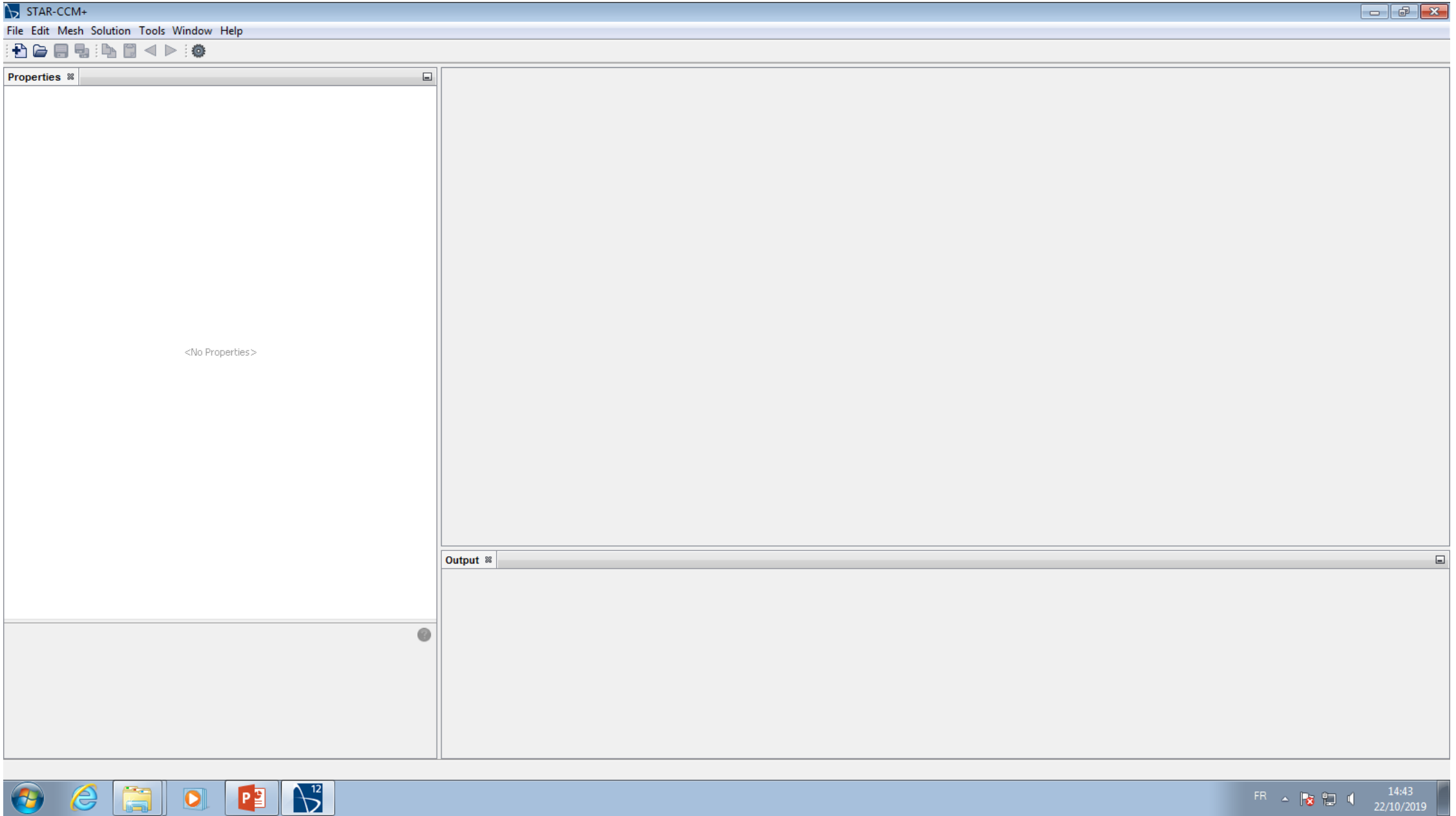
Master REST
Modelization of a wind farm using a 1D momentum model

This tutorial was done using StarCCM+ v 12.06.011

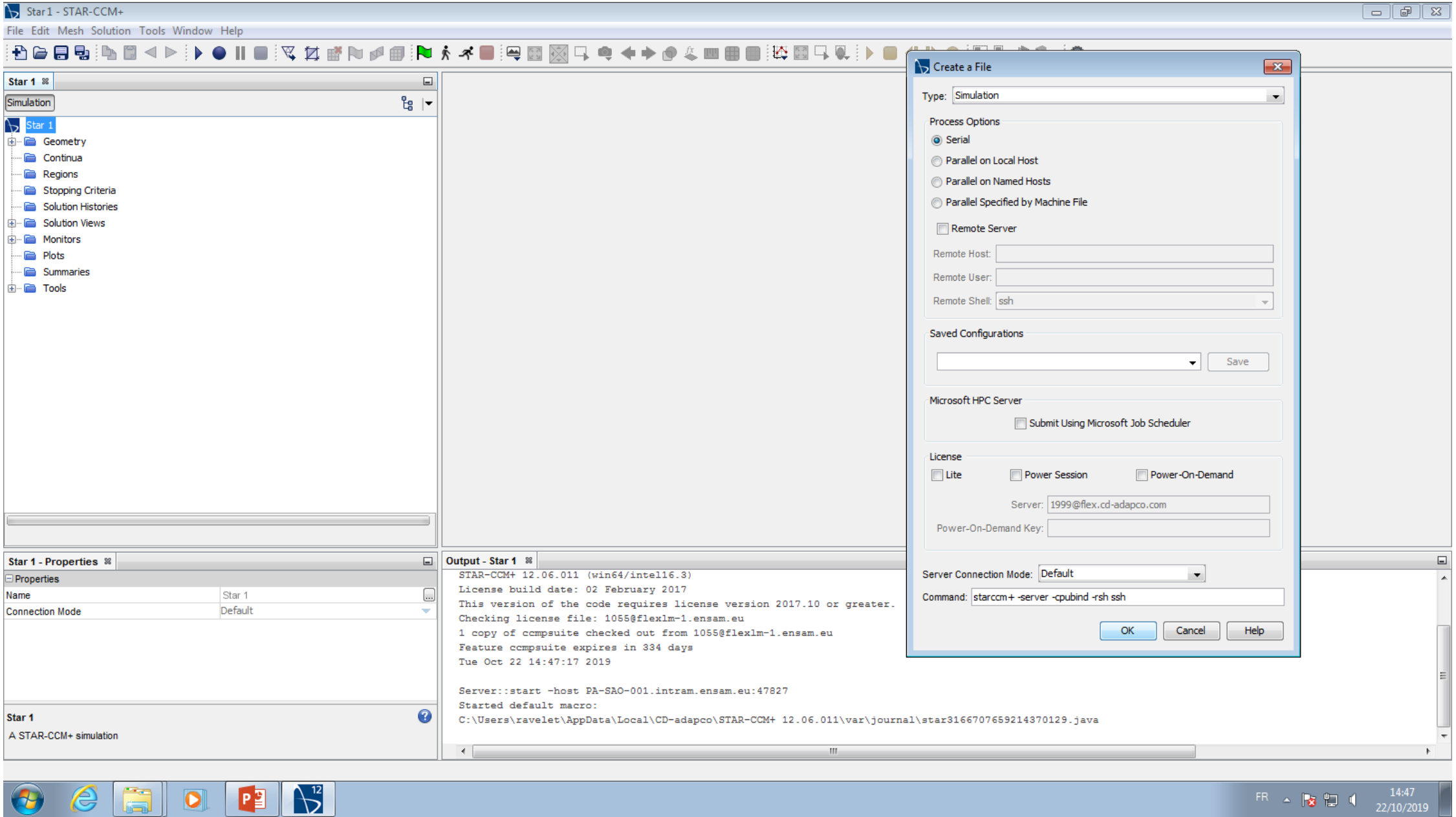
The interface can slightly change between the different versions of the software. Different versions may be installed in the different classrooms of the school.

Please do not follow slide by slide this tutorial without asking you what you are currently doing. Be clever, smart, etc, etc, ask questions and refer to the online help of the software.

