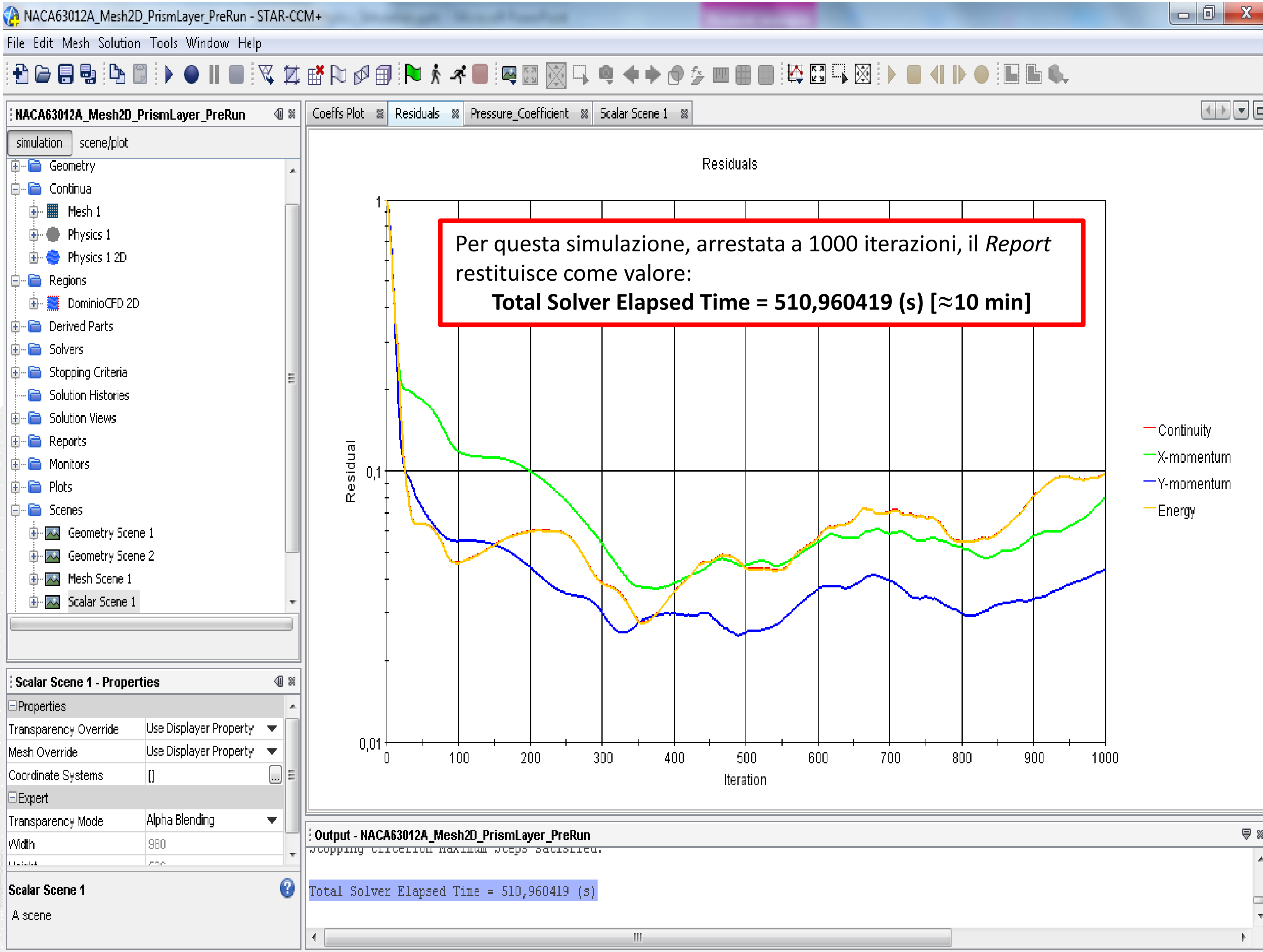
Impostazione della fisica: soluzione non viscosa, basso subsonico (M_∞=0.1 , α=10 °)



Per questa applicazione, utilizzeremo la seconda griglia generata (quella con il *Prism Layer*) e valuteremo sia l'incremento in termini di tempi di calcolo, sia l'eventuale variazione dei coefficienti aerodinamici.

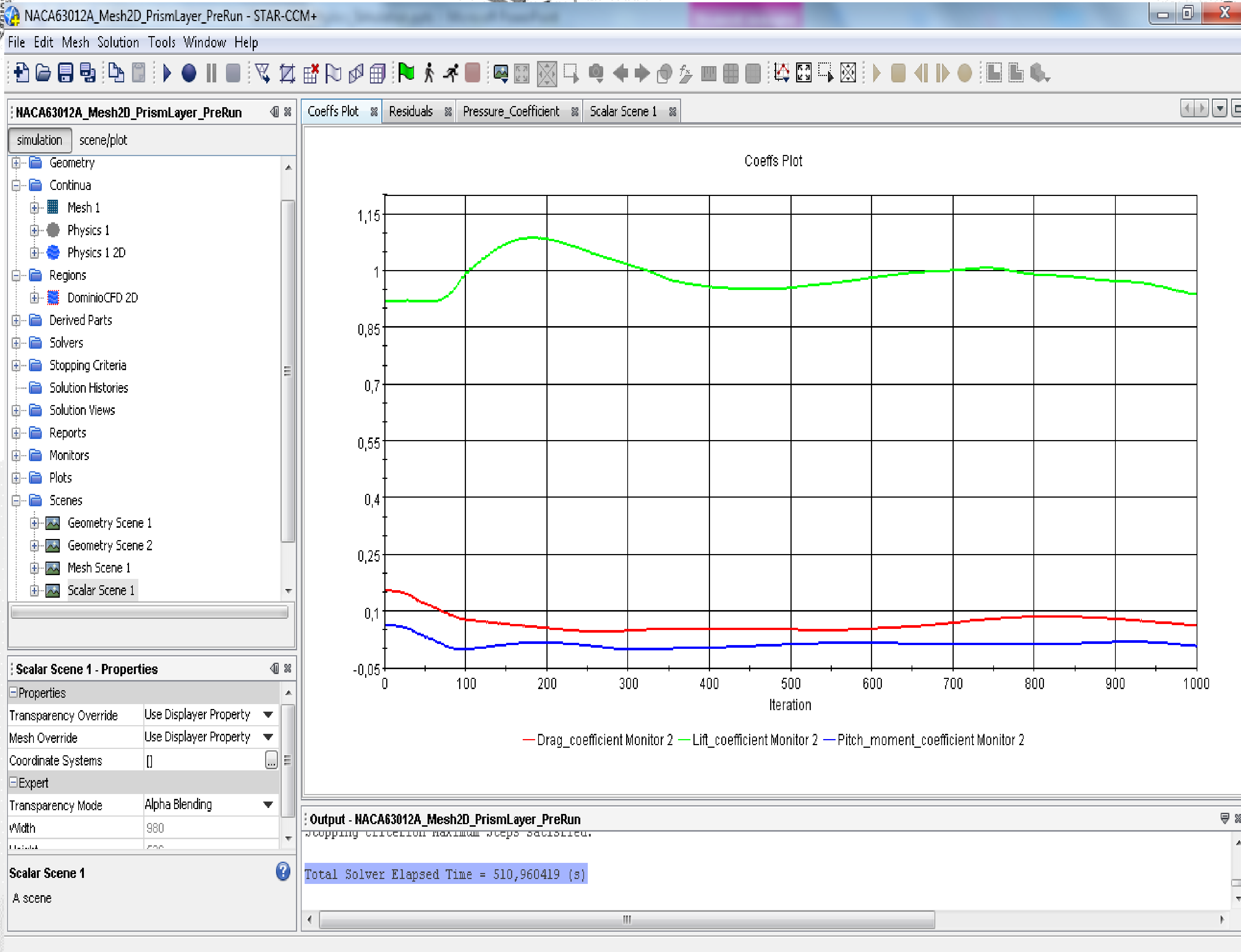
N.B.: Nel file di simulazione relativo alla nuova griglia vanno eseguite esattamente tutte le operazioni svolte finora

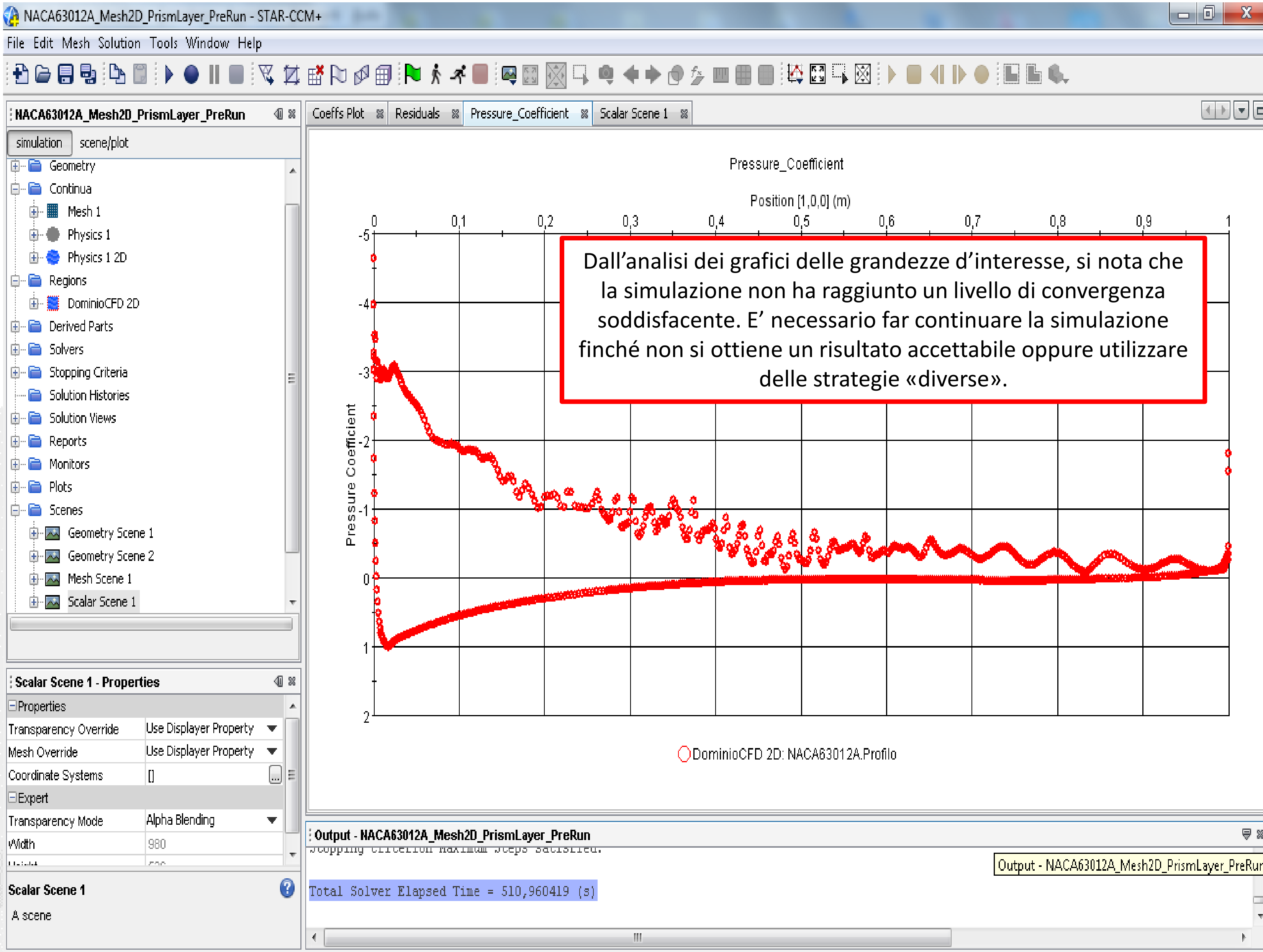

NACA63012A_Mesh2D_PrismLayer_PreRun - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Coeffs Plot
Residuals
Pressure_Coefficient
Scalar Scene 1
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
DominioCFD 2D
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Pitch_moment_coefficient: 0.00697633
Drag_coefficient: 0.0614025
Lift_coefficient: 0.936566
Mach_Number: 0.1
Reynolds_Number: 2.78077e+06
AOA: 10 (deg)
Mach Number
0.0017801
0.047575
0.093371
0.13917
0.18496
0.23076
Scalar Scene 1 - Properties
Transparency Override
Use Displayer Property
Mesh Override
Use Displayer Property
Coordinate Systems
Transparency Mode
Alpha Blending
Width
980
A scene
Output - NACA63012A_Mesh2D_PrismLayer_PreRun
Total Solver Elapsed Time = 510,960419 (s)



Per questa simulazione, arrestata a 1000 iterazioni, il *Report* restituisce come valore:

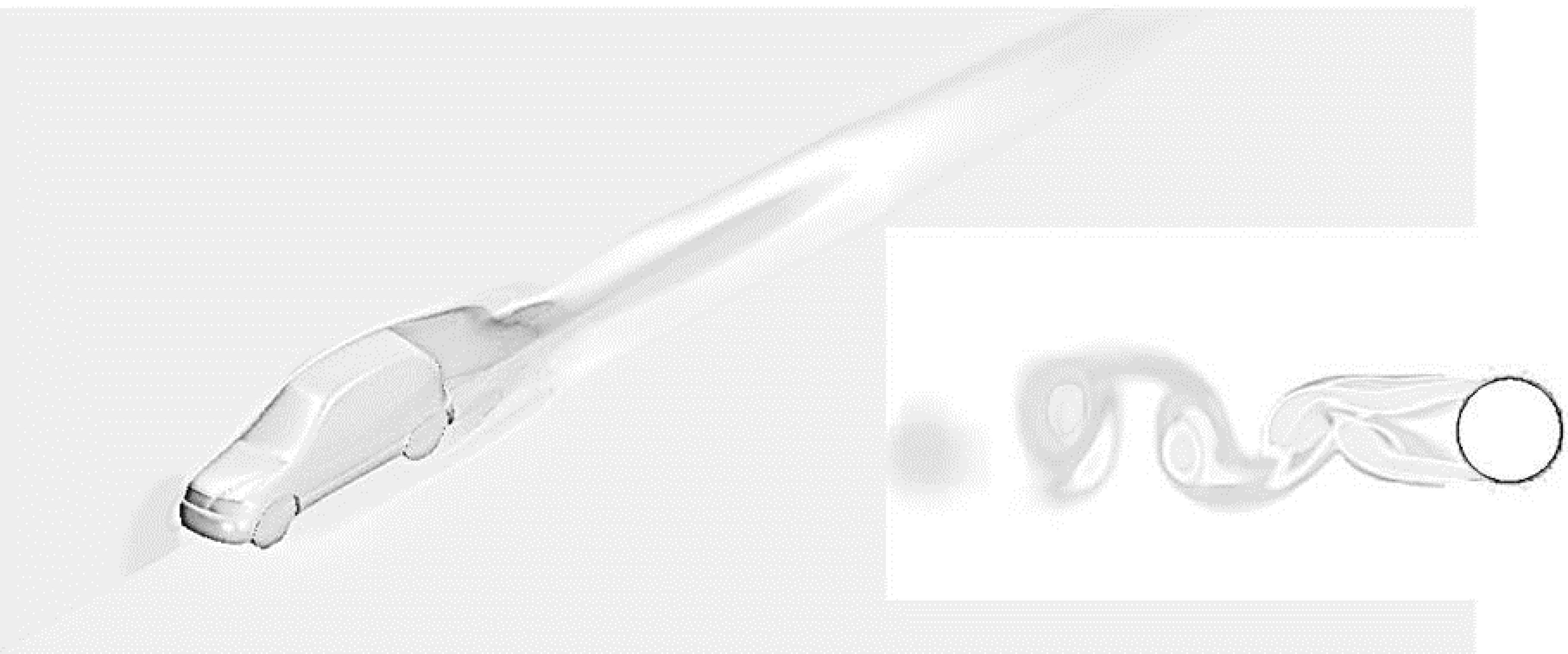
Total Solver Elapsed Time = 510,960419 (s) [≈10 min]


NACA63012A_Mesh2D_PrismLayer_PreRun - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PrismLayer_PreRun
simulation
scene/plot
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
DominioCFD 2D
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Scalar Scene 1 - Properties
Properties
Transparency Override
Use Displayer Property
Mesh Override
Use Displayer Property
Coordinate Systems
[]
Expert
Transparency Mode
Alpha Blending
Width
980
Scalar Scene 1
A scene
Coeffs Plot
Residuals
Pressure_Coefficient
Scalar Scene 1
Coeffs Plot
1,15
1
0,85
0,7
0,55
0,4
0,25
0,1
-0,05
0
100
200
300
400
500
600
700
800
900
1000
Iteration
Drag_coefficient Monitor 2
Lift_coefficient Monitor 2
Pitch_moment_coefficient Monitor 2
Output - NACA63012A_Mesh2D_PrismLayer_PreRun
Total Solver Elapsed Time = 510,960419 (s)



Dall'analisi dei grafici delle grandezze d'interesse, si nota che la simulazione non ha raggiunto un livello di convergenza soddisfacente. E' necessario far continuare la simulazione finché non si ottiene un risultato accettabile oppure utilizzare delle strategie «diverse».

Cosa fare se il calcolo non converge?