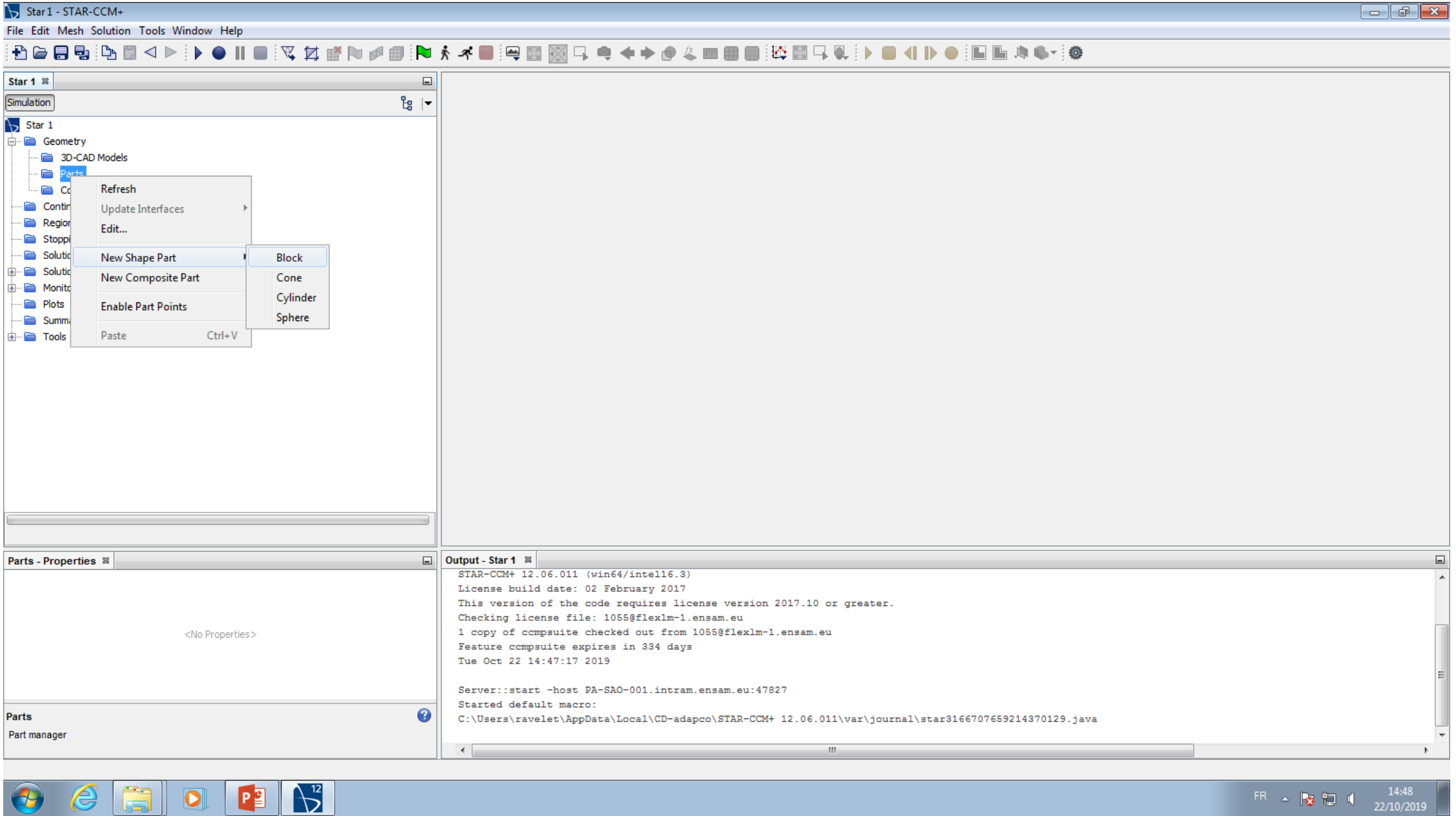
Launch StarCCM+



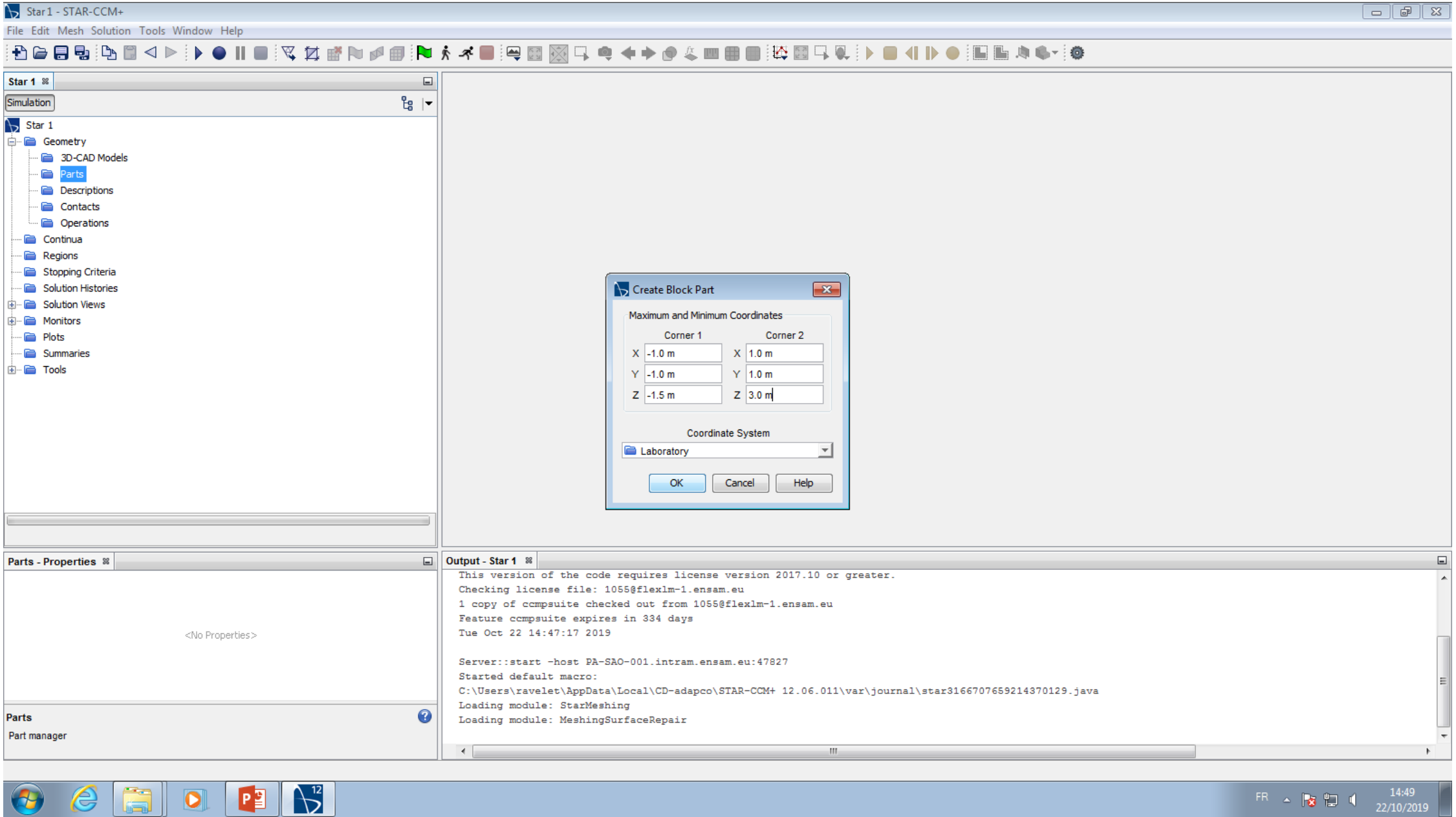
Create a new simulation



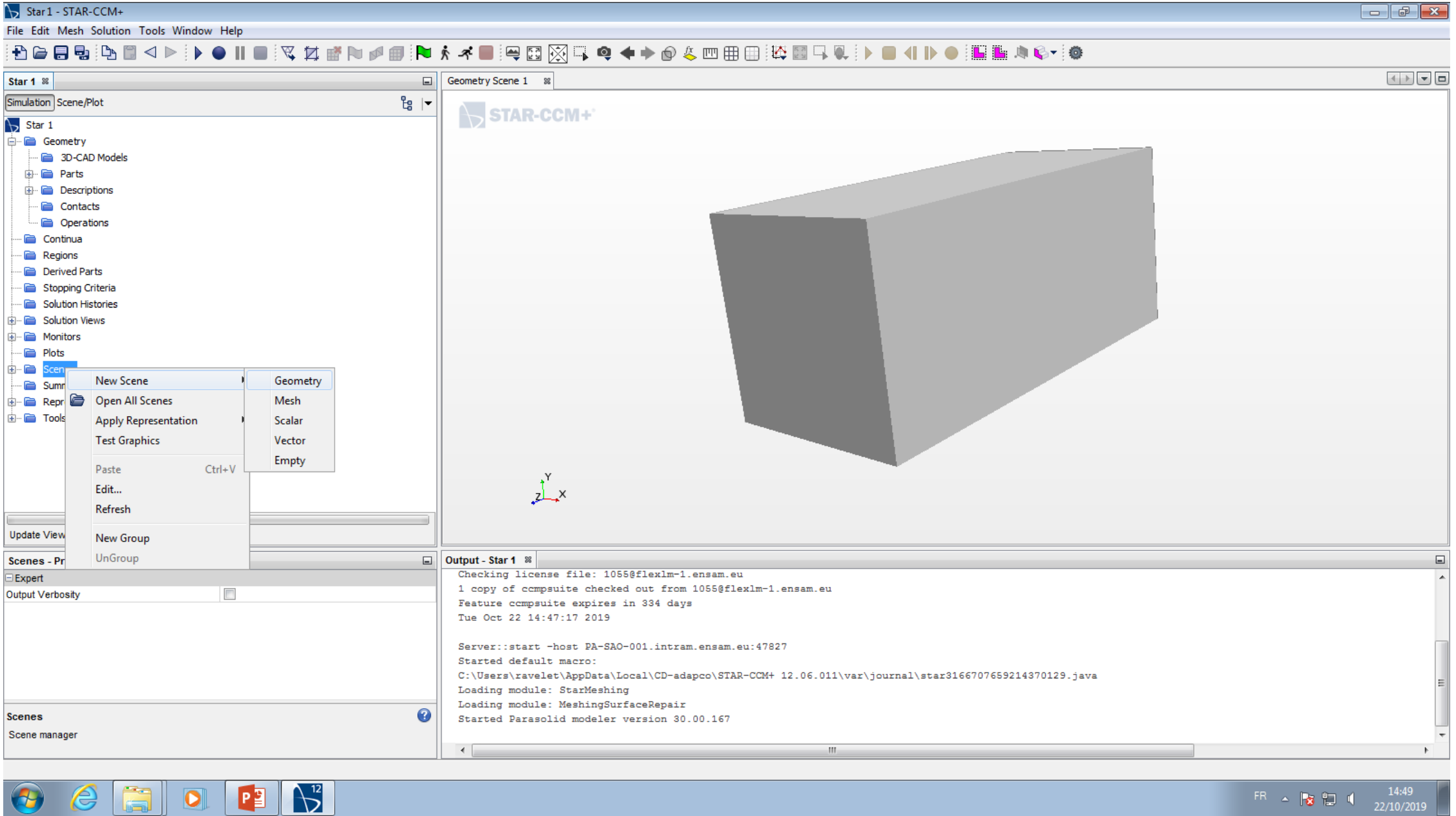
Create a Block



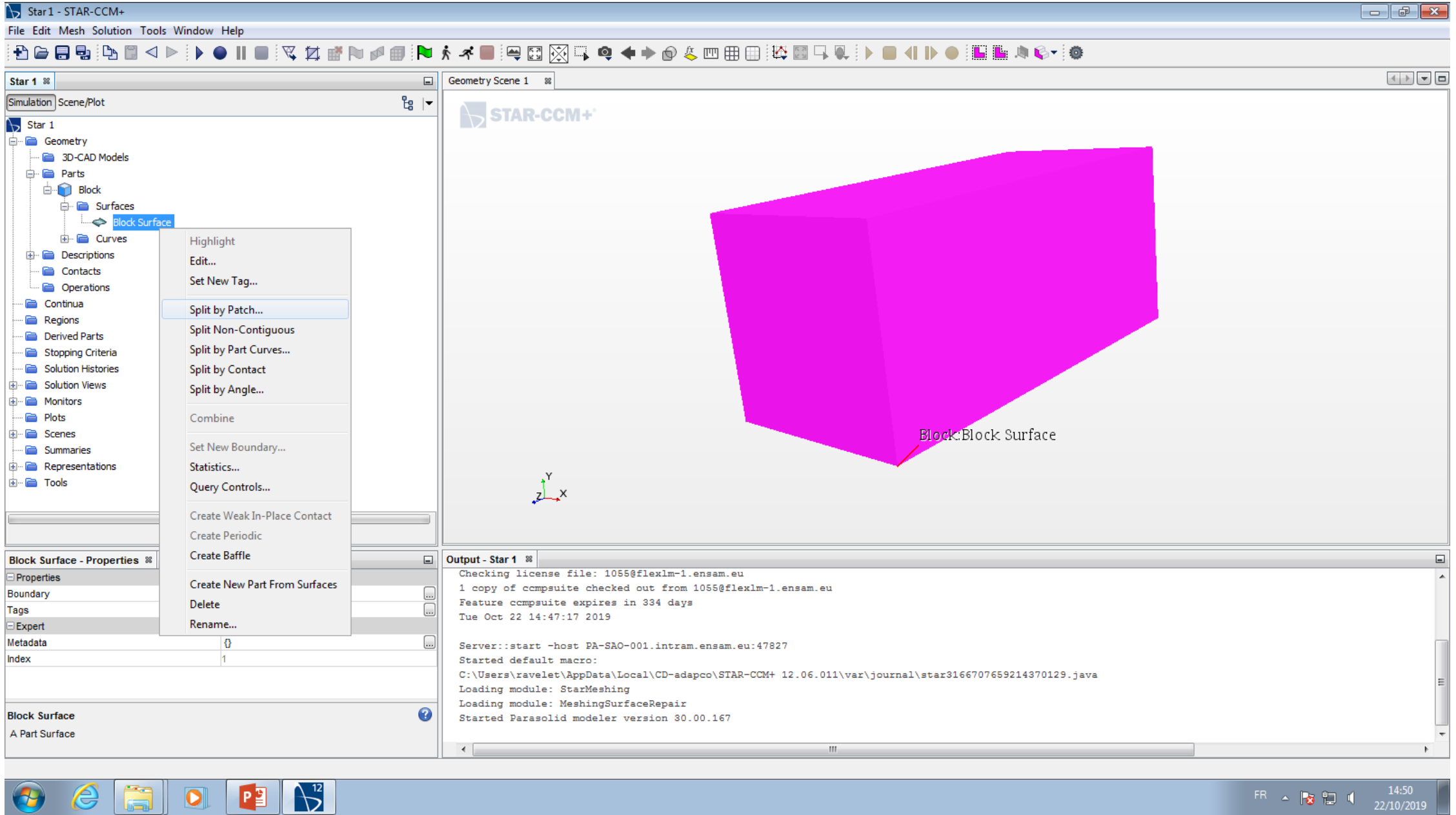
Adapt the size of the block to your case



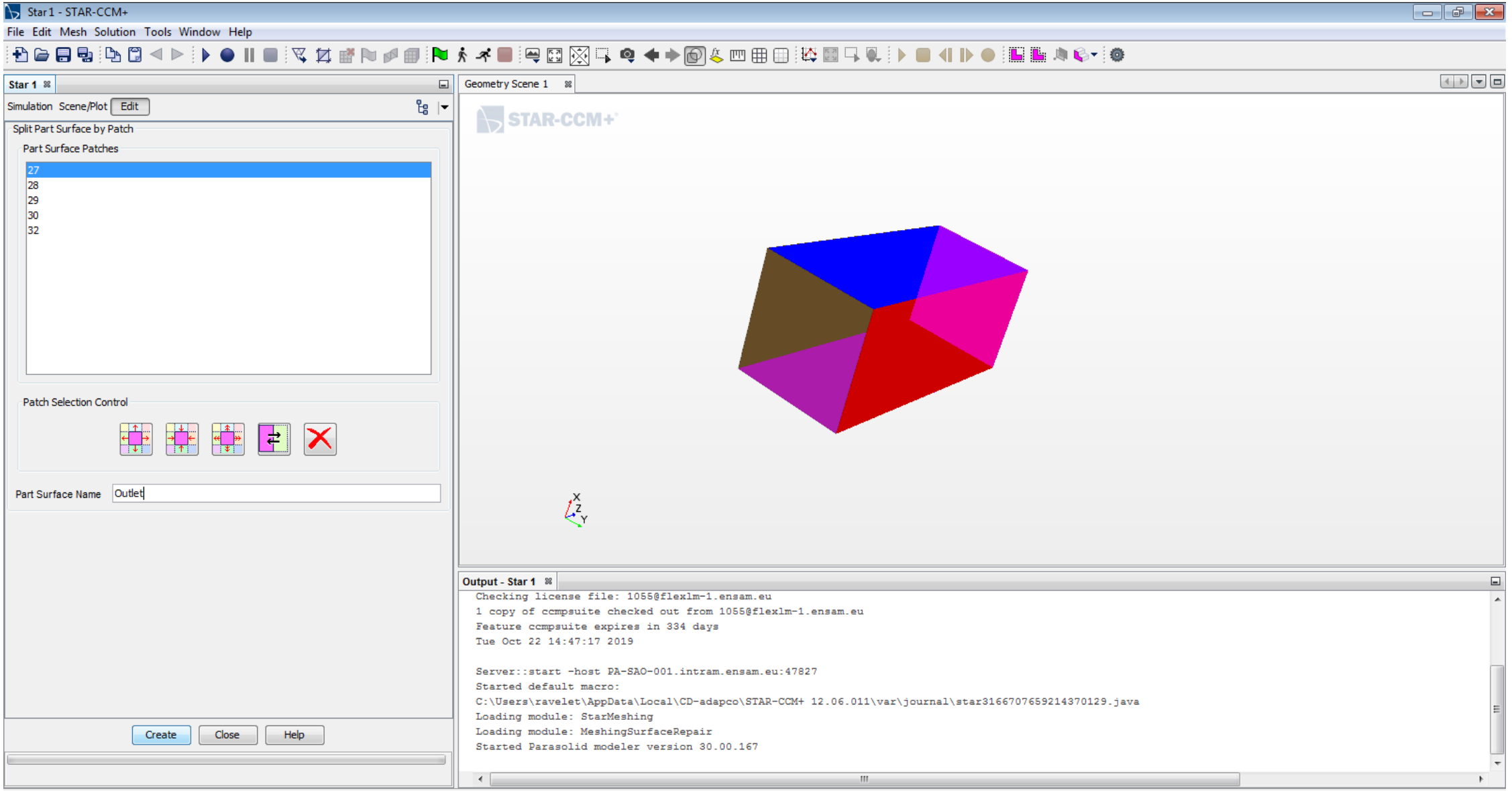
Visualize: create a new « scene »



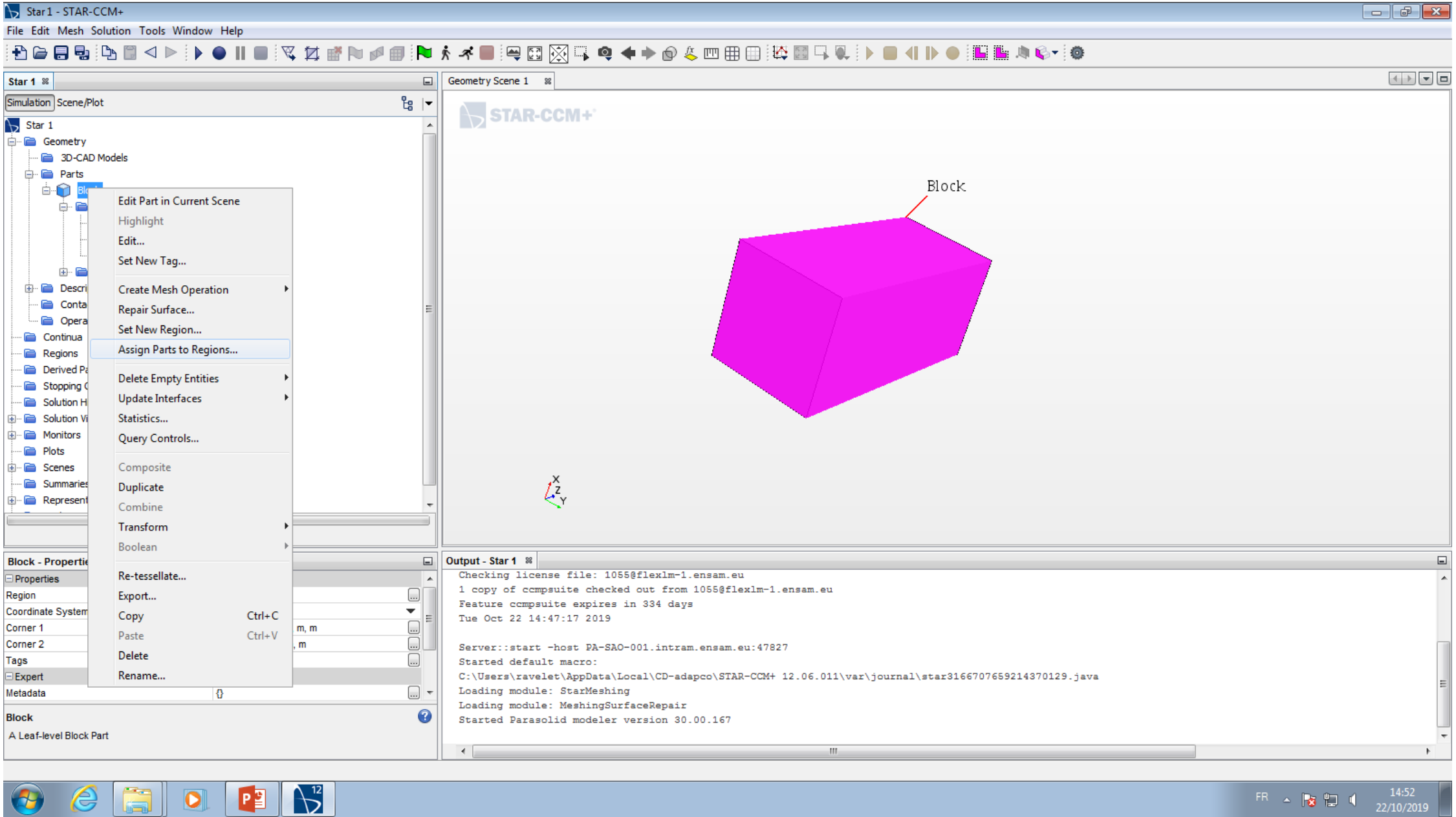
Prepare the boundaries: « split by patch »



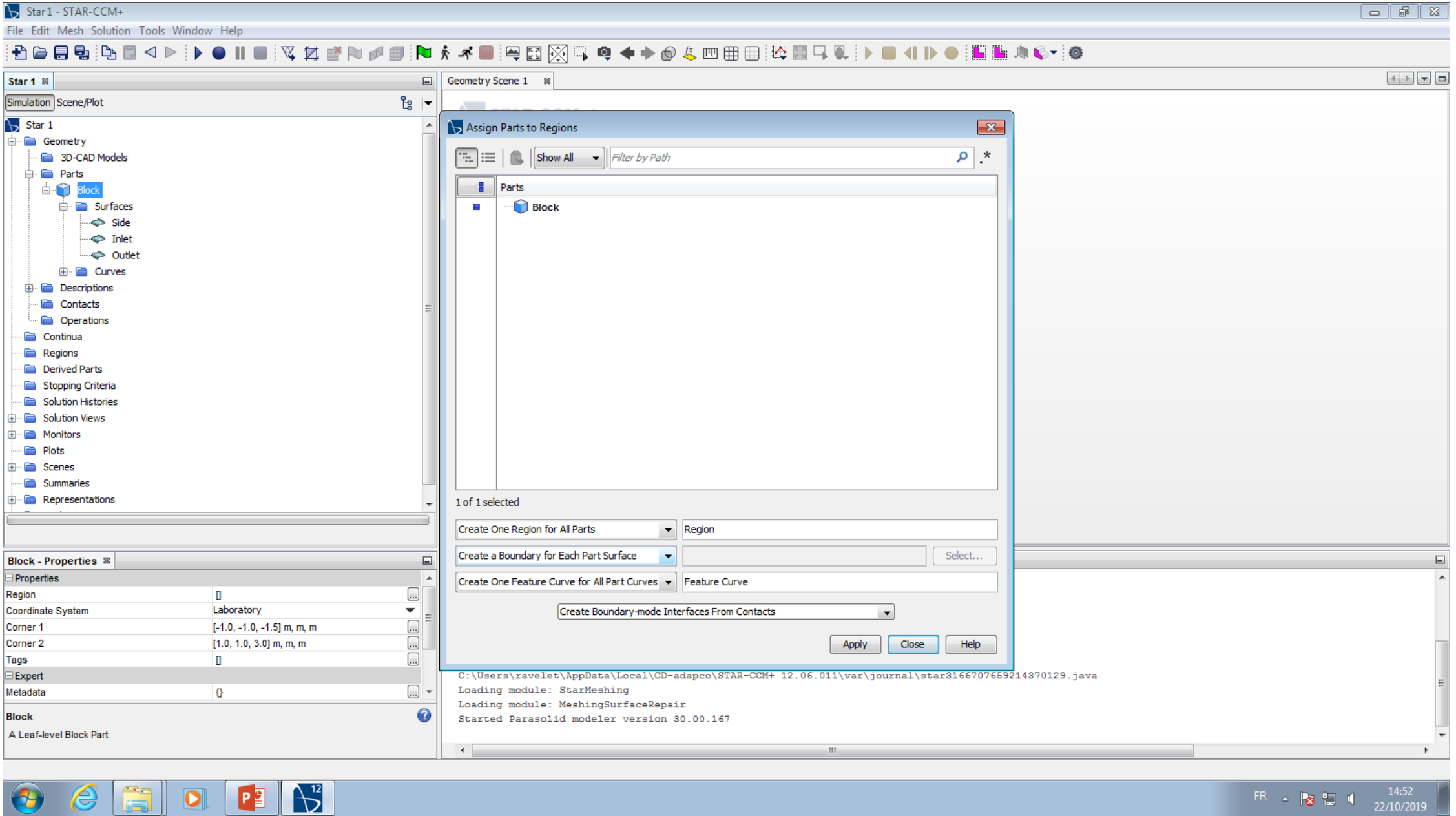
Select the inlet, rename it, create the boundary
Idem for the outlet; the remaining four patches will be the side walls



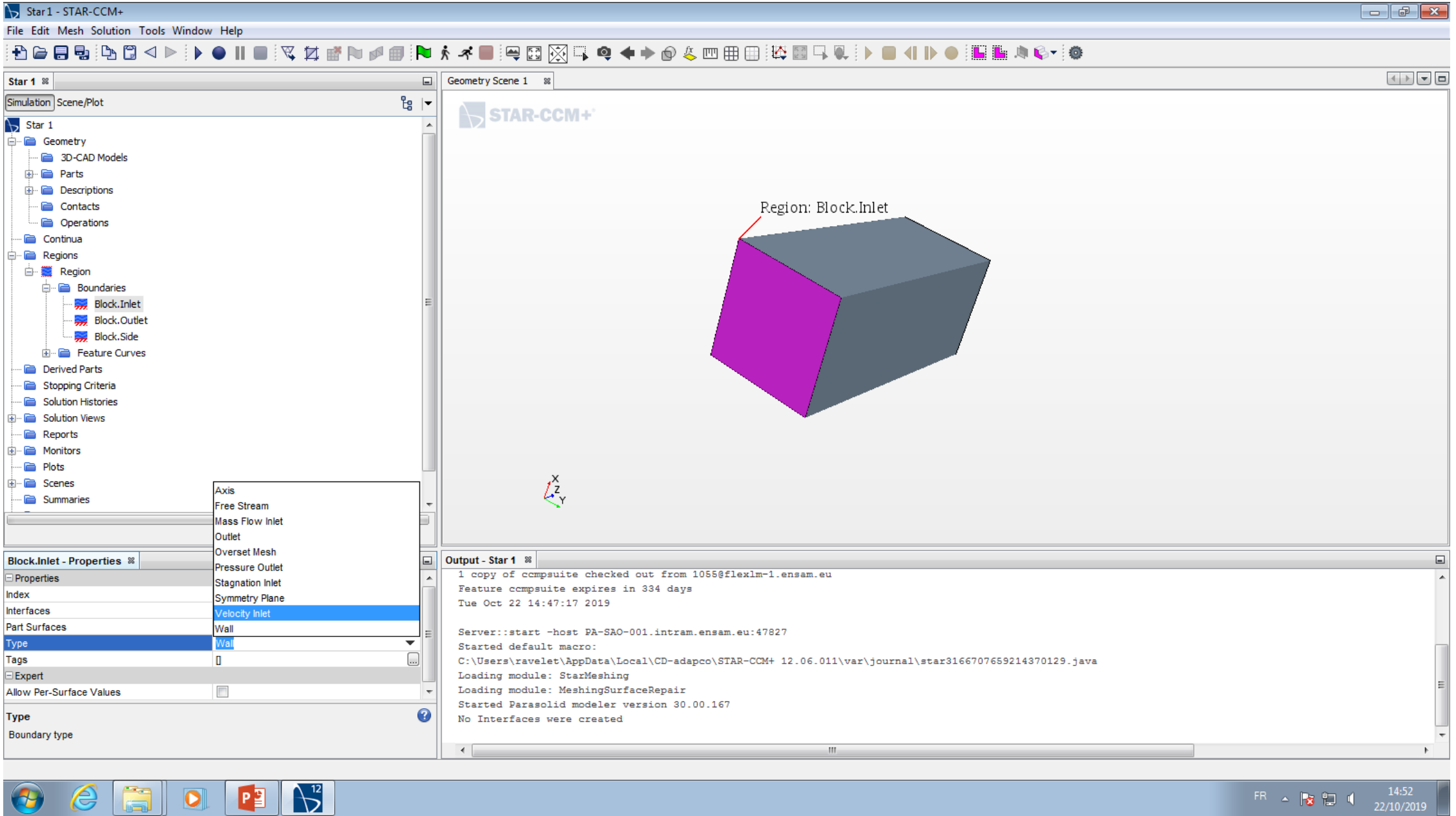
Create the « Region »: the fluid domain

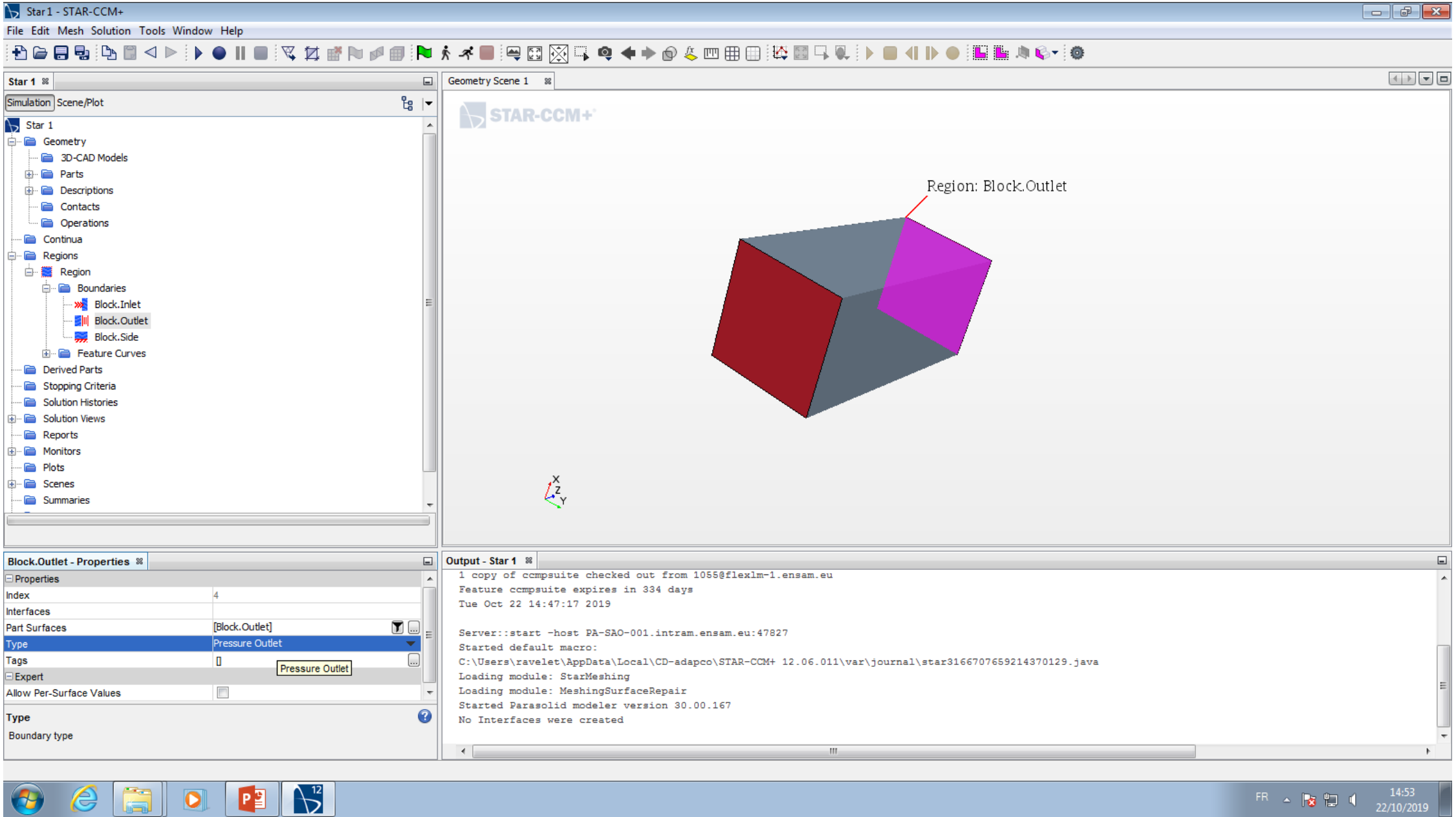


Be sure to create « a boundary for each part surface »

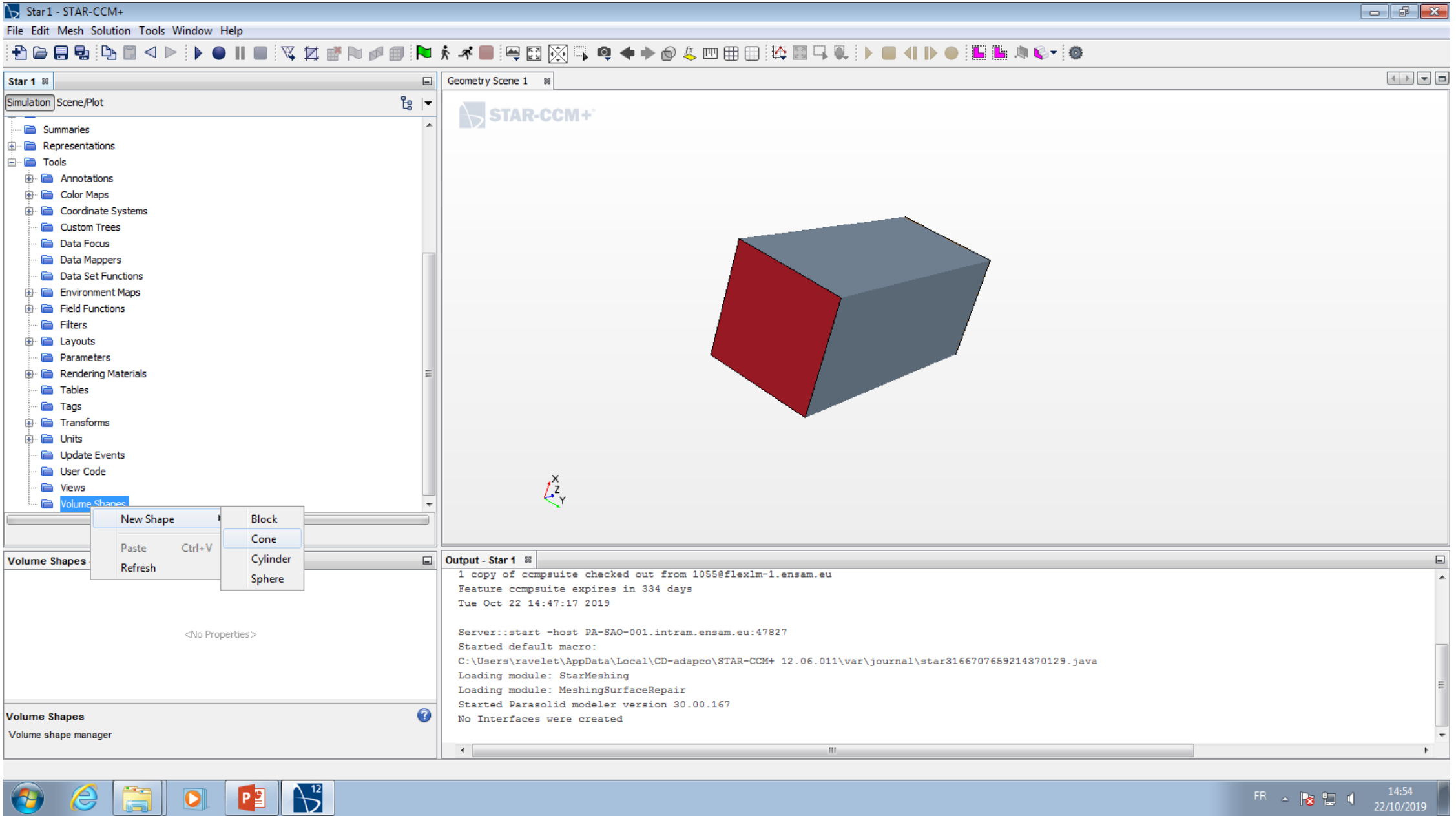


Change the boundary type: « velocity inlet » for the inlet,
« pressure outlet » for the outlet and wall for the side walls

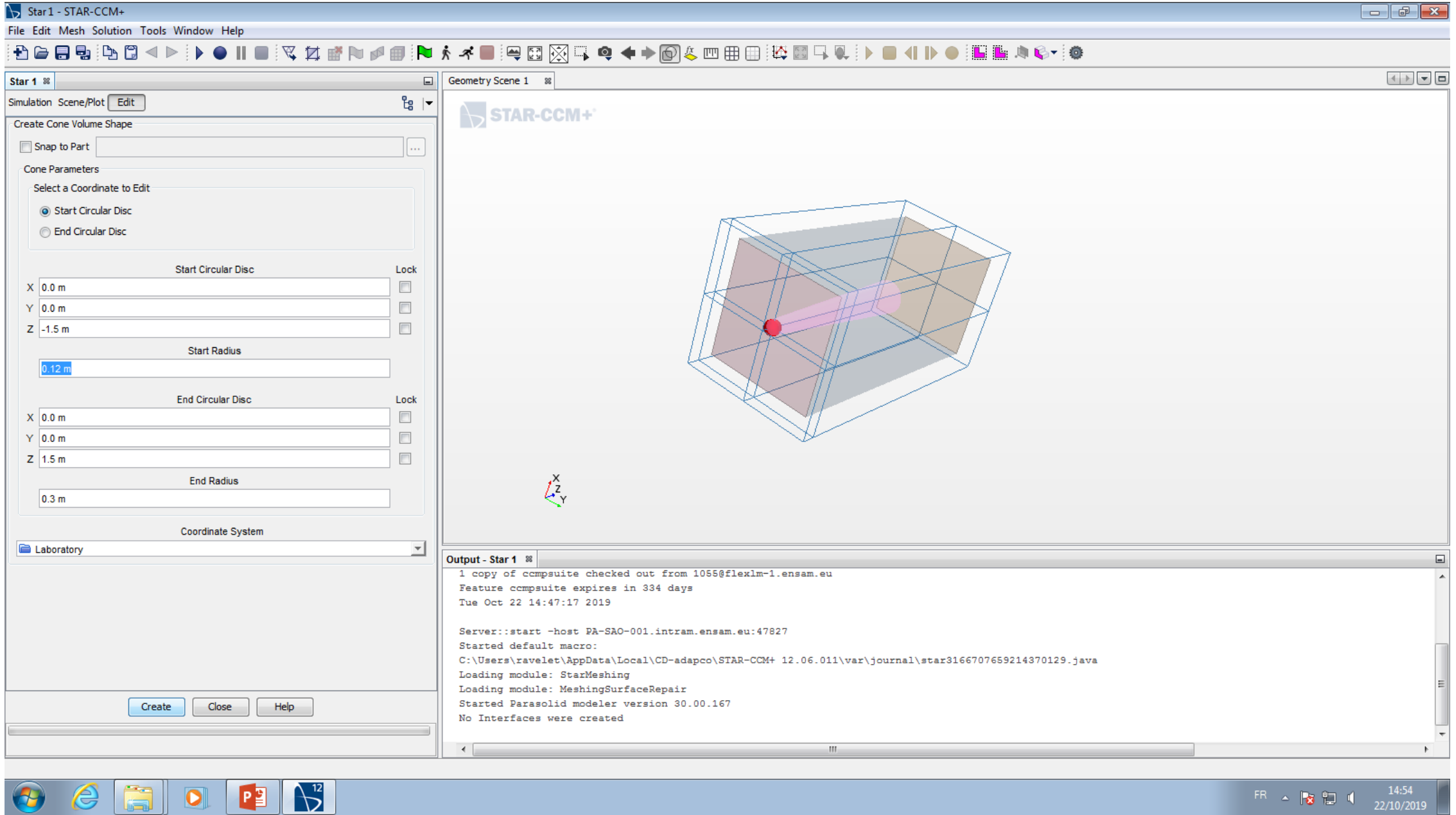



Star 1 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Star 1
Simulation Scene/Plot
Star 1
Geometry
3D-CAD Models
Parts
Descriptions
Contacts
Operations
Continua
Regions
Region
Boundaries
Block.Inlet
Block.Outlet
Block.Side
Feature Curves
Derived Parts
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Summaries
Geometry Scene 1
STAR-CCM+
Region: Block.Outlet
X
Z
Y
Block.Outlet - Properties
Properties
Index 4
Interfaces
Part Surfaces [Block.Outlet]
Type Pressure Outlet
Tags []
Expert
Allow Per-Surface Values
Pressure Outlet
Type
Boundary type
Output - Star 1
1 copy of ccmpsuite checked out from 1055@flexlm-1.ensam.eu
Feature ccmpsuite expires in 334 days
Tue Oct 22 14:47:17 2019
Server::start -host PA-SAO-001.intram.ensam.eu:47827
Started default macro:
C:\Users\ravelet\AppData\Local\CD-adapco\STAR-CCM+ 12.06.011\var\journal\star3166707659214370129.java
Loading module: StarMeshing
Loading module: MeshingSurfaceRepair
Started Parasolid modeler version 30.00.167
No Interfaces were created
FR
14:53
22/10/2019

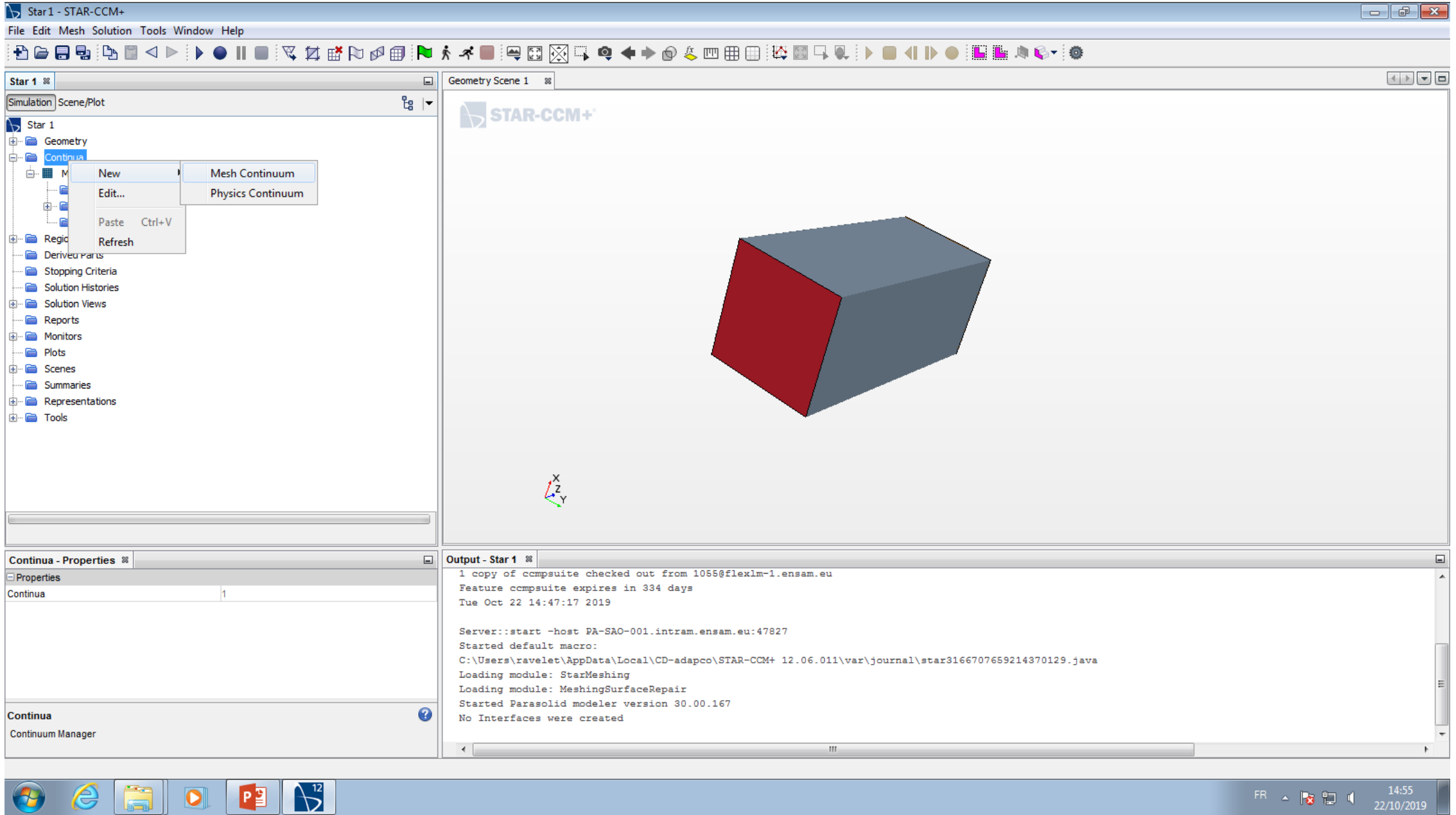
Create a voulme shape for refinment of the (future) mesh