Analisi del profilo NACA 63012A con il solutore STAR-CCM+

Convergence vs. Mesh Indipendence

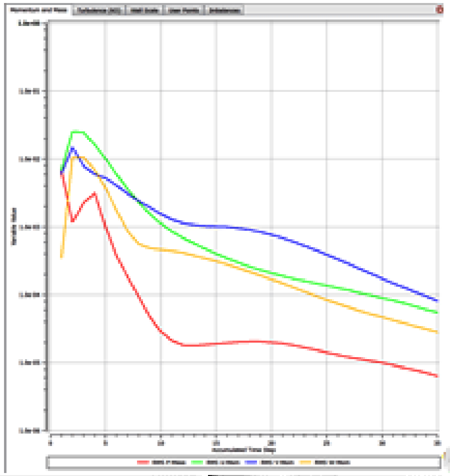
(http://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/)

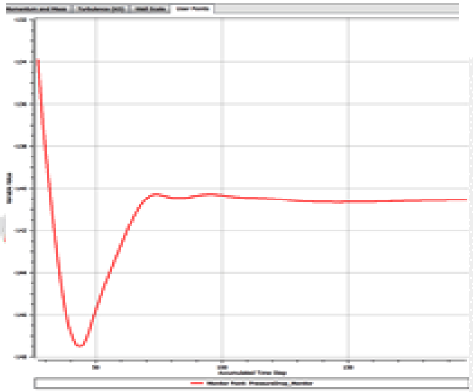
It is important to remember that your solution is the numerical solution to the problem that you posed by defining your mesh and boundary conditions. The more accurate your mesh and boundary conditions, the more accurate your "converged" solution will be.

CONVERGENCE

Convergence is something that all CFD Engineers talk about, but we must remember that **the way we generally define convergence (by looking at Residual values) is only a small part of ensuring that we have a valid solution.** For a Steady State simulation we need to ensure that the solution satisfies the following three conditions:

- Residual RMS Error values have reduced to an acceptable value (typically 10^{-4} or 10^{-5})
- Monitor points for our values of interest have reached a steady solution
- The domain has imbalances of less than 1%.





MESH INDEPENDENCE STUDY

Although we are happy that this has "converged" based on RMS Error values, monitor points and imbalances, we need to make sure that the solution is also independent of the mesh resolution. Not checking this is a common cause of erroneous results in CFD, and this process should at least be carried out once for each type of problem that you deal with so that the next time a similar problem arises, you can apply the same mesh sizing. In this way you will have more confidence in your results.

- Step 1

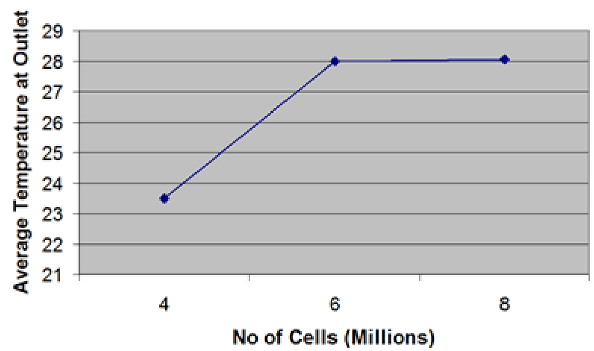
Run the initial simulation on your initial mesh and ensure convergence; if not, refine the mesh and repeat.

- Step 2

Once you have met the convergence criteria, refine the mesh globally so that you have finer cells throughout the domain. Run the simulation and ensure convergence again. Then, compare the monitor point values: if they are the same (within your own allowable tolerance), then the mesh at Step 1 was accurate enough to capture the result; otherwise, this means that your solution is changing because of your mesh resolution, and hence the solution is not yet independent of the mesh.

- Step 3

Because your solution is changing with the refinements, you need to refine the mesh more, and repeat the process until you have a solution that is independent of the mesh. You should then always use the smallest mesh that gives you this mesh independent solution (to reduce your simulation run time).



Troubleshooting the Solution (pp. 6752)

(STAR-CCM+ User Guide)

Divergence of the solution is the situation where cell residuals increase from one iteration to another. If left, it can eventually result in a floating point exception being reported. Generally, the behavior of the residual monitor plots indicates when divergence happens. An increase in one or more of the residual monitors during the first few iterations is normal and acceptable.

For example, consider that a (constant) initial solution often satisfies the discretized equations exactly, everywhere except at the boundaries. The boundary conditions propagate into the interior as the solution iterates, causing the global residuals to increase for several iterations before decreasing. However, if this situation does not change after 5-10 iterations, it is likely to be an indication of divergence.

Solution divergence could be due to one of two reasons:

- In a case with difficult physics, the initial conditions specify that the initial solution is inappropriate and special steps must be taken to get past a poor initial solution; or
- There is some other fundamental issue with the simulation, such as the mesh quality, the boundary conditions or the solution parameters, and special steps must be taken to debug the simulation setup.

If divergence occurs, you do not need to exit from the simulation. Instead, you can clear the solution and run it again after investigating the cause of divergence and making the required changes to the simulation.

Getting Past a Poor Initial Solution

As mentioned in the previous section dealing with solution divergence, a possible cause of divergence can be related to the initial solution being radically different from a converged solution. This situation occurs because implicit numerical methods rely on linearization techniques to advance the solution of equations toward convergence, and it is possible that the linearization is inadequate to advance the solution.

There are basically two techniques to deal with this issue:

- Improve the initial conditions. Alternatively, it can be achieved using techniques such as gradually activating models, changing boundary conditions, activating unsteady models from a steady solution, or ramping up solution accuracy.
- Advance the solution less by ramping up Courant numbers and relaxation factors.

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

When using these techniques, be sure to follow the guidelines which tell you if you are required to stop the solution or not before you change the conditions. These guidelines are described in the section that deals with changing the simulation while running.

Gradually Activating Models

There is no need to activate all the models at the beginning of the solution; models can be activated during the solution.

Examples of where it can be beneficial to activate models during the solution process are as follows:

- Compressible flows can be started inviscid (or laminar) before turning on turbulence. However, take care to ensure that the meshes are appropriate for the substituted models. It is possible that the Courant numbers and relaxation factors need ramped up after the models are changed.
- Unsteady models can be activated from a steady solution.

Changing Boundary Conditions or Ramping Boundary Values

In some simulations, it can be difficult to start the solution with the final boundary conditions and boundary values.

Possible examples of where it is advantageous to start with one set of boundary conditions or boundary values before changing them are as follows:

- When solving high-Mach number flows over bluff bodies, it can be beneficial to ramp-up the free-stream Mach number gradually.
- In some complex turbomachinery cases that use a combination of stagnation and pressure boundaries, establishing the correct flow direction at the inlet boundary can be difficult. It is therefore prudent to obtain an initial solution with another sort of inlet boundary (mass flow or velocity inlet) before switching to a stagnation inlet.

When changing the boundary type, take care to ensure that the types are compatible.

For example, exercise caution in the following scenarios:

- For a highly compressible flow, it is possible that a velocity inlet is not a well-posed type.
- In incompressible flows, exercise caution when changing a flow split outlet to a pressure outlet. The pressure at a flow split outlet is not specified and is potentially considerably different from the pressure that is specified at a pressure outlet.

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

Activating Unsteady Models from a Steady Solution

It is common practice to start an unsteady simulation with a steady solution.

Potentially advantageous examples of this practice are:

• When doing a simulation of vortex shedding from a bluff body, you can often first obtain a steady-state solution, though there is no guarantee that the solution you obtain is converged. In some cases, if a converged steady-state solution exists, it is possible that a change in inlet boundary conditions for a few time-steps is necessary. This action can cause a perturbation and prompt the shedding to occur.

• In some turbomachinery problems where a transient, sliding mesh calculation is required, you can often start with a steady solution using the multiple-reference frame (MRF) approach. Then you can initiate the unsteady and mesh motion models.

Ramping up Solution Accuracy

As discussed in the section on judging convergence, the dissipative errors resulting from first-order solutions are stabilizing and can enhance convergence. Therefore, it can be beneficial to start the solution with a first-order numerical scheme.

For example:

• Models involving convective transport equations have either a Convection property (for segregated models), or a Discretization property (for the Coupled Flow model), that can be set to first order. The Convection property for segregated models affects only the upwind scheme for convection, leaving the diffusion and other gradients still fully second-order. The Discretization property for the Coupled Flow model neglects higher-order reconstruction for convection and diffusion.

• Solvers that are related to models that involve convective transport equations have a Reconstruction Zeroed expert property. Irrespective of the model type, this property causes reconstruction gradients to be neglected, effectively resulting in a first-order scheme.