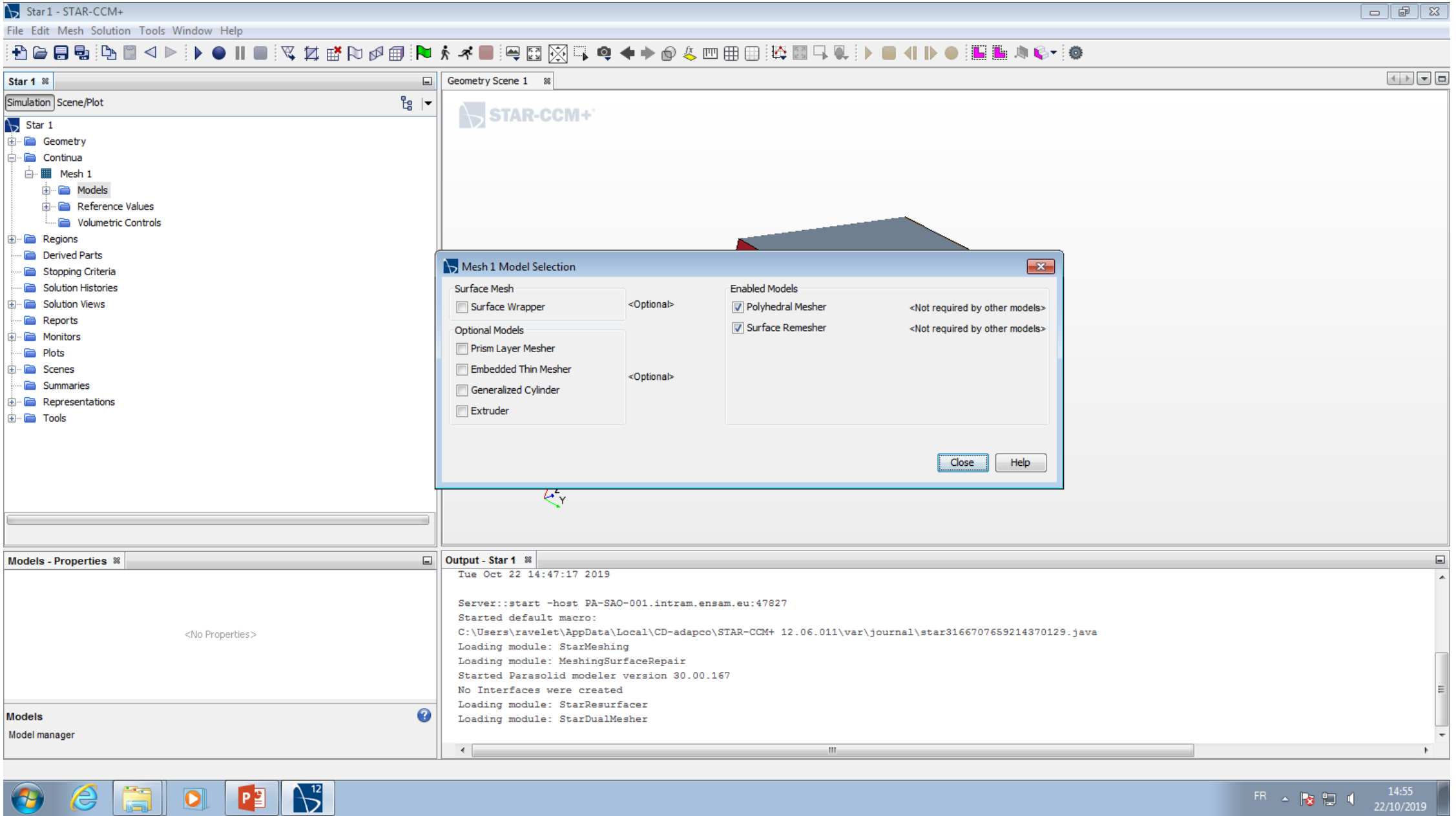
Adpat the values to your case



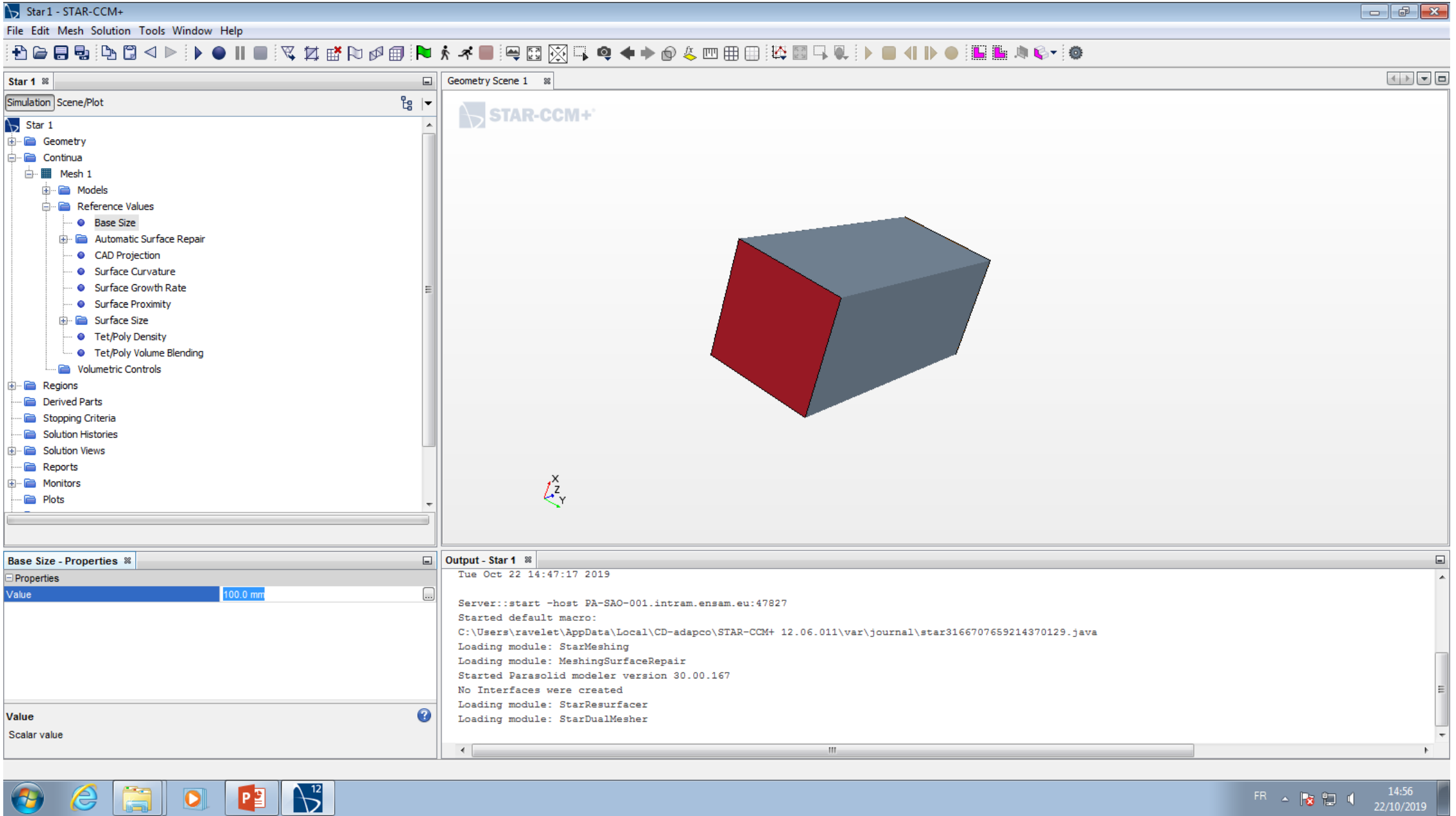
Create a « Mesh Continuum »



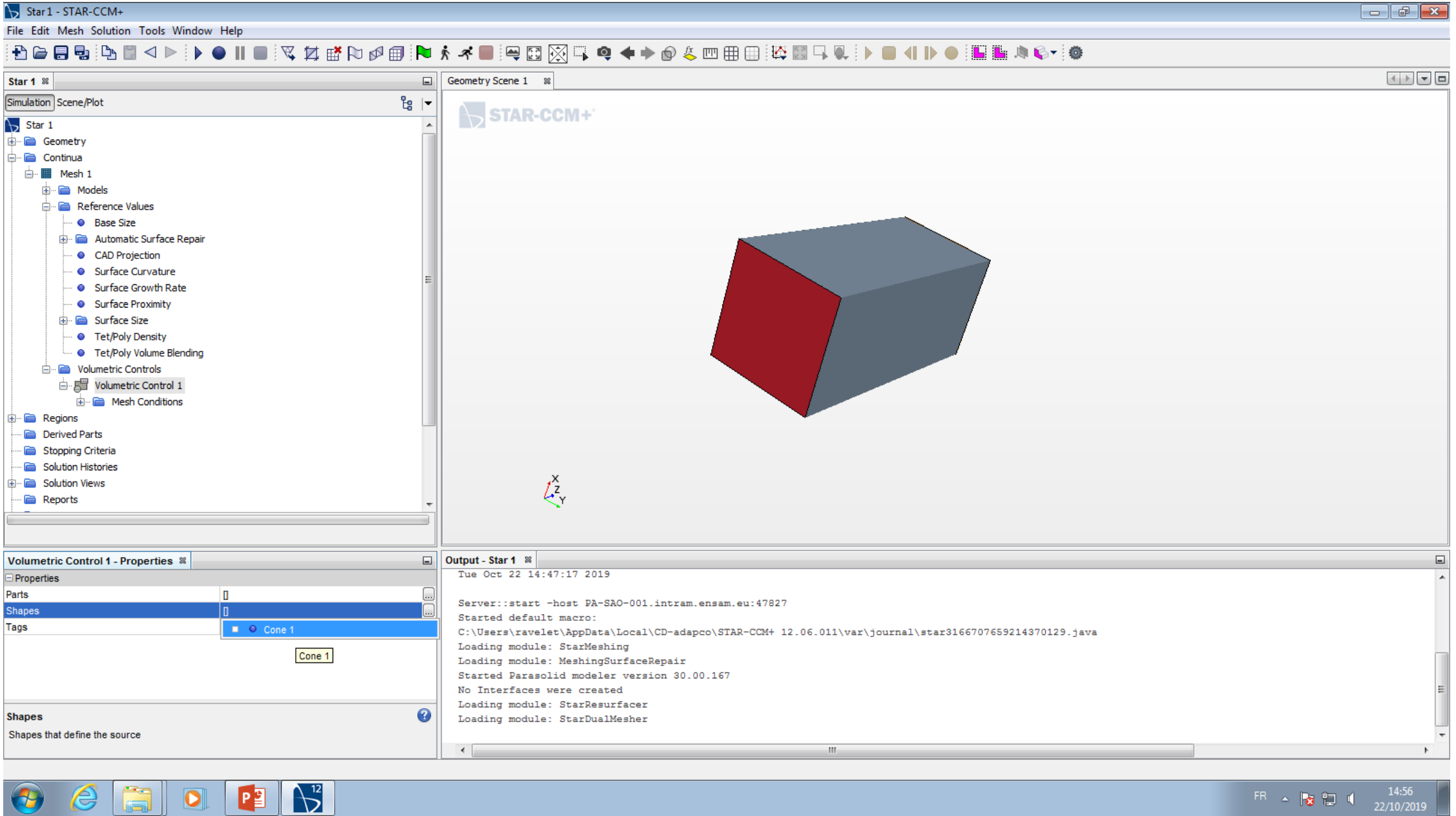
Select the mesh models



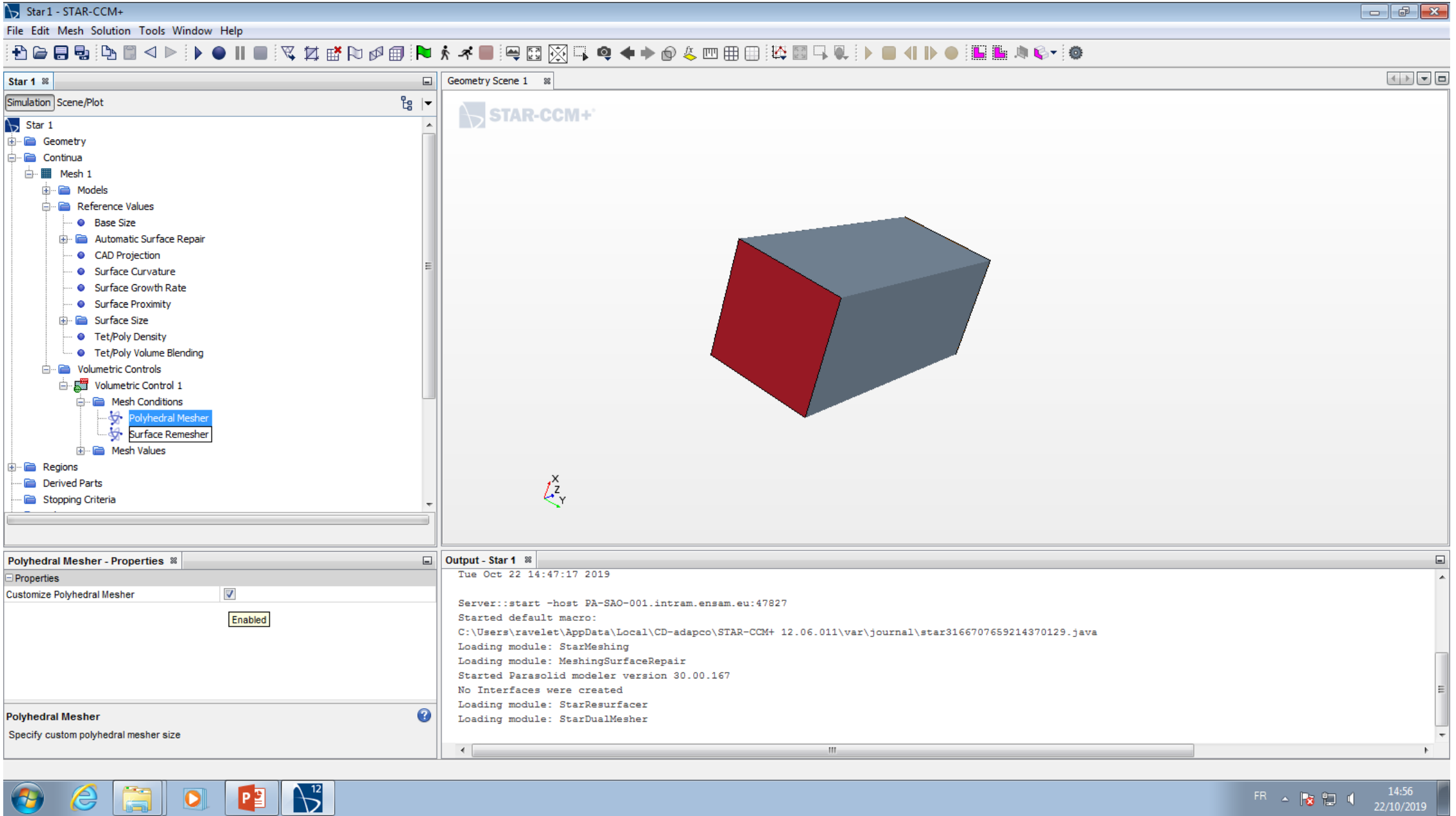
Change the base size: adpat the value to your case



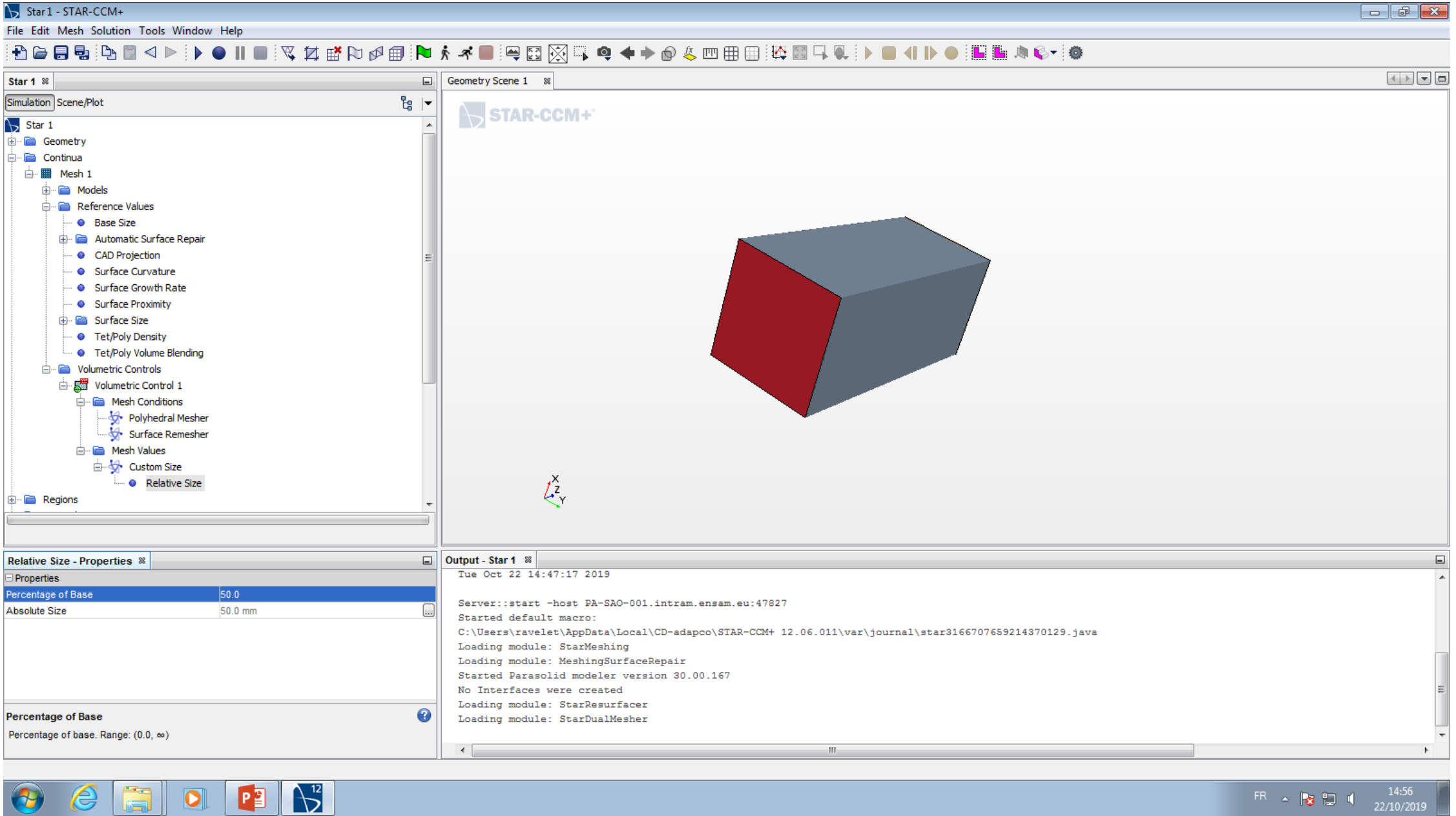
Use a volumetric control in the cone shape previously defined



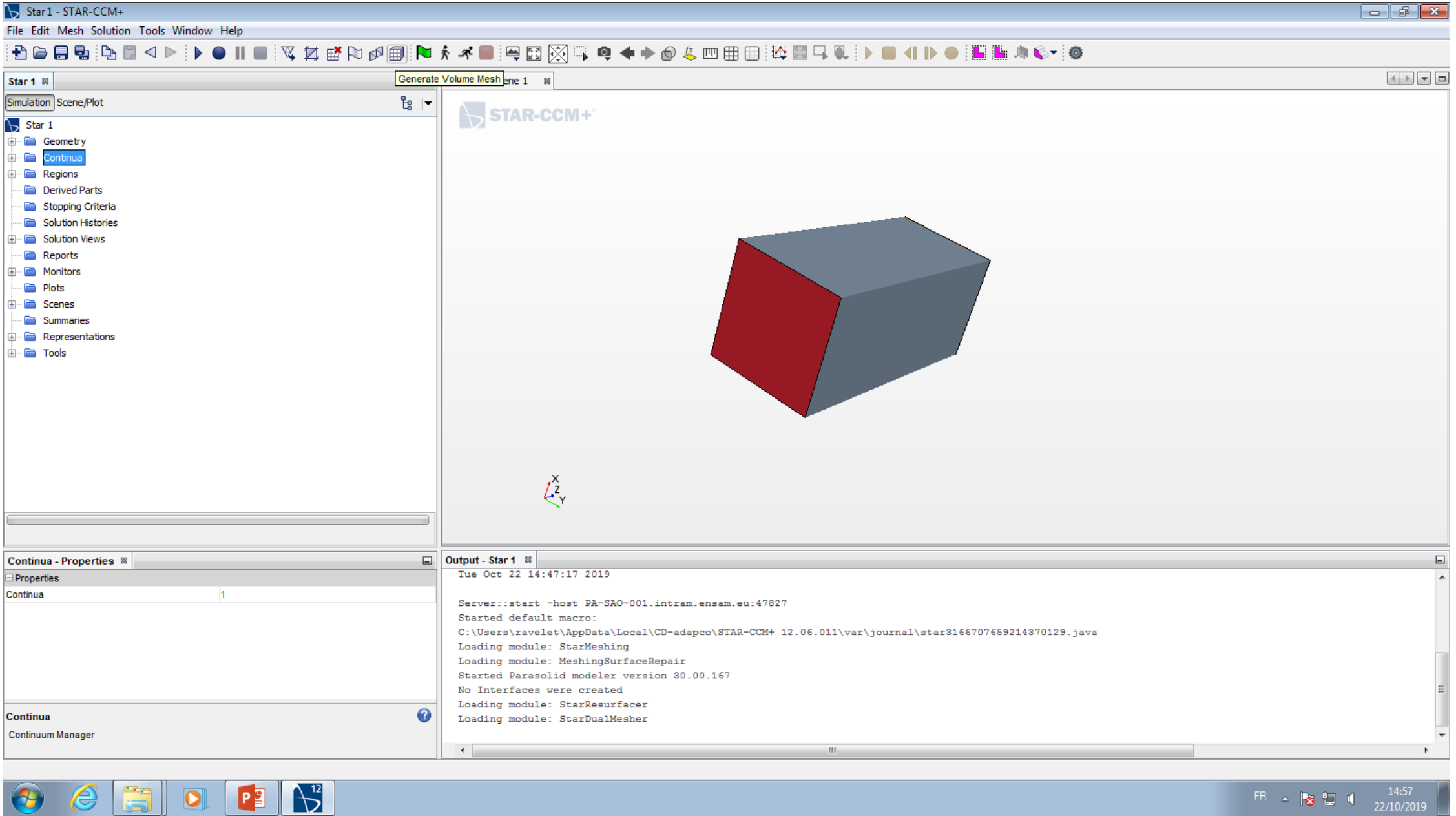
Customize the « Surface Remesher » and the « Polyhedral Mesher »



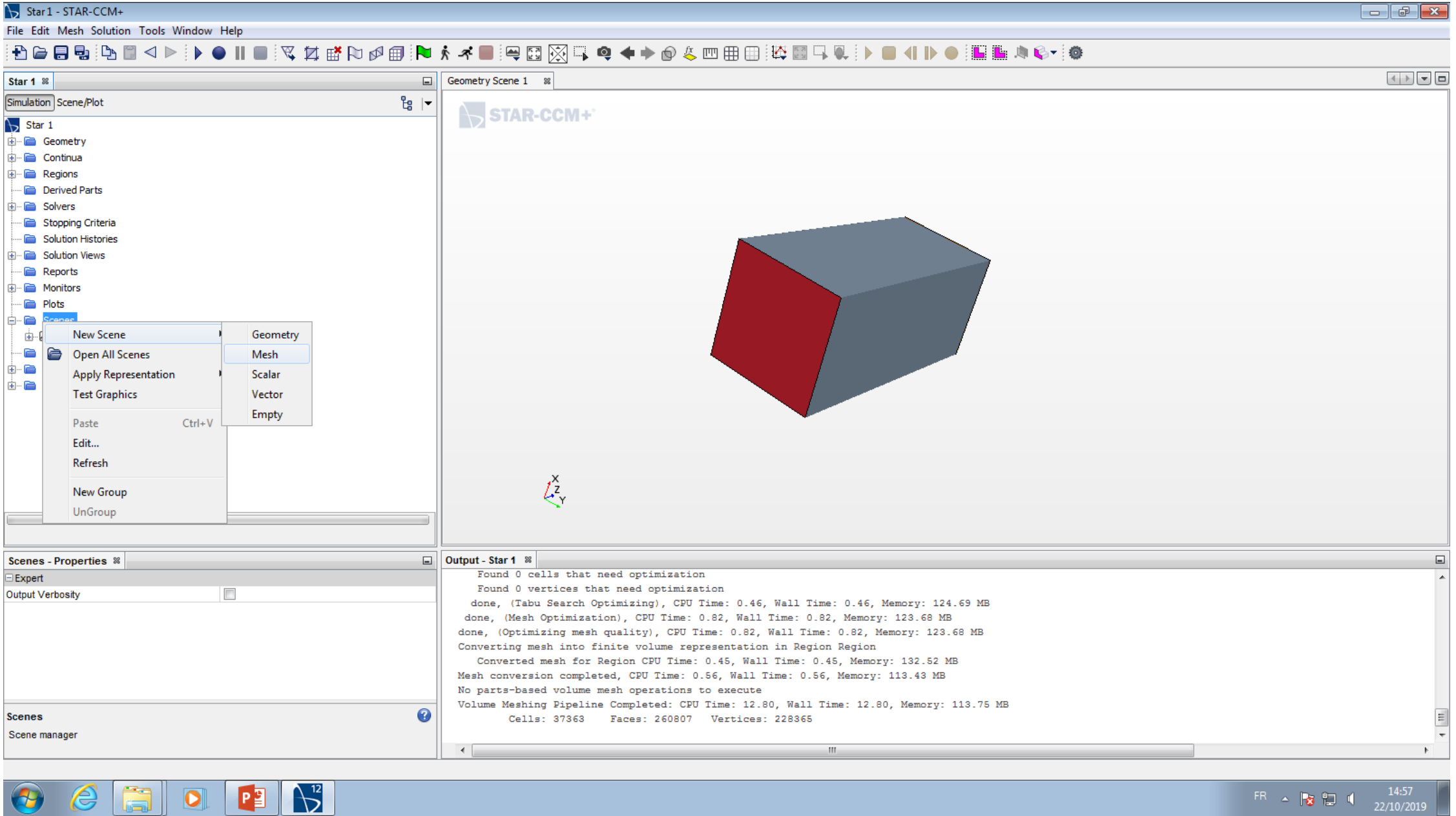
Customize the size: adapt it to your case

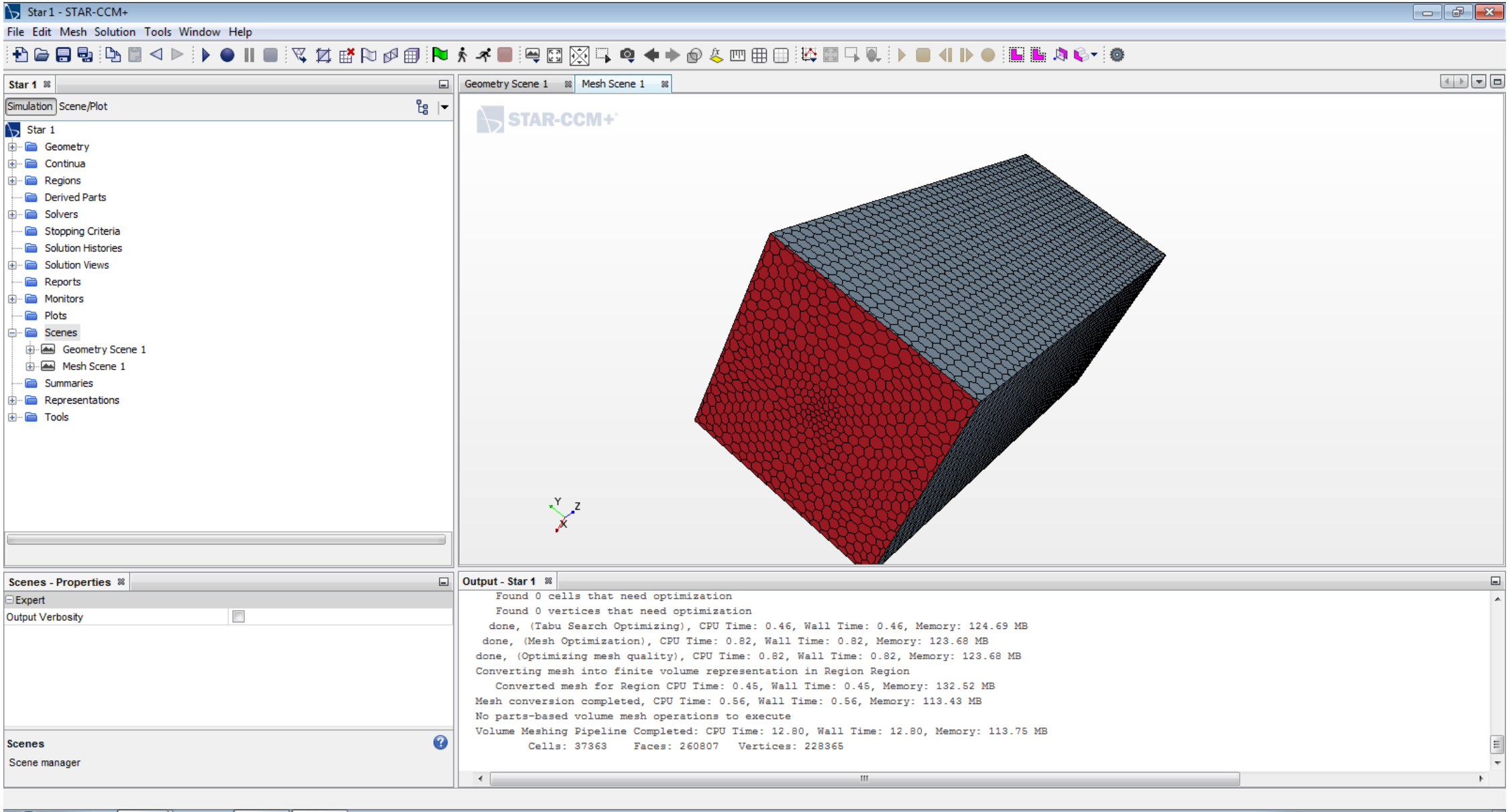


Generate the mesh

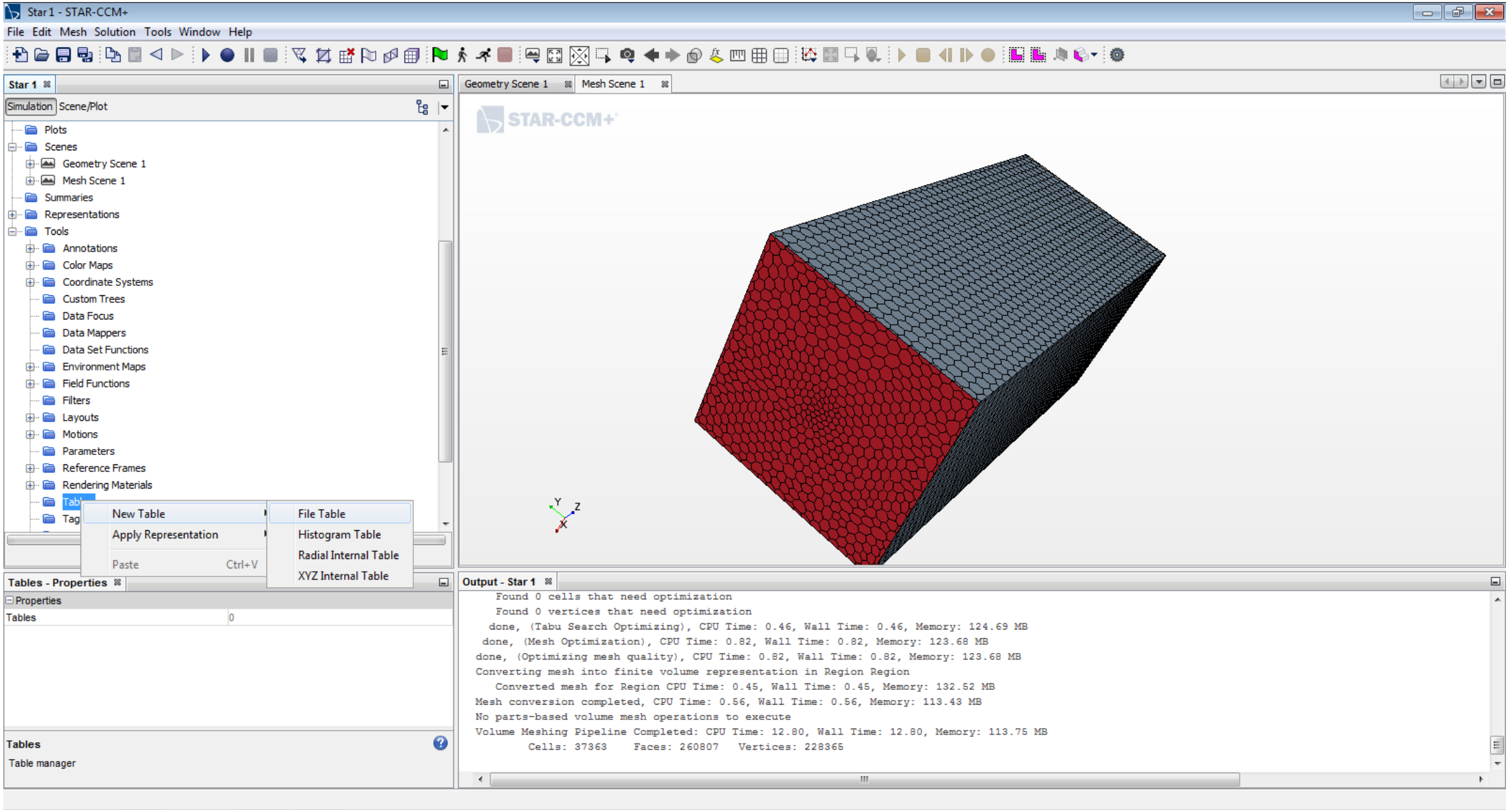


Visualize the mesh (a new « mesh scene »)