Ramping up Courant Numbers and Relaxation Factors

The default values of Courant number and the under-relaxation factors in STAR-CCM+ work in many situations, but are unlikely to be optimal in all situations, particularly when dealing with poor initial solutions. Suggestions for suitable alternative values are contained in the troubleshooting sections of the following relevant modeling guides:

- Coupled Flow and Coupled Energy
- Segregated Flow
- Segregated Fluid Energy
- Turbulence

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

Coupled Flow Controls and General Setup Recommendations (pp. 2811)

The STAR-CCM+ Coupled Flow model and Coupled Implicit Solver make available several parameters which allow control of stability and convergence for practical CFD cases:

- The Courant Number (CFL) property of the Coupled Implicit and Coupled Explicit solvers controls the size of the local time-steps that are used in the time-marching procedure these two solvers employ. The Courant number plays the same role as under-relaxation parameters in the segregated solver.
  Generally speaking, for a steady-state simulation, a larger CFL number increases the local pseudo-time step size and produces faster convergence. Thus, use the largest CFL number possible while still ensuring that the solver remains within the bounds of stability. (This means that the solver does not start reducing corrections, or hitting temperature and pressure limits, and that the residual has a healthy downward trend.)
- The Explicit Relaxation (an expert property of the Coupled Flow Model) is a scaling factor that is used to relax all corrections explicitly before they are applied to the variables. It is intended to add stability in cases where the solver's first-order linearization of a second-order discretization has trouble. The default value is 1.0, which results in 100% of the corrections being applied. Use a value less than 1.0 to limit the applied corrections explicitly.
- The Positivity Rate Limit (an expert property of the Coupled Flow Model) defines the amount that temperature, a positive definite quantity, is allowed to decrease when corrections are applied at each iteration. It is expressed as a percentage of the current value. The default is 0.2, which says temperature is not allowed to decrease by more than 20% (there is no upward limit). If temperature corrections are reduced to meet the Positivity Rate Limit, then all other variable corrections are equally scaled in an attempt to keep everything changing at the same rate.

Note: The last two Coupled Flow model expert properties are linked, in that the Explicit relaxation factor is applied to the corrections first, then the Positivity Rate Limit condition is checked. It is possible that the Explicit relaxation factor could reduce temperature corrections such that the Positivity Rate Limit does not get invoked where it otherwise would.

- Deactivate the Preconditioning Enabled option (an expert property of the Coupled Flow Model) when you are trying to capture unsteady phenomena at small time scales (for instance, in an aeroacoustics analysis). Deactivate this property when the Integration property is set to either Implicit or Explicit .

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

Setup Recommendations (pp. 2812)

The setup recommendations are as follows:

- The default CFL number for the Coupled Implicit solver is 50.0.
- You can often increase the CFL number value to 100, 200 or even more, depending on the physical and numerical difficulty of the actual flow problem being solved and grid quality. Laminar flow and good grid quality allow even higher values and very rapid convergence. When using the Coupled Explicit Solver, limit the CFL number to values near unity.
- Turbulent flows require lower CFL numbers and more iterations than laminar flows, since equations for turbulence quantities are solved in a segregated manner outside the coupled solver. However, even for turbulent flows, the CFL number can be increased beyond 1000, especially when the mesh is refined and the simulation is initialized with a coarser mesh solution.
- In many situations, it is helpful to perform some iterations in the segregated solver before switching to the coupled solver. This procedure provides a better initial solution and allows you to set a higher Courant number in the coupled solver.
- If an approximate initial solution is used (uniform values), smaller CFL number values are required at startup.
- You can employ the Linear Ramp capability of the Coupled Implicit solver, to automate desired change of the CFL number during the simulation.
- A low CFL number value of 0.1, or even lower, may be required for startup of some difficult problems (for example, hypersonics or combustion). To ensure that the solver stability is maintained during the entire simulation, use the Linear Ramp to increase the CFL number based on iteration count.
- Depending on the problem (such as very high-speed flow or mesh quality issues) you could use an Explicit relaxation factor as low as 0.2 or 0.25. Very difficult cases may require a value of 0.15 to be able to maintain non-linear stability.
- Typically an Explicit relaxation factor of 0.75–0.85 helps convergence for most problems, without greatly affecting the speed of convergence for those cases that do not require any explicit relaxation.
- Besides the CFL number and the Explicit relaxation factor, a practical way to keep hypersonic cases within physical “bounds” is to use a lower value for the Positivity Rate Limit. The default value is 0.2, although for extreme hypersonic cases it may be necessary to use values as low as 0.05.
- For time accurate simulations, choose the time step to keep the CFL number around 1.0 on average. In this case, it is possible to set under-relaxation factors to 1.0 for all transport equations. The CFL value can be larger where the mesh is fine, and smaller where cells are large.

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

Mesh Quality Considerations (pp. 2813)

The mesh quality considerations are as follows:

- Mesh quality does influence the quality of the numerical results, irrespective of the setup: coupled vs. segregated solver, discretization schemes...
- On a mesh of a given quality and sufficient fineness, higher-order schemes yield more accurate results than lower-order ones. For a given cell count, aim to optimize the grid quality because a better mesh gives a more accurate solution regardless of the differencing schemes that are used and other parameters.
- With a poor-quality mesh, the gradient limiter is invoked more often, thus affecting its nominal second-order accuracy. Grid skewing is an important contributing factor (much more than grid stretching) to the loss in nominal accuracy of the solution. Grid design (distribution of cell size, local refinement using feature edges, boundary region, or volume shapes) is important in maximizing the accuracy for a given effort. Two grids with the same number of cells may lead to discretization errors that differ by an order of magnitude. Thus, improving mesh quality is a top priority.
- When solving steady-state problems on fine meshes, start with a much coarser mesh, and then successively refine the mesh by reducing the base size. For example:
  - Design the desired mesh for the highest cell count affordable.
  - Increase the base size by a factor of 8 and start the computation with this coarse mesh. You may require lower under-relaxation values for the coarsest mesh, but iterations are fast and this does not increase the computing effort much.
  - After the convergence criteria are satisfied or maximum specified number of iterations is reached, halve the mesh base size, remesh, and restart the simulation. Repeat until the final mesh size is reached.

In this way, you obtain faster convergence on finer grids by providing a good initial solution. Usually, one needs only a third or a quarter of the number of iterations that would be needed if starting with an initial guess, like constant or zero values. In addition, you obtain solutions on a series of grids which are systematically refined (same design, only base sizes reduced), which allows for an estimate of discretization errors using Richardson extrapolation. For example, if base size is reduced by a factor of 2 and second-order discretization is used, discretization errors on the finest mesh are equal to one third of the difference between solutions on the finest and next coarser grid. For base size reduction by a factor of 1.5, the errors amount to about 80% of the difference in the two solutions.

Troubleshooting the Solution (cont.)

(STAR-CCM+ User Guide)

Troubleshooting (pp. 2814)

Flow solution diverges during initial iterations

If the residuals show that the solution is diverging during the first several iterations, and there is no sign that they are decreasing, this indicates a problem.

Make sure that the overall setup is correct. Some things to look for are:

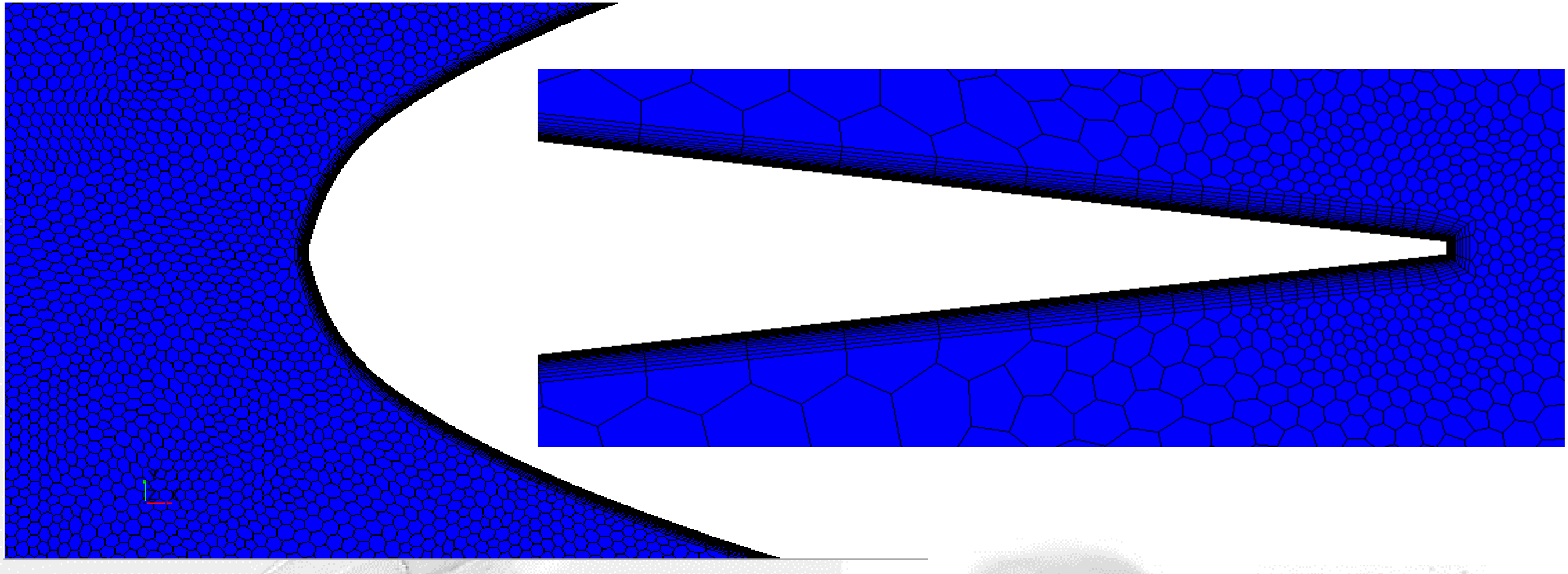
- Mesh extents for viscous flow
- Mesh quality
- Boundary conditions for both flow and turbulence, if turbulent flow is activated
- Try reducing the Courant number.
- It is possible to make the Courant number as small as desired to get the solution started. If it seems necessary to run at a Courant number significantly less than 1 for more than just a few iterations, there is most likely something wrong with your solution setup.
- Once the residuals start coming down again, it is possible to increase the Courant number.