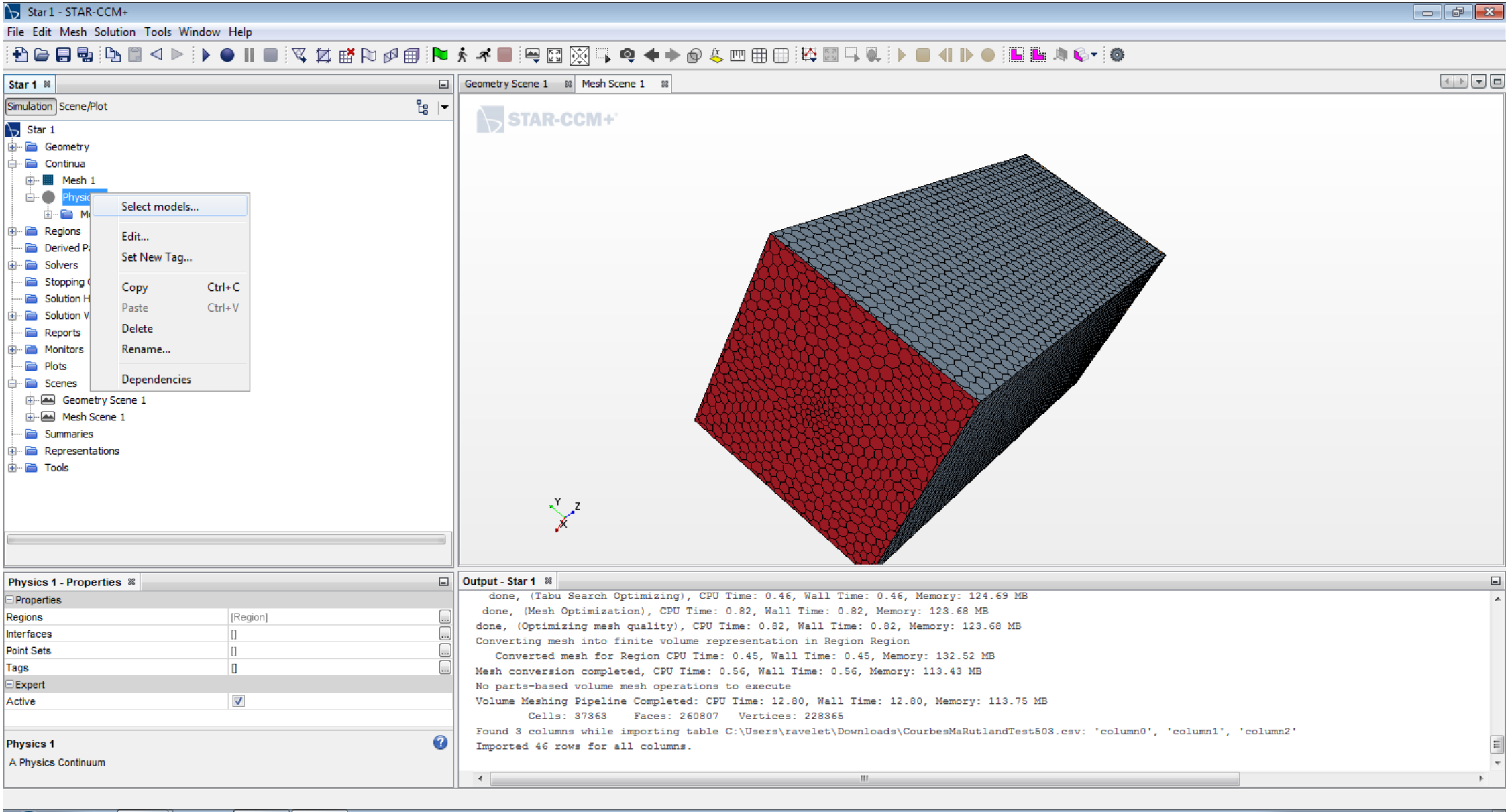


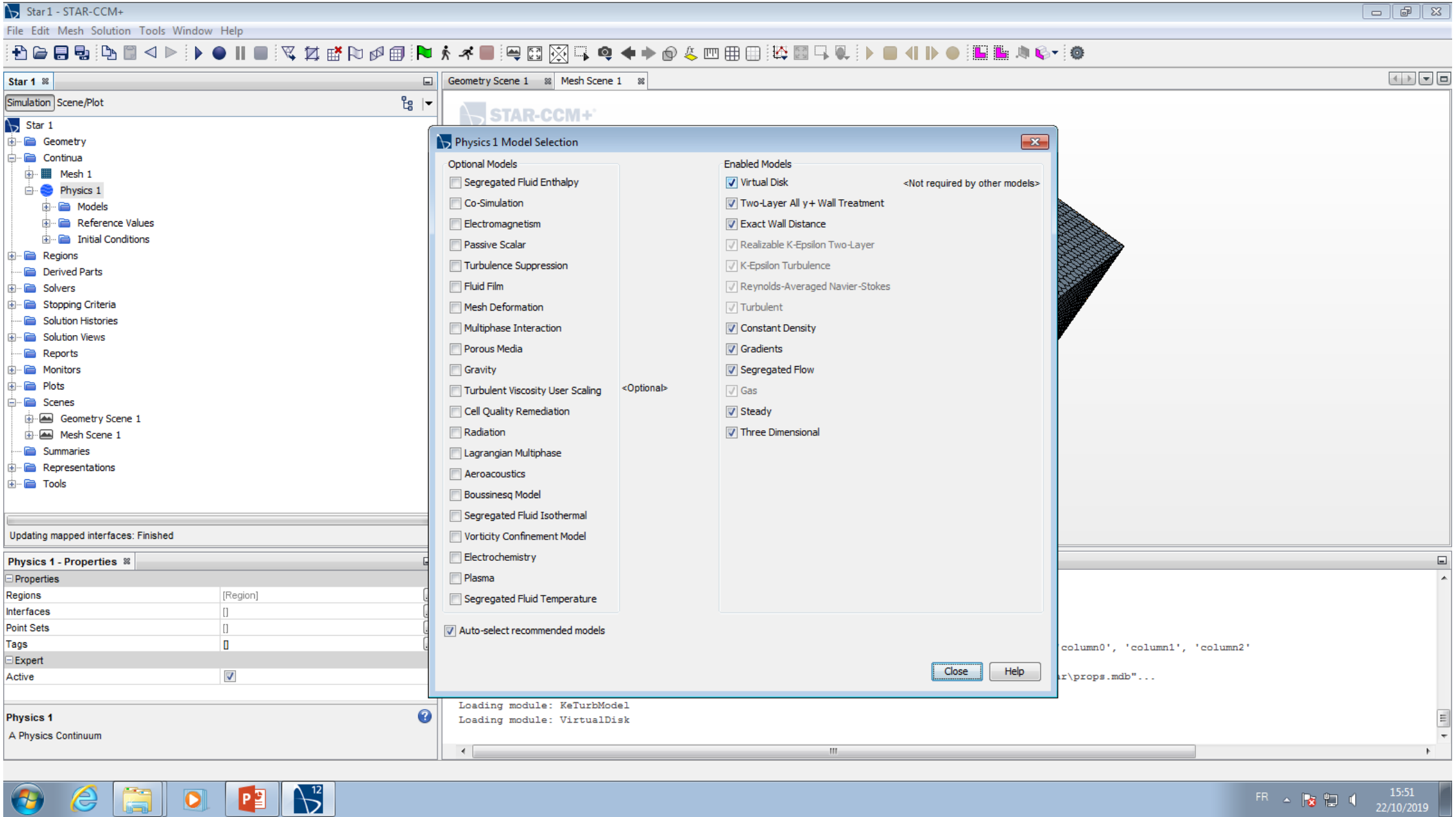

Star 1 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Star 1
Simulation Scene/Plot
Star 1
Geometry
Continua
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Mesh Scene 1
Summaries
Representations
Tools
Geometry Scene 1
Mesh Scene 1
STAR-CCM+
Y
Z
X
Scenes - Properties
Expert
Output Verbosity
Scenes
Scene manager
Output - Star 1
Found 0 cells that need optimization
Found 0 vertices that need optimization
done, (Tabu Search Optimizing), CPU Time: 0.46, Wall Time: 0.46, Memory: 124.69 MB
done, (Mesh Optimization), CPU Time: 0.82, Wall Time: 0.82, Memory: 123.68 MB
done, (Optimizing mesh quality), CPU Time: 0.82, Wall Time: 0.82, Memory: 123.68 MB
Converting mesh into finite volume representation in Region Region
Converted mesh for Region CPU Time: 0.45, Wall Time: 0.45, Memory: 132.52 MB
Mesh conversion completed, CPU Time: 0.56, Wall Time: 0.56, Memory: 113.43 MB
No parts-based volume mesh operations to execute
Volume Meshing Pipeline Completed: CPU Time: 12.80, Wall Time: 12.80, Memory: 113.75 MB
Cells: 37363 Faces: 260807 Vertices: 228365
FR
14:58
22/10/2019

Import the table that contains the Wind Turbine characteristics

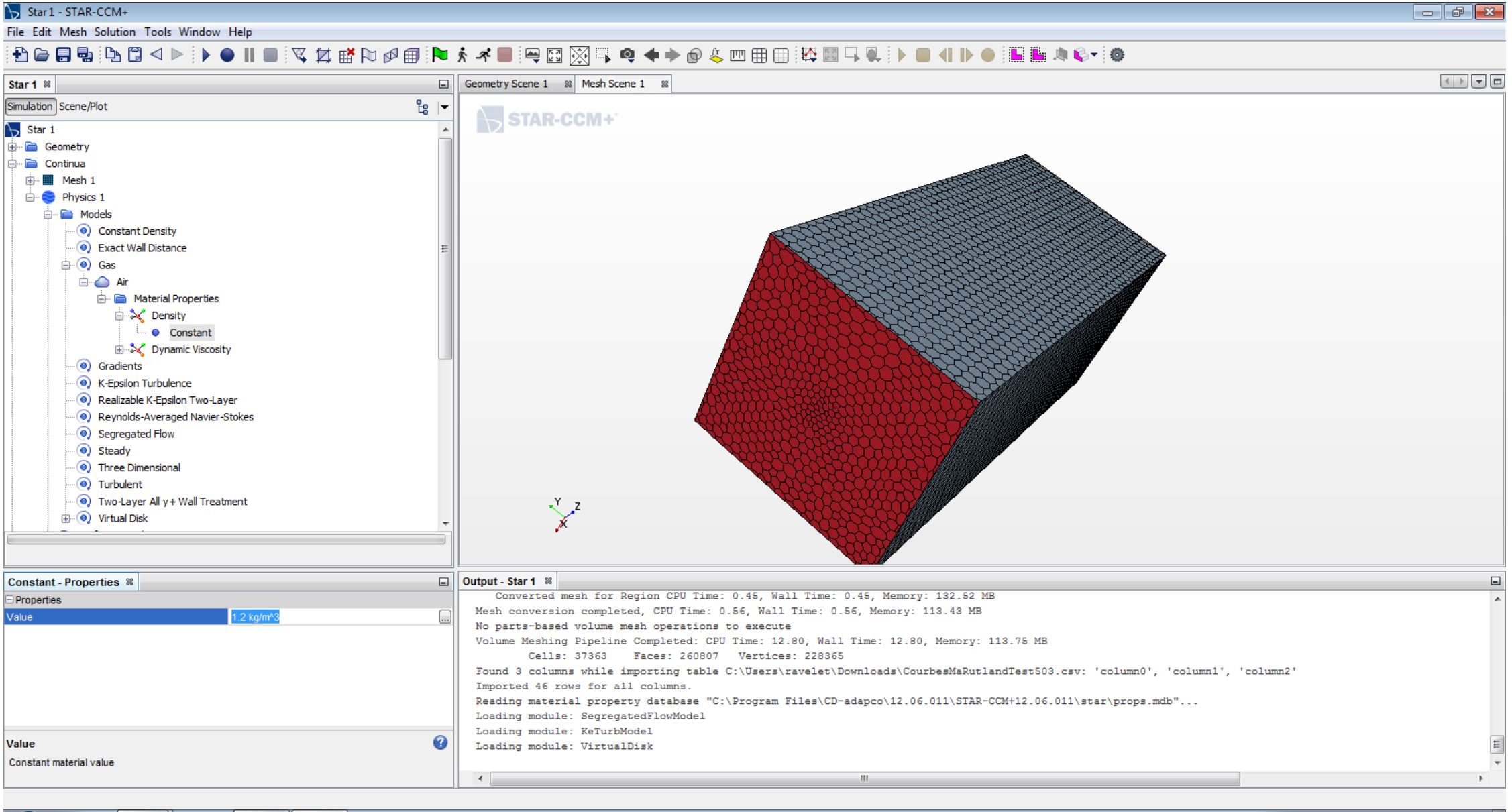


Create a « Physics Continuum »

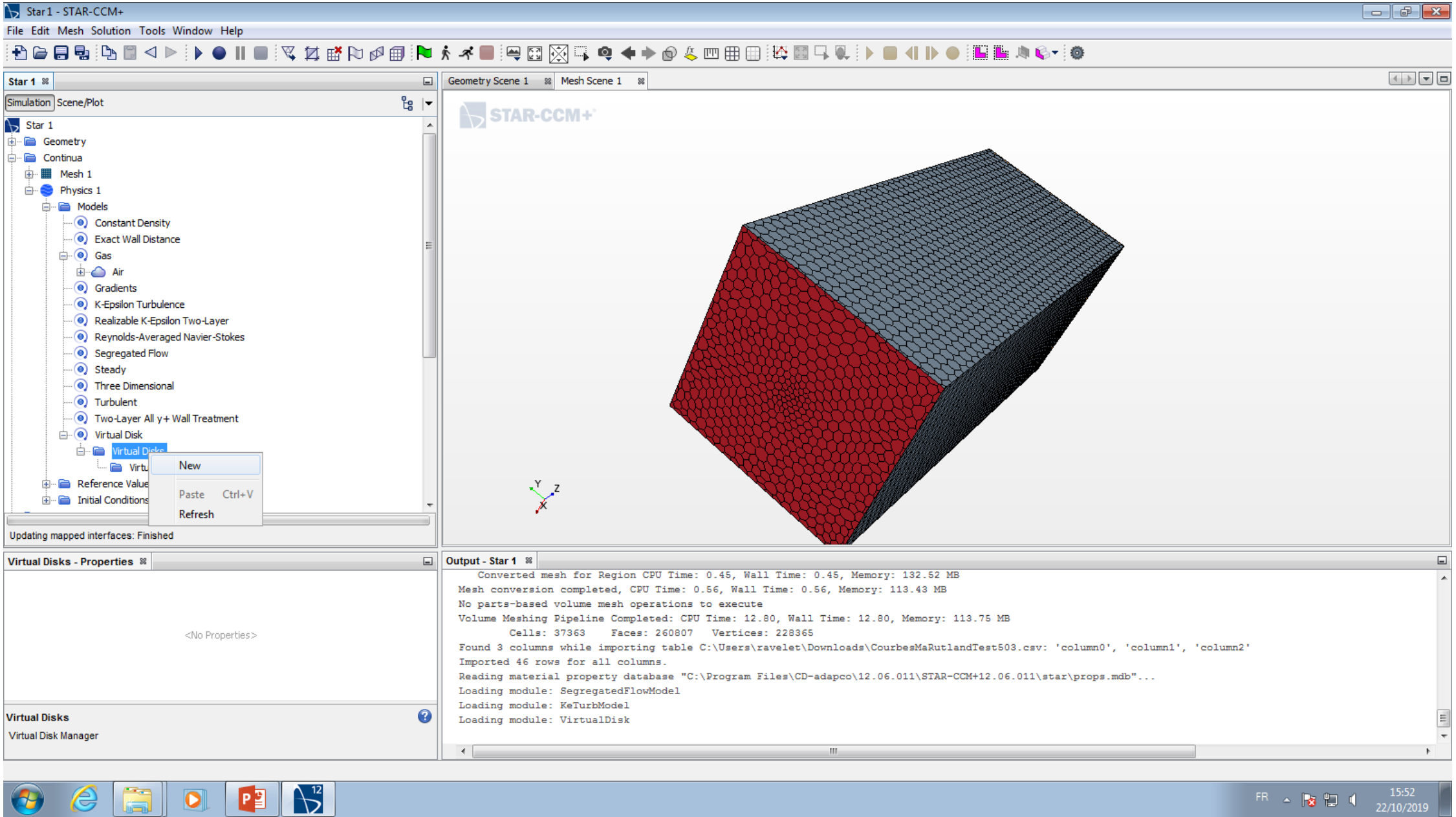



Star 1 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Star 1
Simulation Scene/Plot
Star 1
Geometry
Continua
Mesh 1
Physics 1
Models
Reference Values
Initial Conditions
Regions
Derived Parts
Solvers
Stopping Criteria
Solution Histories
Solution Views
Reports
Monitors
Plots
Scenes
Geometry Scene 1
Mesh Scene 1
Summaries
Representations
Tools
Updating mapped interfaces: Finished
Physics 1 - Properties
Properties
Regions [Region]
Interfaces []
Point Sets []
Tags []
Expert
Active
Physics 1
A Physics Continuum
Geometry Scene 1
Mesh Scene 1
STAR-CCM+
Physics 1 Model Selection
Optional Models
Segregated Fluid Enthalpy
Co-Simulation
Electromagnetism
Passive Scalar
Turbulence Suppression
Fluid Film
Mesh Deformation
Multiphase Interaction
Porous Media
Gravity
Turbulent Viscosity User Scaling
Cell Quality Remediation
Radiation
Lagrangian Multiphase
Aeroacoustics
Boussinesq Model
Segregated Fluid Isothermal
Vorticity Confinement Model
Electrochemistry
Plasma
Segregated Fluid Temperature
<Optional>
Enabled Models
Virtual Disk
<Not required by other models>
Two-Layer All y+ Wall Treatment
Exact Wall Distance
Realizable K-Epsilon Two-Layer
K-Epsilon Turbulence
Reynolds-Averaged Navier-Stokes
Turbulent
Constant Density
Gradients
Segregated Flow
Gas
Steady
Three Dimensional
Auto-select recommended models
Close
Help
column0', 'column1', 'column2'
r\props.mdb"...
Loading module: KeTurbModel
Loading module: VirtualDisk
FR
15:51
22/10/2019