Warnings occur of corrections being limited

Because temperature is a positive definite quantity, STAR-CCM+ tries to limit the corrections so that large changes do not tend to drive this quantity negative.

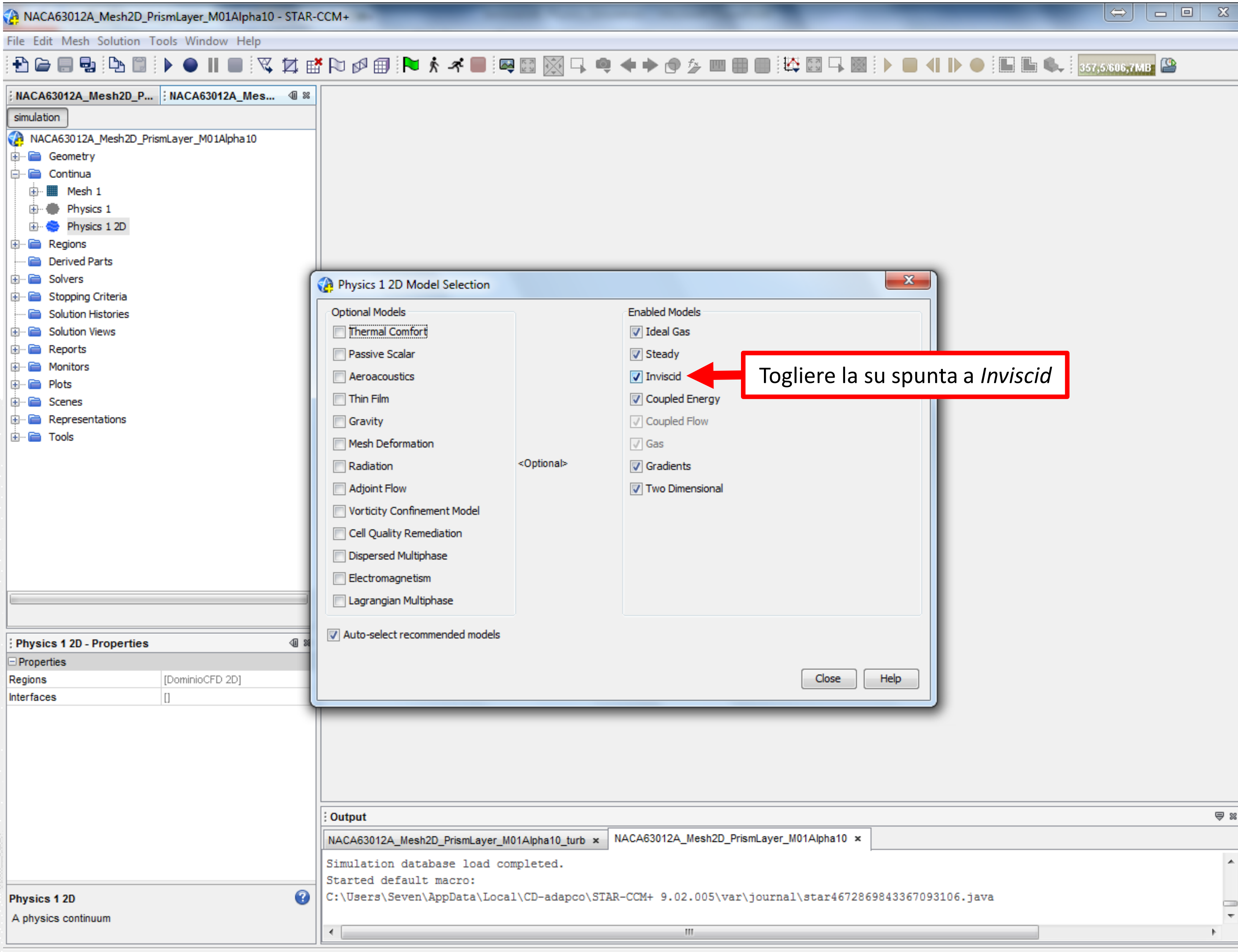
- It is acceptable for these warnings to occur during startup, as long as they cease after several iterations. If they do not, it is necessary to reduce the Courant number

Impostazione della fisica: soluzione viscosa, basso subsonico (M_∞=0.1, α=10 °)



La griglia che utilizziamo in questa applicazione è uguale alla precedente,
ma attiveremo un modello per la turbolenza

N.B.: le griglie create finora, a rigore, non sono adatte per essere utilizzate ad alti valori di angolo di attacco.


NACA63012A_Mesh2D_PrismLayer_M01Alpha10 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PrismLayer_M01Alpha10
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Representations
Tools
Physics 1 2D - Properties
Regions [DominioCFD 2D]
Interfaces []
Physics 1 2D
A physics continuum
Physics 1 2D Model Selection
Optional Models
Thermal Comfort
Passive Scalar
Aeroacoustics
Thin Film
Gravity
Mesh Deformation
Radiation
Adjoint Flow
Vorticity Confinement Model
Cell Quality Remediation
Dispersed Multiphase
Electromagnetism
Lagrangian Multiphase
<Optional>
Enabled Models
Ideal Gas
Steady
Inviscid
Coupled Energy
Coupled Flow
Gas
Gradients
Two Dimensional
Auto-select recommended models
Close
Help
Togliere la su spunta a *Inviscid*
Output
NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb
NACA63012A_Mesh2D_PrismLayer_M01Alpha10
Simulation database load completed.
Started default macro:
C:\Users\Seven\AppData\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star4672869843367093106.java

Selecting a Turbulence Modeling Approach (pp. 3329)

(STAR-CCM+ User Guide)

Three basic approaches to modeling turbulence are available in STAR-CCM+:

- Models that provide closure of the Reynolds-Averaged Navier-Stokes (RANS) equations.
- Large eddy simulation (LES).
- Detached eddy simulation (DES).

How Do I Decide on a Turbulence Model? (pp. 3338)

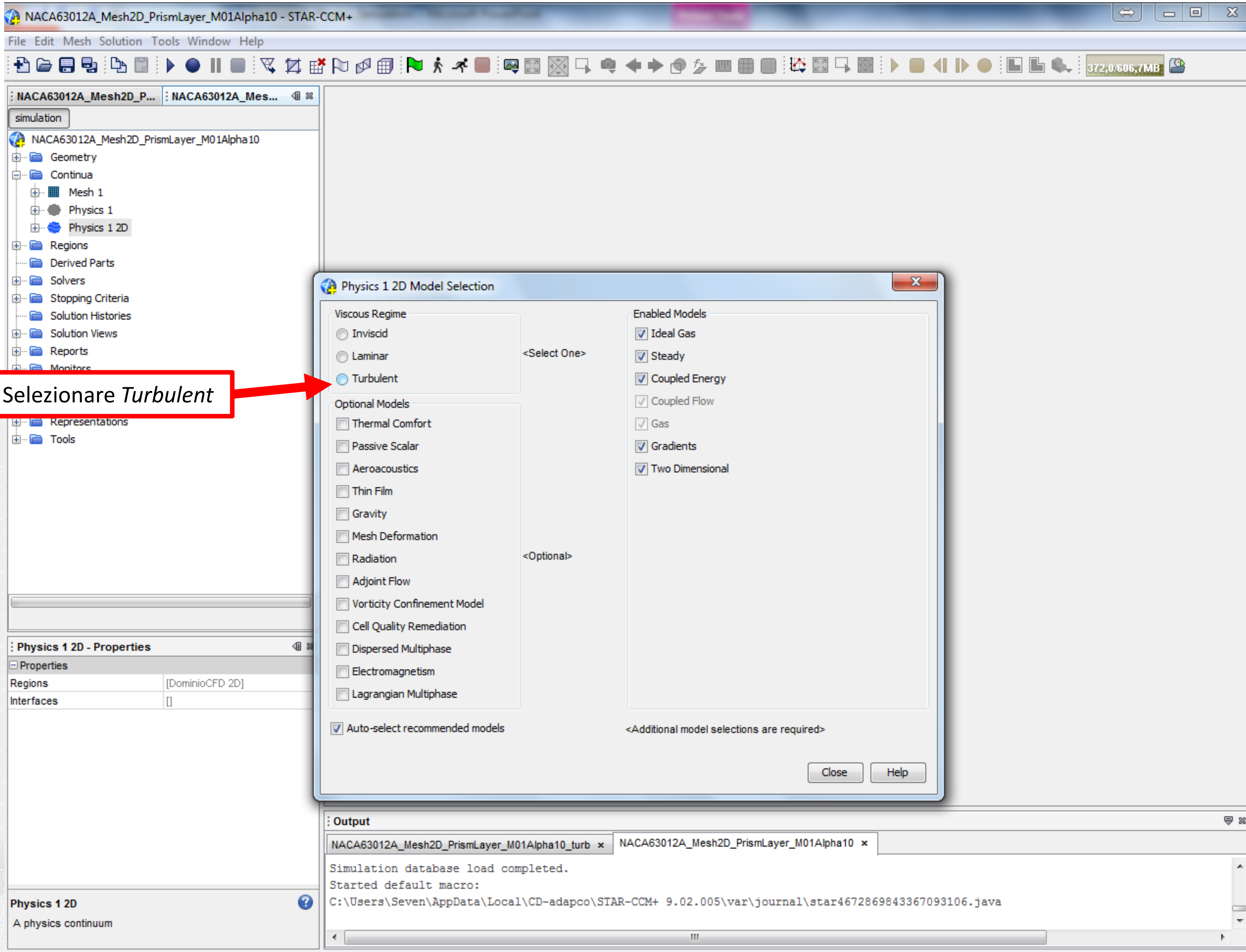
There are four major classes of turbulence models currently in STAR-CCM+. This section presents broad guidelines as to the applicability of each of these. Further guidance on selecting the specific model variants can be found within the sections that provide details about the models. If divergence occurs, you do not need to exit from the simulation. Instead, you can clear the solution and run it again after investigating the cause of divergence and making the required changes to the simulation.

- Spalart-Allmaras models are a good choice for applications in which the boundary layers are largely attached and separation is mild if it occurs. Typical examples would be flow over a wing, fuselage or other aerospace external-flow applications. The Spalart-Allmaras models for RANS equations are not suited to flows that are dominated by free-shear layers, flows where complex recirculation occurs (particularly with heat transfer), or natural convection. This statement does not apply to the Spalart-Allmaras detached eddy model.

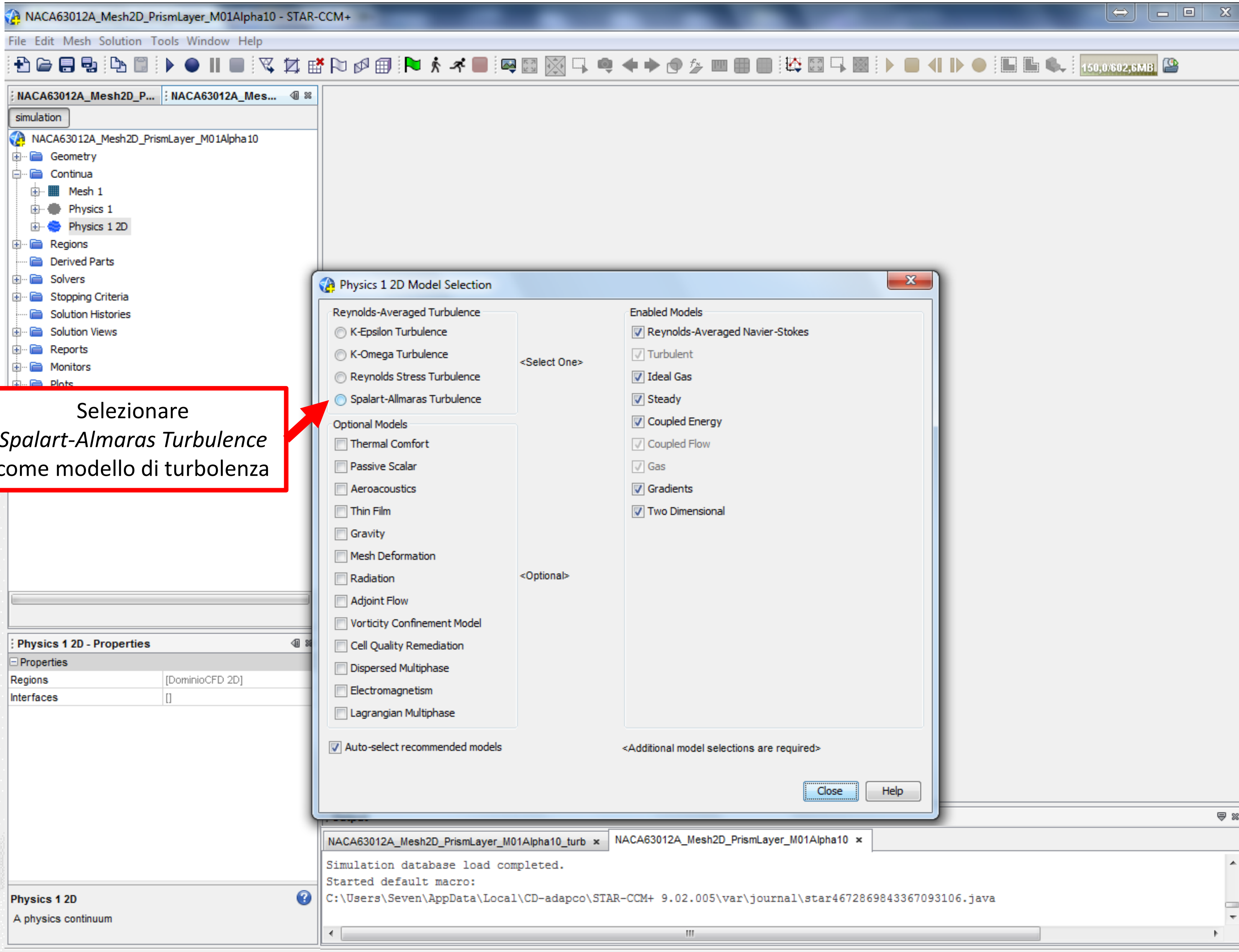
- K-Epsilon models provide a good compromise between robustness, computational cost and accuracy. They are generally well suited to industrial-type applications that contain complex recirculation, with or without heat transfer.

- K-Omega models are similar to K-Epsilon models in that two transport equations are solved, but differ in the choice of the second transported turbulence variable. The performance differences are likely to be a result of the subtle differences in the models, rather than a higher degree of complexity in the physics being captured. These models have seen most application in the aerospace industry. Therefore, they are recommended as an alternative to the Spalart-Allmaras models for similar types of applications.

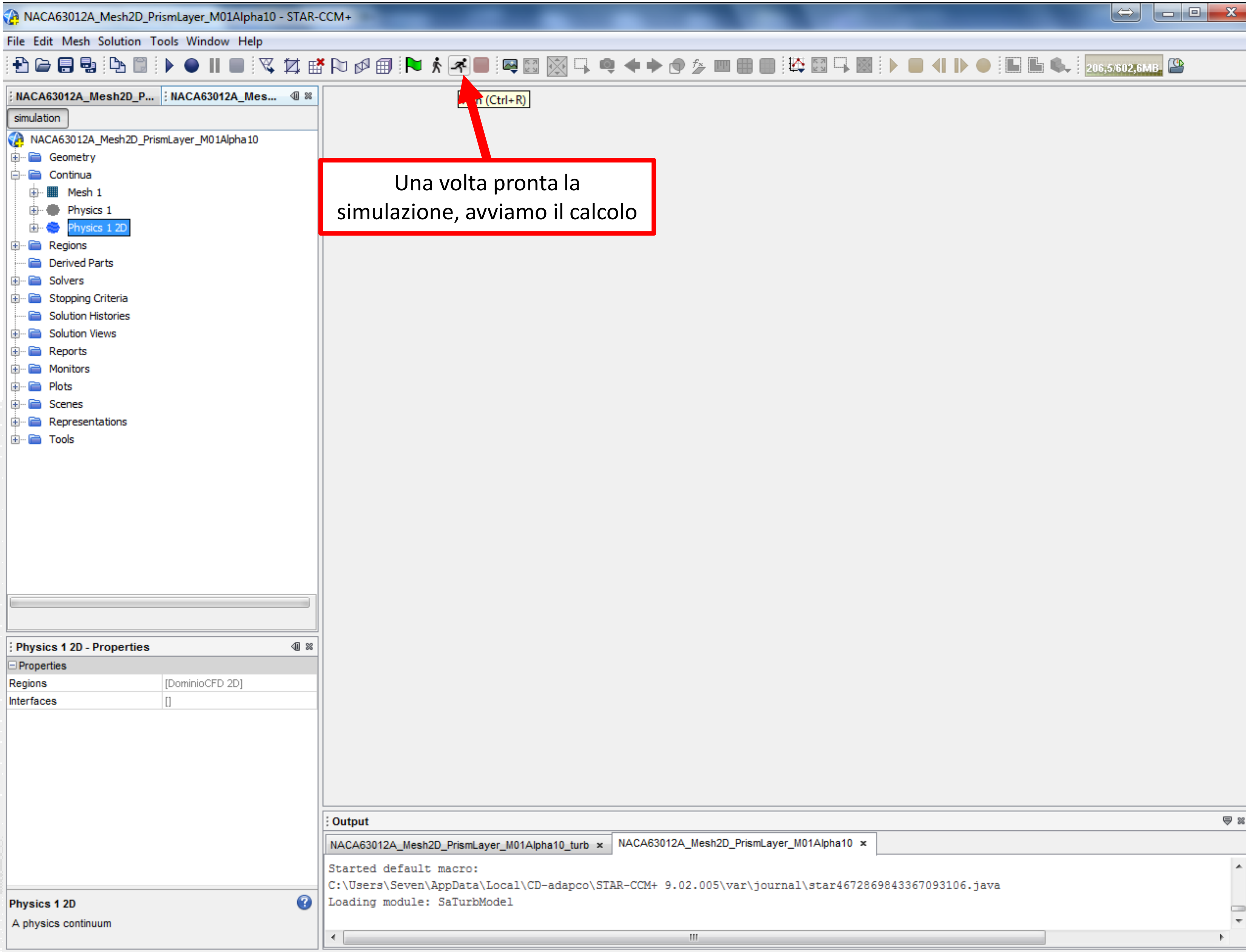
- Reynolds stress transport models are the most complex and computationally expensive models offered in STAR-CCM+. They are recommended for situations in which the turbulence is strongly anisotropic, such as the swirling flow in a cyclone separator.