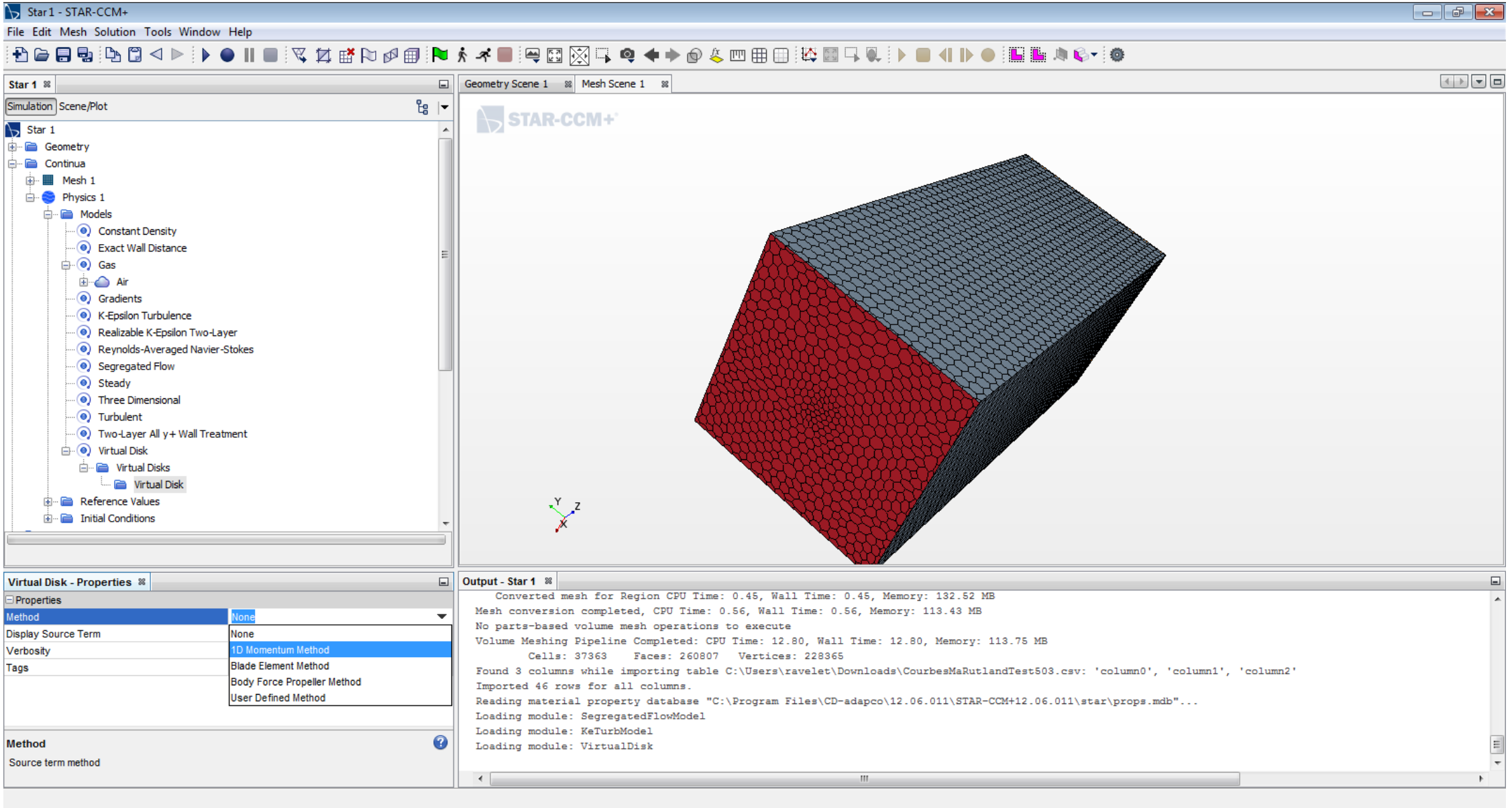
Control the properties of the fluid



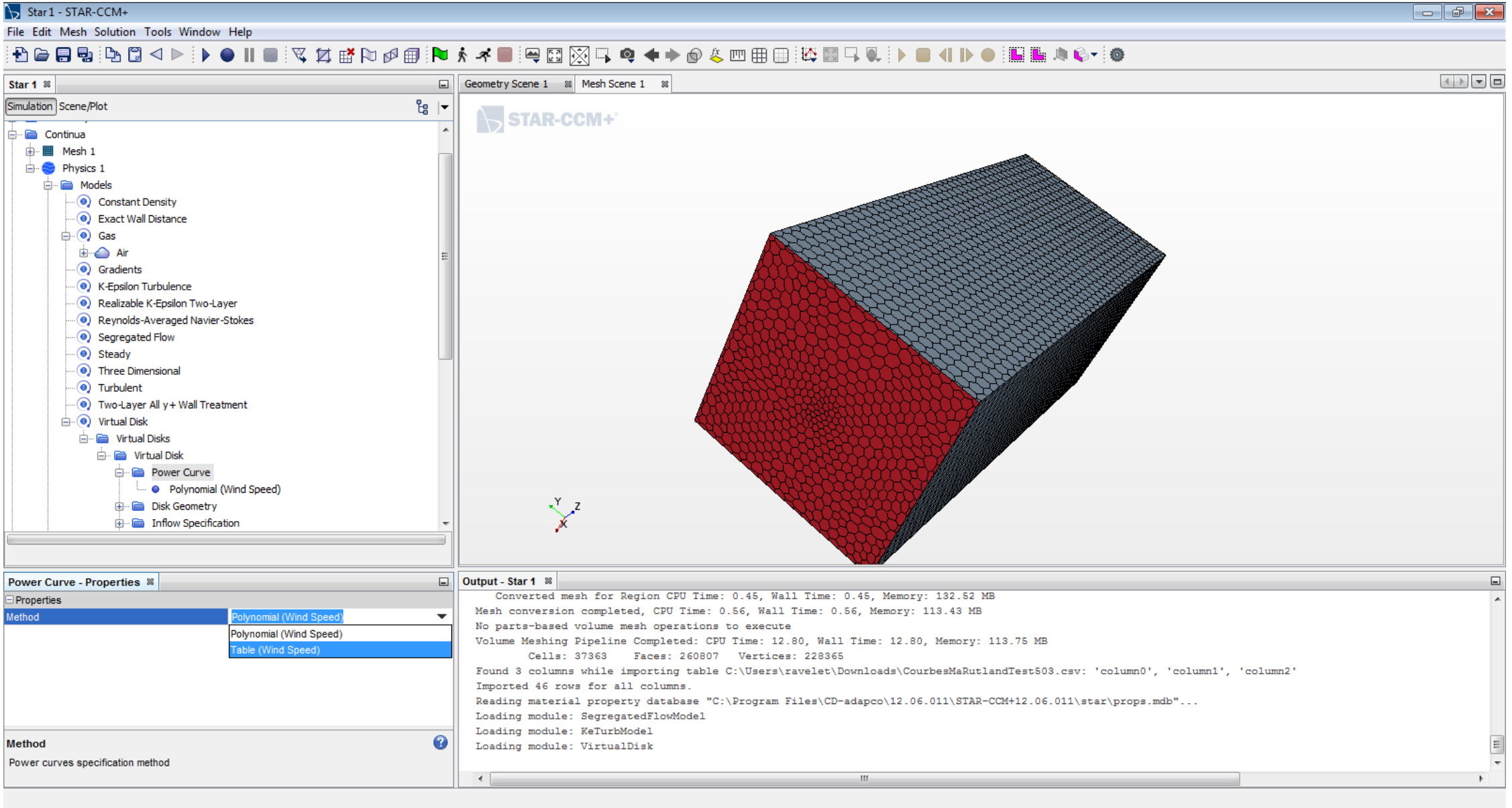
Create a new « Virtual Disk »

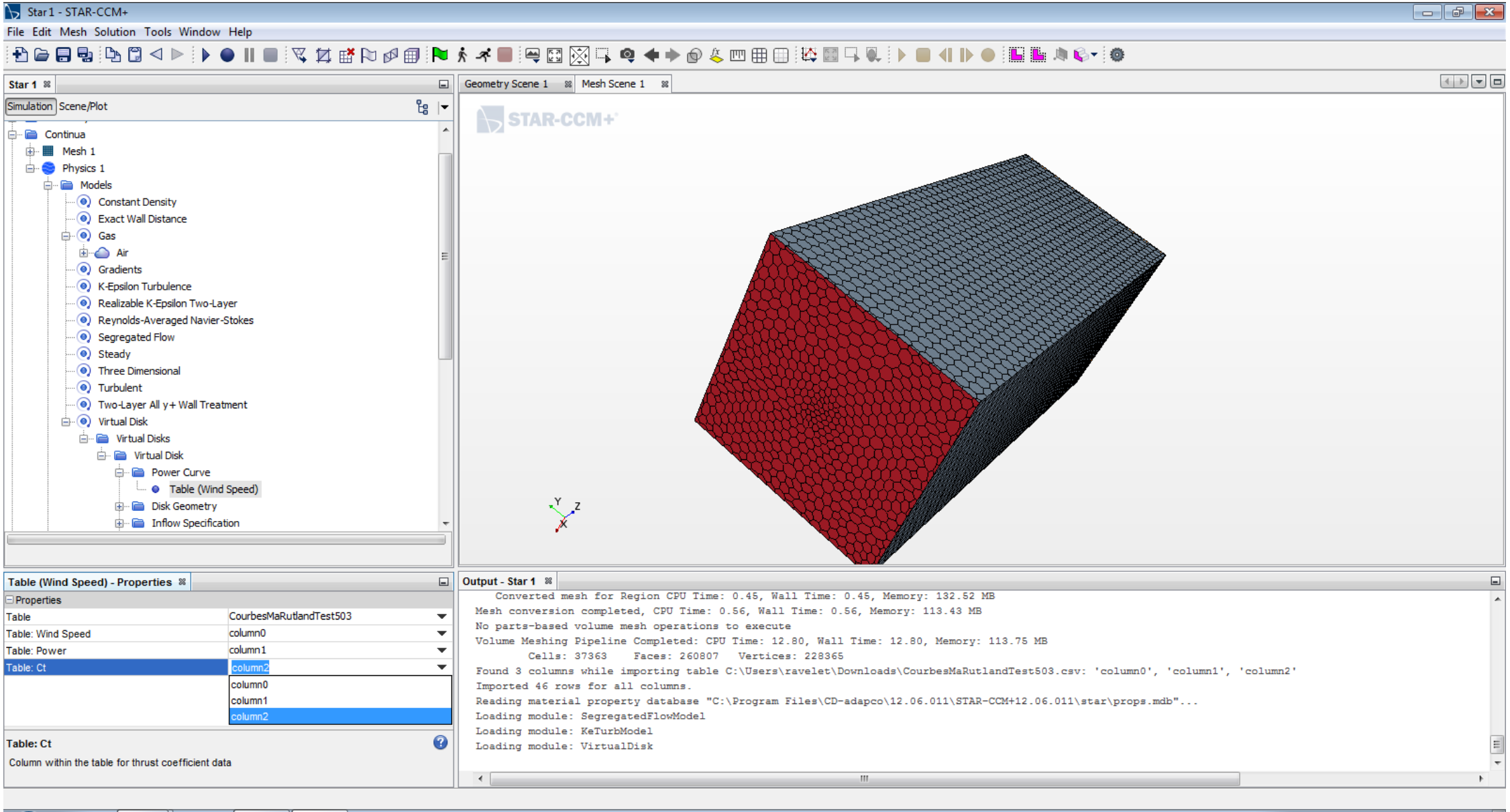


Select the model

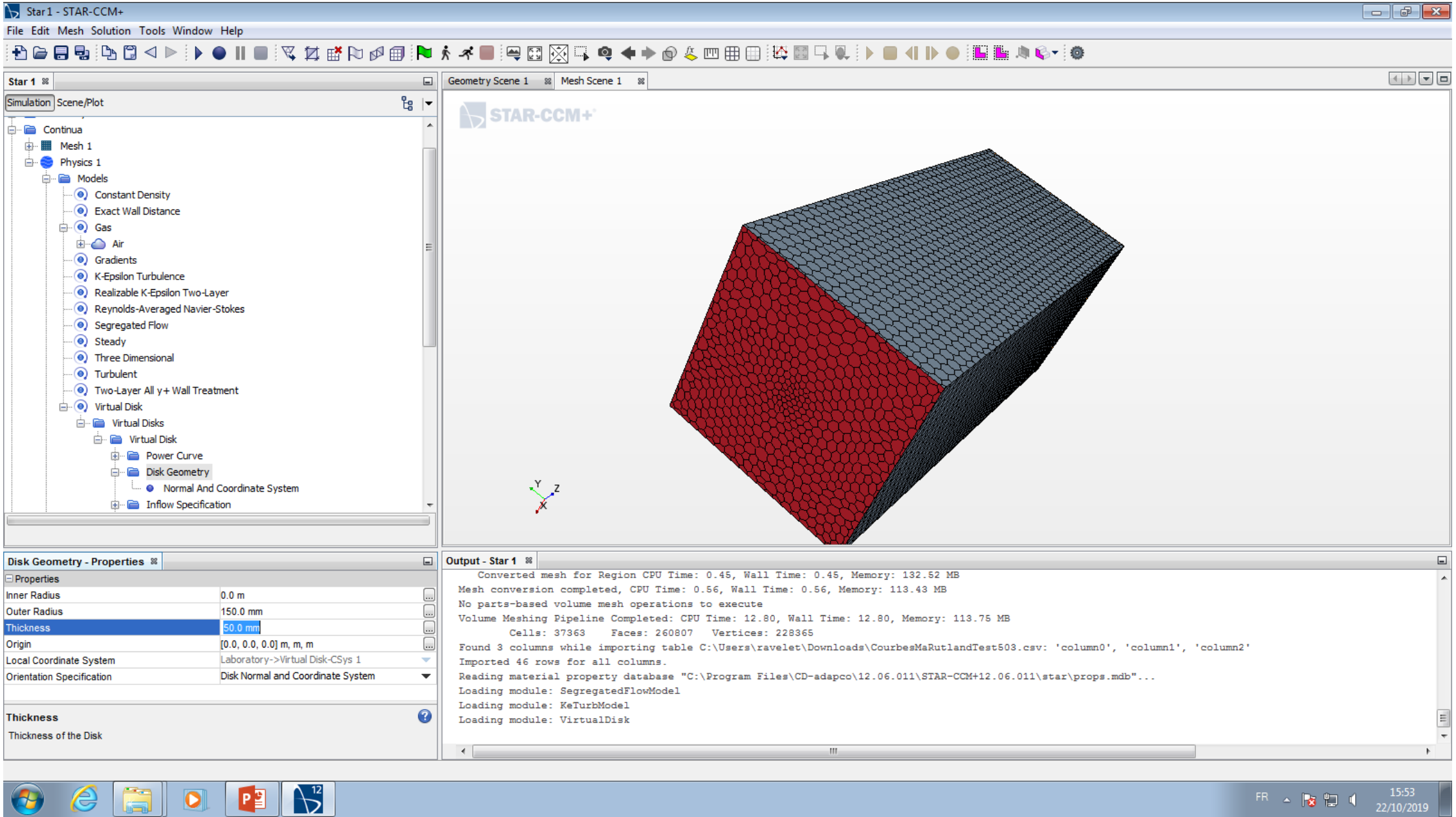


Use the table

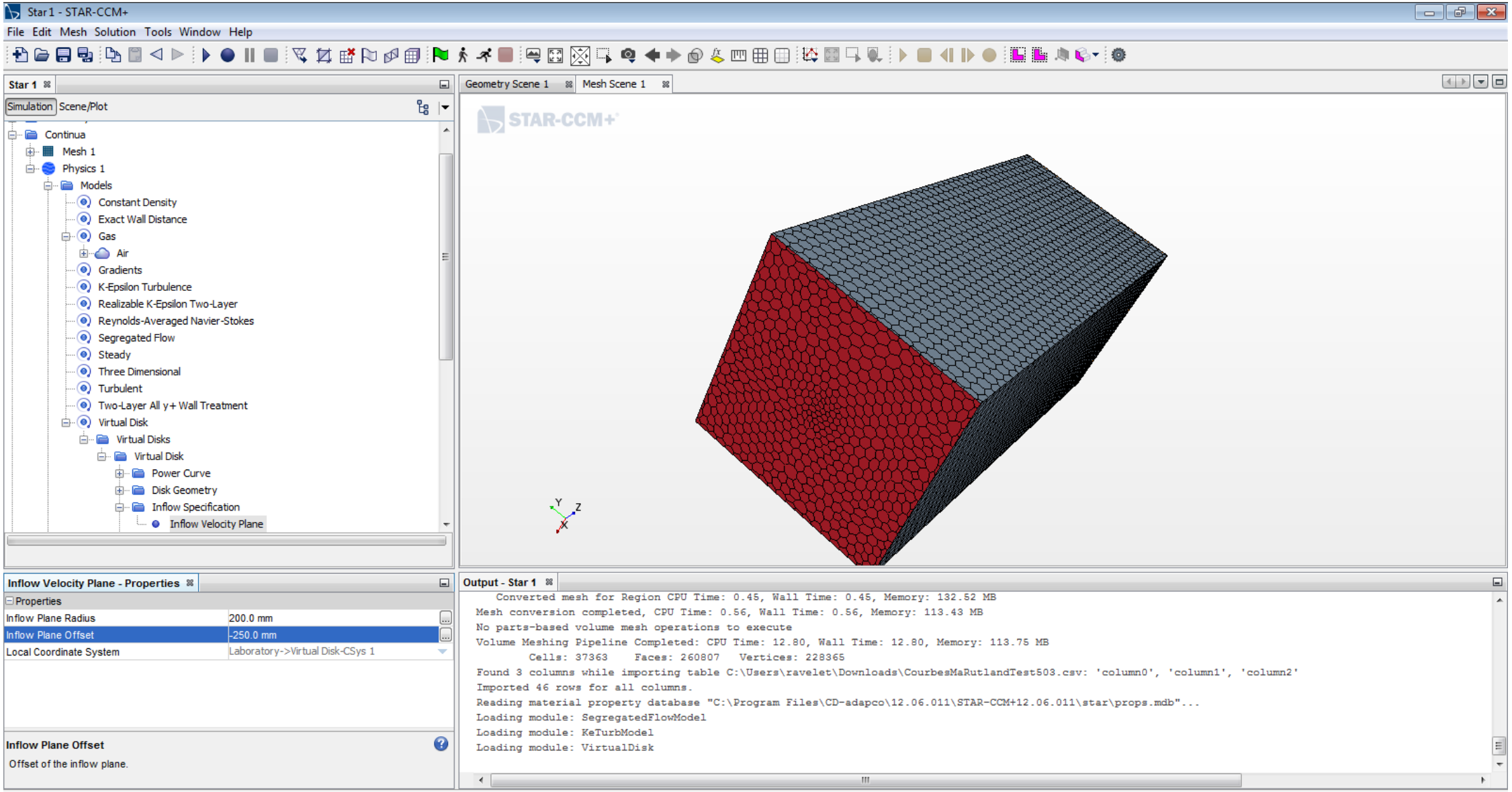



Star 1 - STAR-CCM+
File Edit Mesh Solution Tools Window Help
Star 1
Simulation Scene/Plot
Continua
Mesh 1
Physics 1
Models
Constant Density
Exact Wall Distance
Gas
Air
Gradients
K-Epsilon Turbulence
Realizable K-Epsilon Two-Layer
Reynolds-Averaged Navier-Stokes
Segregated Flow
Steady
Three Dimensional
Turbulent
Two-Layer All y+ Wall Treatment
Virtual Disk
Virtual Disks
Virtual Disk
Power Curve
Table (Wind Speed)
Disk Geometry
Inflow Specification
Geometry Scene 1
Mesh Scene 1
STAR-CCM+
Y
Z
X
Table (Wind Speed) - Properties
Properties
Table
CourbesMaRutlandTest503
Table: Wind Speed
column0
Table: Power
column1
Table: Ct
column2
column0
column1
column2
Table: Ct
Column within the table for thrust coefficient data
Output - Star 1
Converted mesh for Region CPU Time: 0.45, Wall Time: 0.45, Memory: 132.52 MB
Mesh conversion completed, CPU Time: 0.56, Wall Time: 0.56, Memory: 113.43 MB
No parts-based volume mesh operations to execute
Volume Meshing Pipeline Completed: CPU Time: 12.80, Wall Time: 12.80, Memory: 113.75 MB
Cells: 37363 Faces: 260807 Vertices: 228365
Found 3 columns while importing table C:\Users\ravelet\Downloads\CourbesMaRutlandTest503.csv: 'column0', 'column1', 'column2'
Imported 46 rows for all columns.
Reading material property database "C:\Program Files\CD-adapco\12.06.011\STAR-CCM+12.06.011\star\props.mdb"...
Loading module: SegregatedFlowModel
Loading module: KeTurbModel
Loading module: VirtualDisk
FR
15:53
22/10/2019