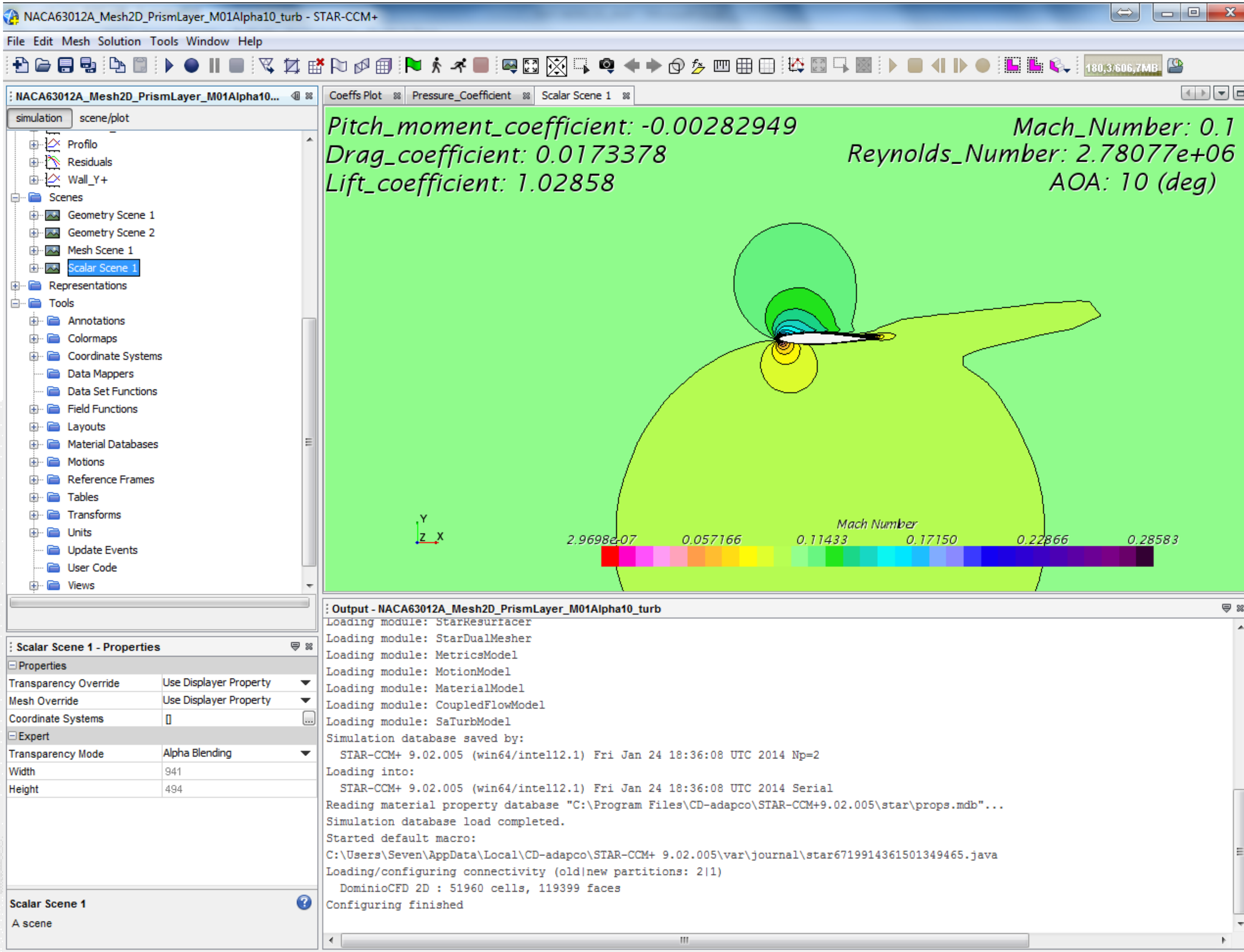


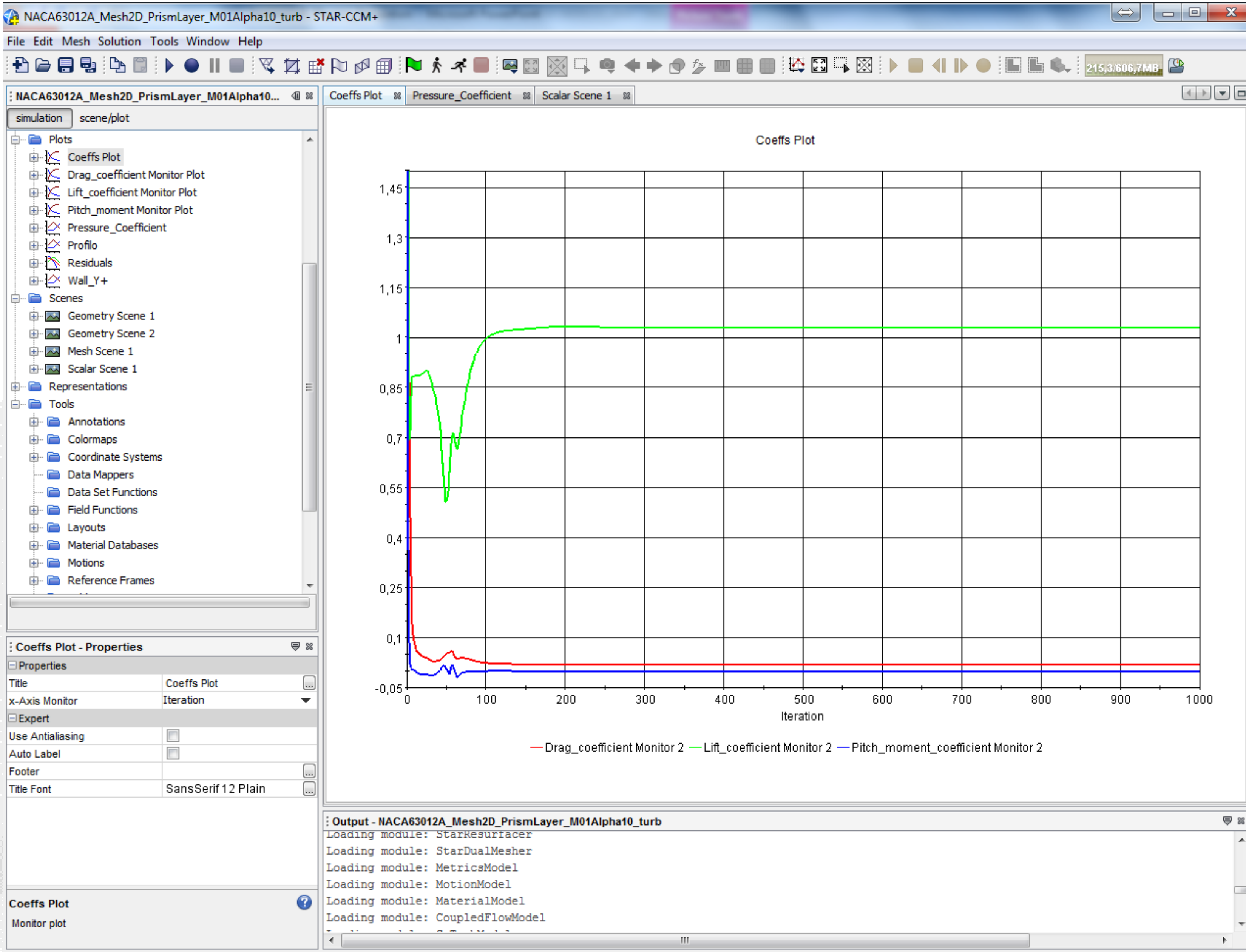
Selezionare *Turbulent*

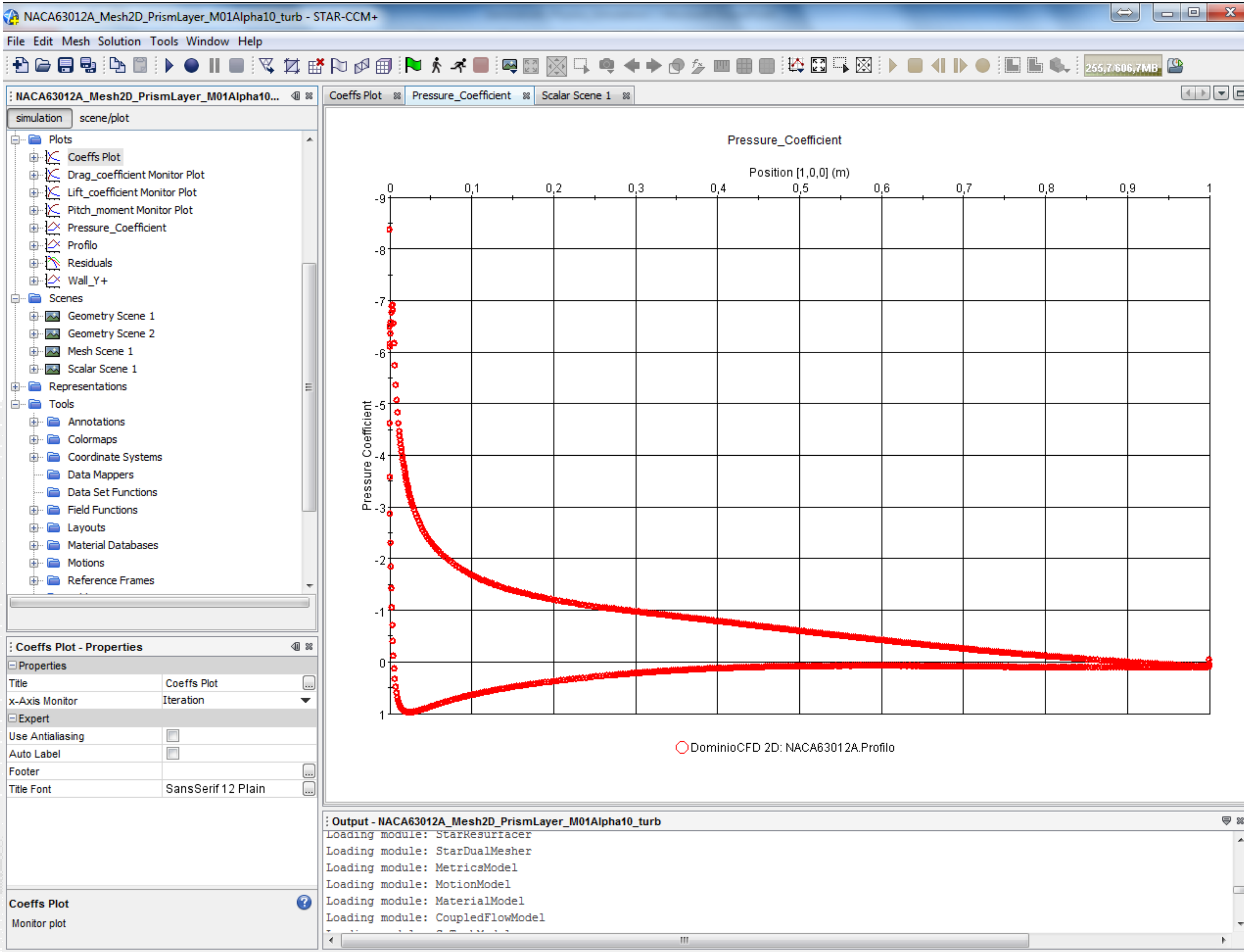


Selezionare
Spalart-Almaras Turbulence
come modello di turbolenza


NACA63012A_Mesh2D_PrismLayer_M01Alpha10 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
(Ctrl+R)
Una volta pronta la simulazione, avviamo il calcolo
simulation
NACA63012A_Mesh2D_PrismLayer_M01Alpha10
Geometry
Continua
Mesh 1
Physics 1
Physics 1 2D
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Representations
Tools
Physics 1 2D - Properties
Properties
Regions [DominioCFD 2D]
Interfaces []
Physics 1 2D
A physics continuum
Output
NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb
NACA63012A_Mesh2D_PrismLayer_M01Alpha10
Started default macro:
C:\Users\Seven\AppData\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star4672869843367093106.java
Loading module: SaTurbModel


NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PrismLayer_M01Alpha10...
simulation
scene/plot
Profilo
Residuals
Wall_Y+
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Representations
Tools
Annotations
Colormaps
Coordinate Systems
Data Mappers
Data Set Functions
Field Functions
Layouts
Material Databases
Motions
Reference Frames
Tables
Transforms
Units
Update Events
User Code
Views
Scalar Scene 1 - Properties
Properties
Transparency Override
Use Displayer Property
Mesh Override
Use Displayer Property
Coordinate Systems
[]
Expert
Transparency Mode
Alpha Blending
Width
941
Height
494
Scalar Scene 1
A scene
Coeffs Plot
Pressure_Coefficient
Scalar Scene 1
Pitch_moment_coefficient: -0.00282949
Drag_coefficient: 0.0173378
Lift_coefficient: 1.02858
Mach_Number: 0.1
Reynolds_Number: 2.78077e+06
AOA: 10 (deg)
Y
Z X
Mach Number
2.9698e-07
0.057166
0.11433
0.17150
0.22866
0.28583
Output - NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb
Loading module: StarResurfacer
Loading module: StarDualMesher
Loading module: MetricsModel
Loading module: MotionModel
Loading module: MaterialModel
Loading module: CoupledFlowModel
Loading module: SaTurbModel
Simulation database saved by:
STAR-CCM+ 9.02.005 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Np=2
Loading into:
STAR-CCM+ 9.02.005 (win64/intel12.1) Fri Jan 24 18:36:08 UTC 2014 Serial
Reading material property database "C:\Program Files\CD-adapco\STAR-CCM+9.02.005\star\props.mdb"...
Simulation database load completed.
Started default macro:
C:\Users\Seven\AppData\Local\CD-adapco\STAR-CCM+ 9.02.005\var\journal\star6719914361501349465.java
Loading/configuring connectivity (old|new partitions: 2|1)
DominioCFD 2D : 51960 cells, 119399 faces
Configuring finished


NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PrismLayer_M01Alpha10...
simulation
scene/plot
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Pressure_Coefficient
Profilo
Residuals
Wall_Y+
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Representations
Tools
Annotations
Colormaps
Coordinate Systems
Data Mappers
Data Set Functions
Field Functions
Layouts
Material Databases
Motions
Reference Frames
Coeffs Plot - Properties
Properties
Title
Coeffs Plot
x-Axis Monitor
Iteration
Expert
Use Antialiasing
Auto Label
Footer
Title Font
SansSerif 12 Plain
Coeffs Plot
Monitor plot
Coeffs Plot
Pressure_Coefficient
Scalar Scene 1
Coeffs Plot
1,45
1,3
1,15
1
0,85
0,7
0,55
0,4
0,25
0,1
-0,05
0
100
200
300
400
500
600
700
800
900
1000
Iteration
Drag_coefficient Monitor 2
Lift_coefficient Monitor 2
Pitch_moment_coefficient Monitor 2
Output - NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb
Loading module: StarDualMesher
Loading module: MetricsModel
Loading module: MotionModel
Loading module: MaterialModel
Loading module: CoupledFlowModel


NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb - STAR-CCM+
File Edit Mesh Solution Tools Window Help
NACA63012A_Mesh2D_PrismLayer_M01Alpha10...
simulation
scene/plot
Plots
Coeffs Plot
Drag_coefficient Monitor Plot
Lift_coefficient Monitor Plot
Pitch_moment Monitor Plot
Pressure_Coefficient
Profilo
Residuals
Wall_Y+
Scenes
Geometry Scene 1
Geometry Scene 2
Mesh Scene 1
Scalar Scene 1
Representations
Tools
Annotations
Colormaps
Coordinate Systems
Data Mappers
Data Set Functions
Field Functions
Layouts
Material Databases
Motions
Reference Frames
Coeffs Plot - Properties
Properties
Title
Coeffs Plot
x-Axis Monitor
Iteration
Expert
Use Antialiasing
Auto Label
Footer
Title Font
SansSerif 12 Plain
Coeffs Plot
Monitor plot
Coeffs Plot
Pressure_Coefficient
Scalar Scene 1
Pressure_Coefficient
Position [1,0,0] (m)
0
0,1
0,2
0,3
0,4
0,5
0,6
0,7
0,8
0,9
1
Pressure Coefficient
-9
-8
-7
-6
-5
-4
-3
-2
-1
0
1
DominioCFD 2D: NACA63012A.Profilo
Output - NACA63012A_Mesh2D_PrismLayer_M01Alpha10_turb
Loading module: StarResurfacer
Loading module: StarDualMesher
Loading module: MetricsModel
Loading module: MotionModel
Loading module: MaterialModel
Loading module: CoupledFlowModel