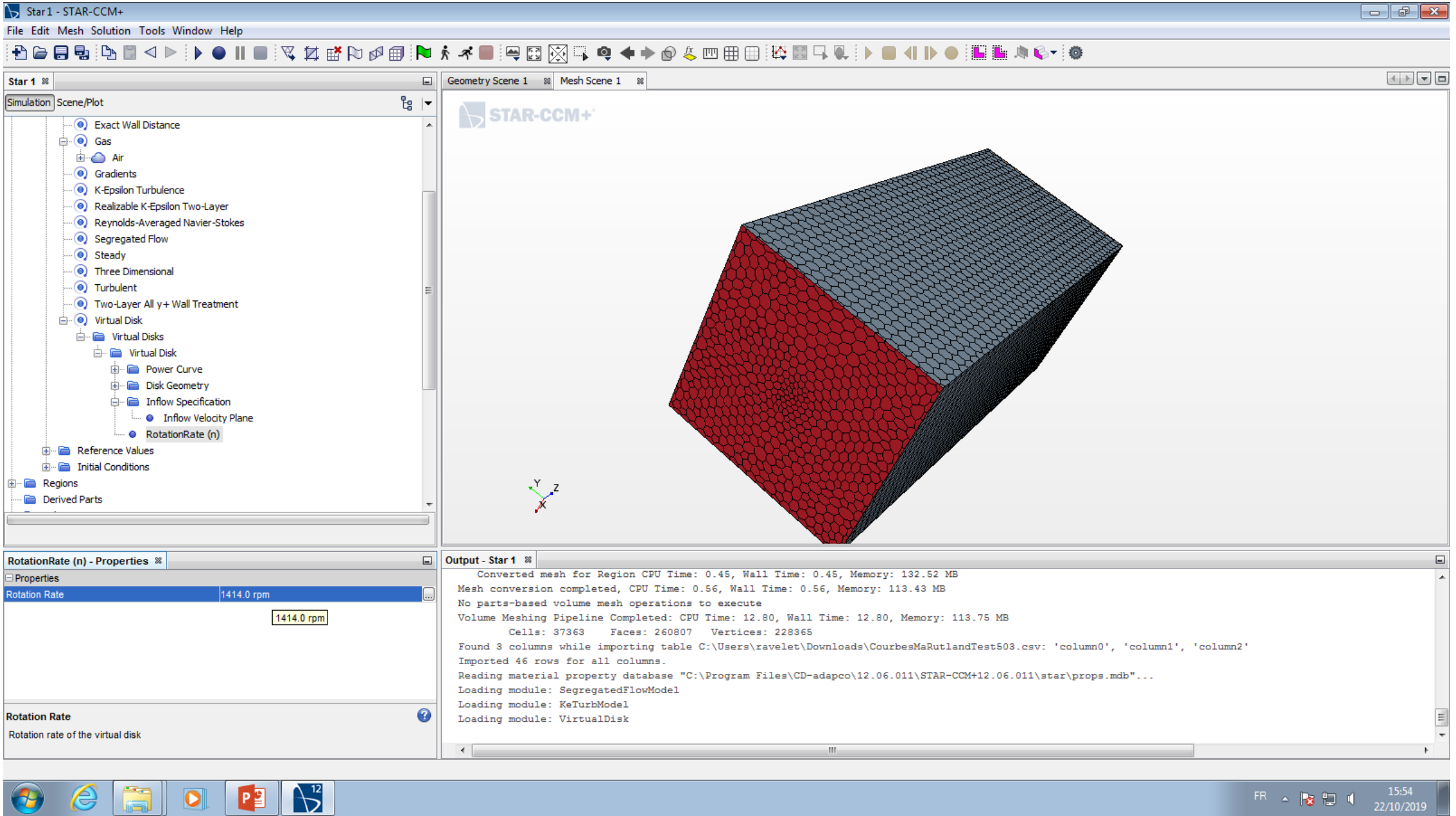
Specify the geometry of the disk



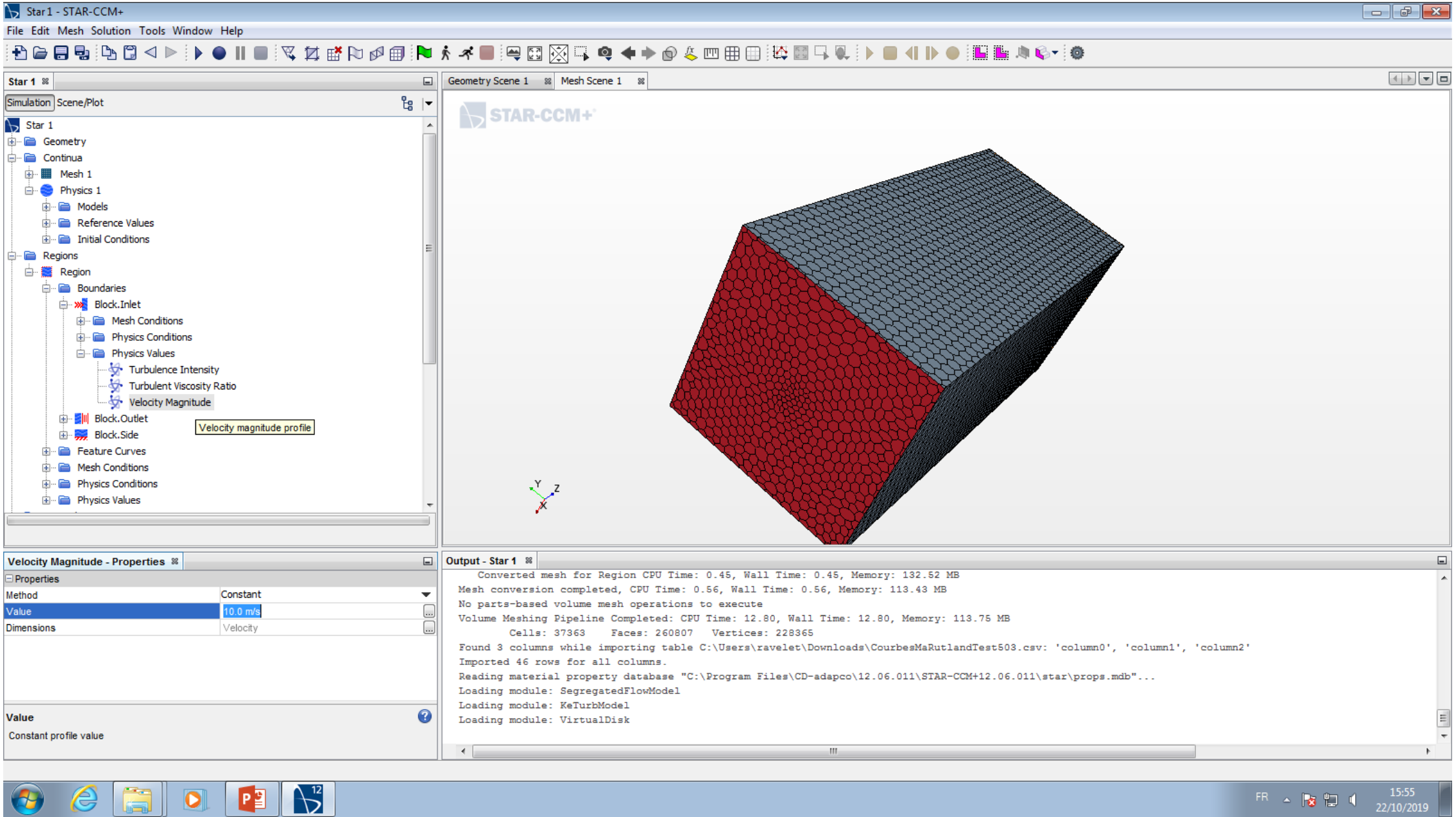
Choose a plane for the inlet reference velocity estimation



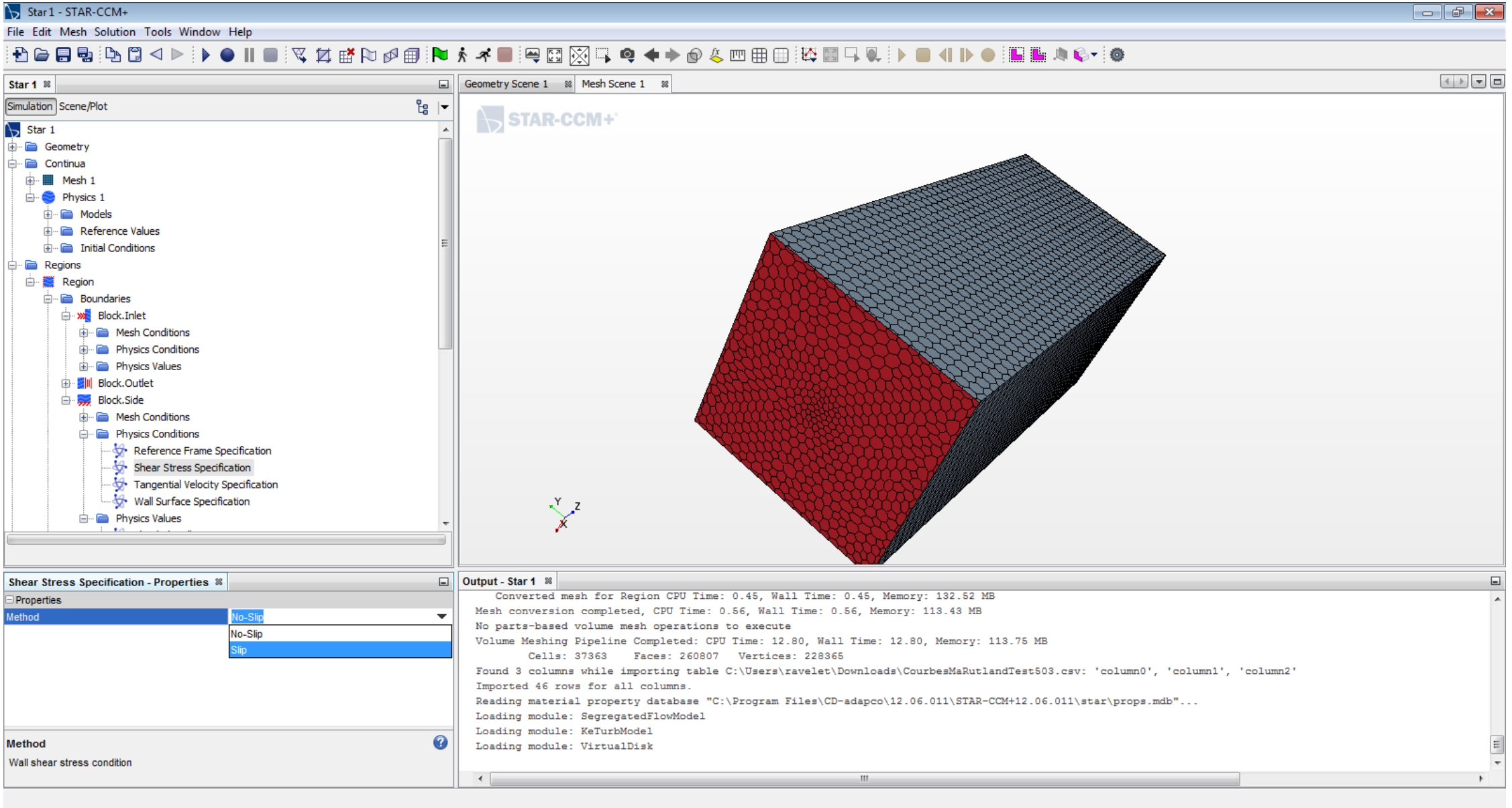
Specify the rotation rate that corresponds to the data in the table



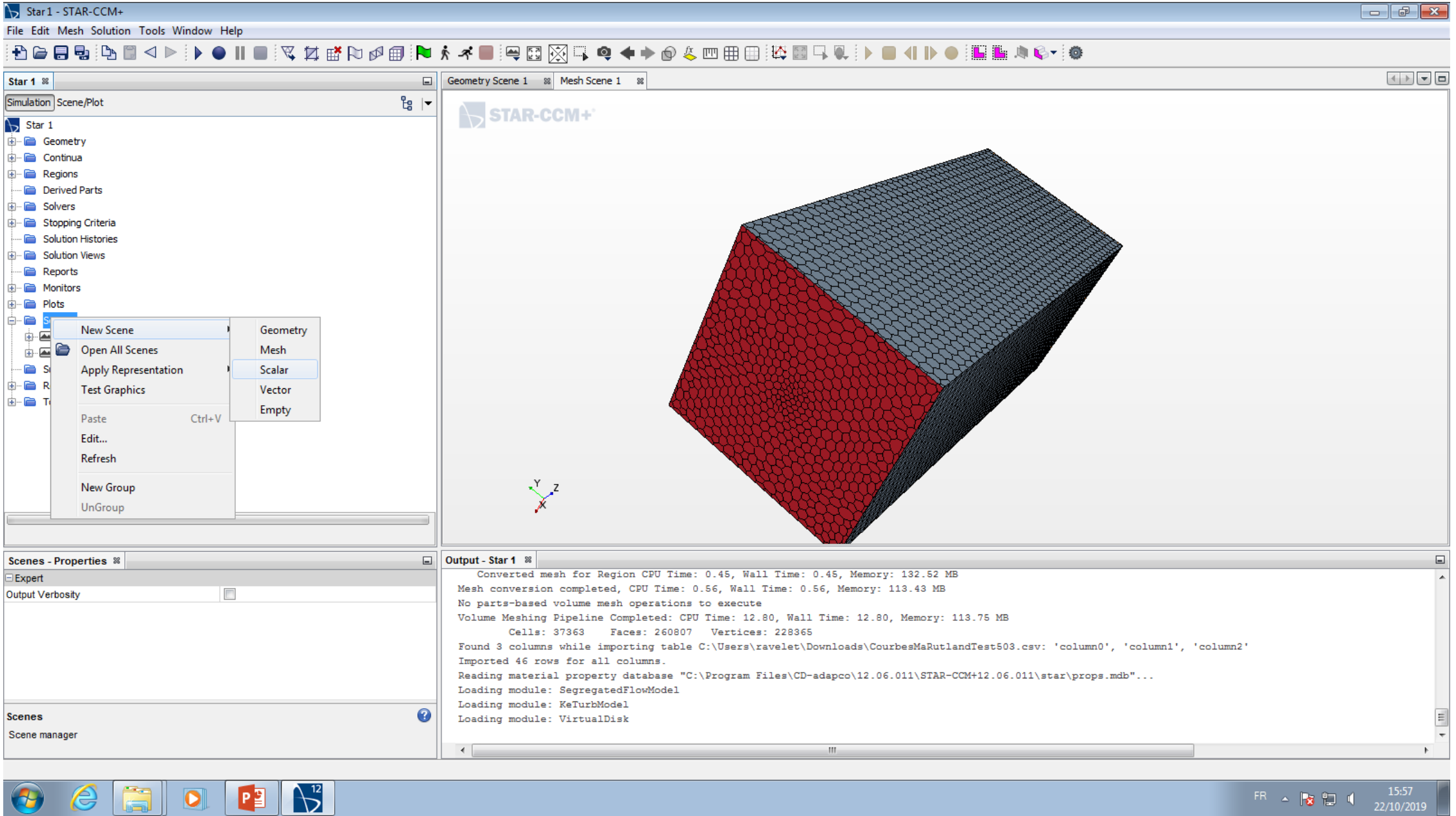
Change the inlet boundary condition



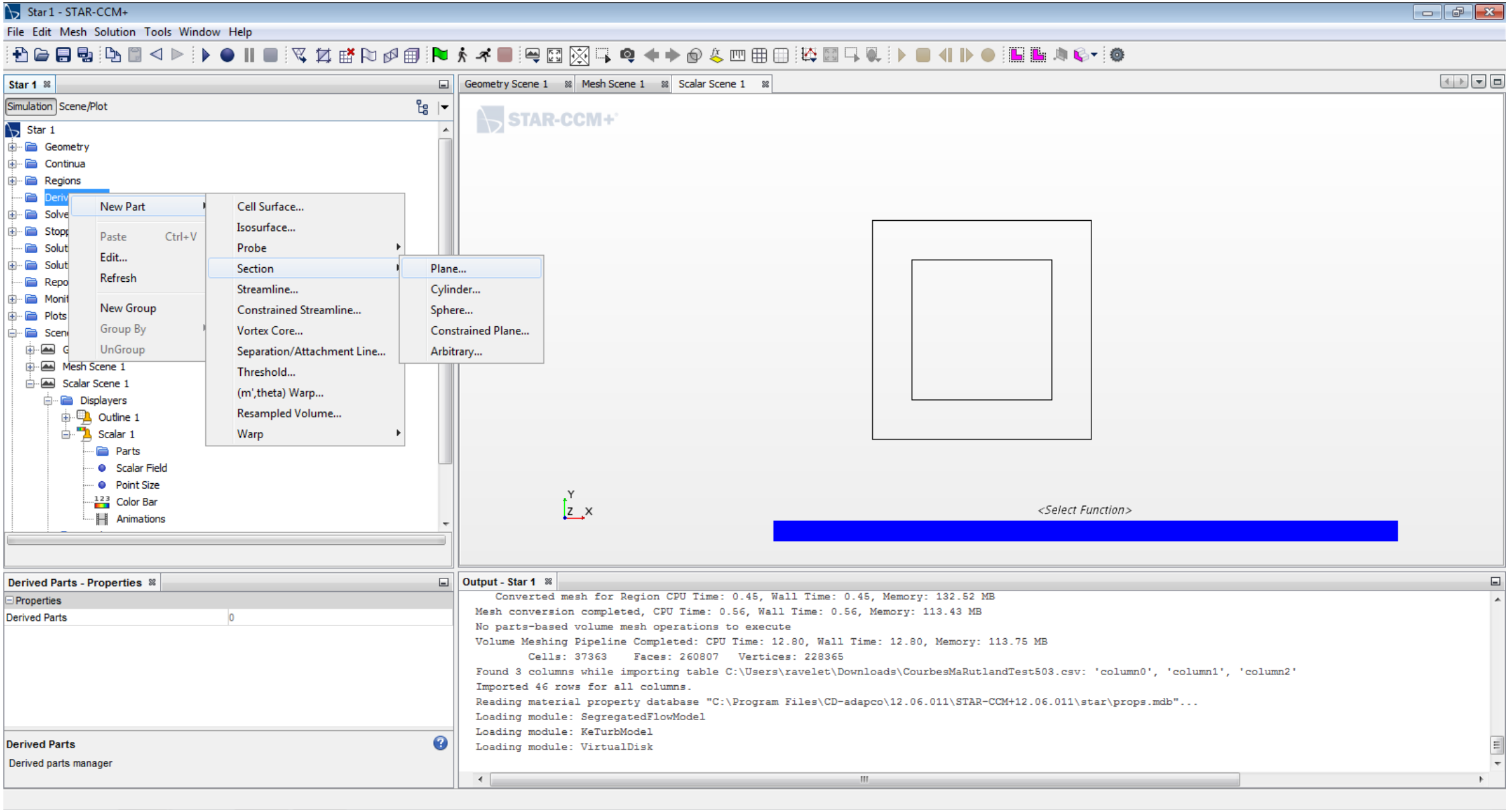
You can change the side wall boundary condition to slip



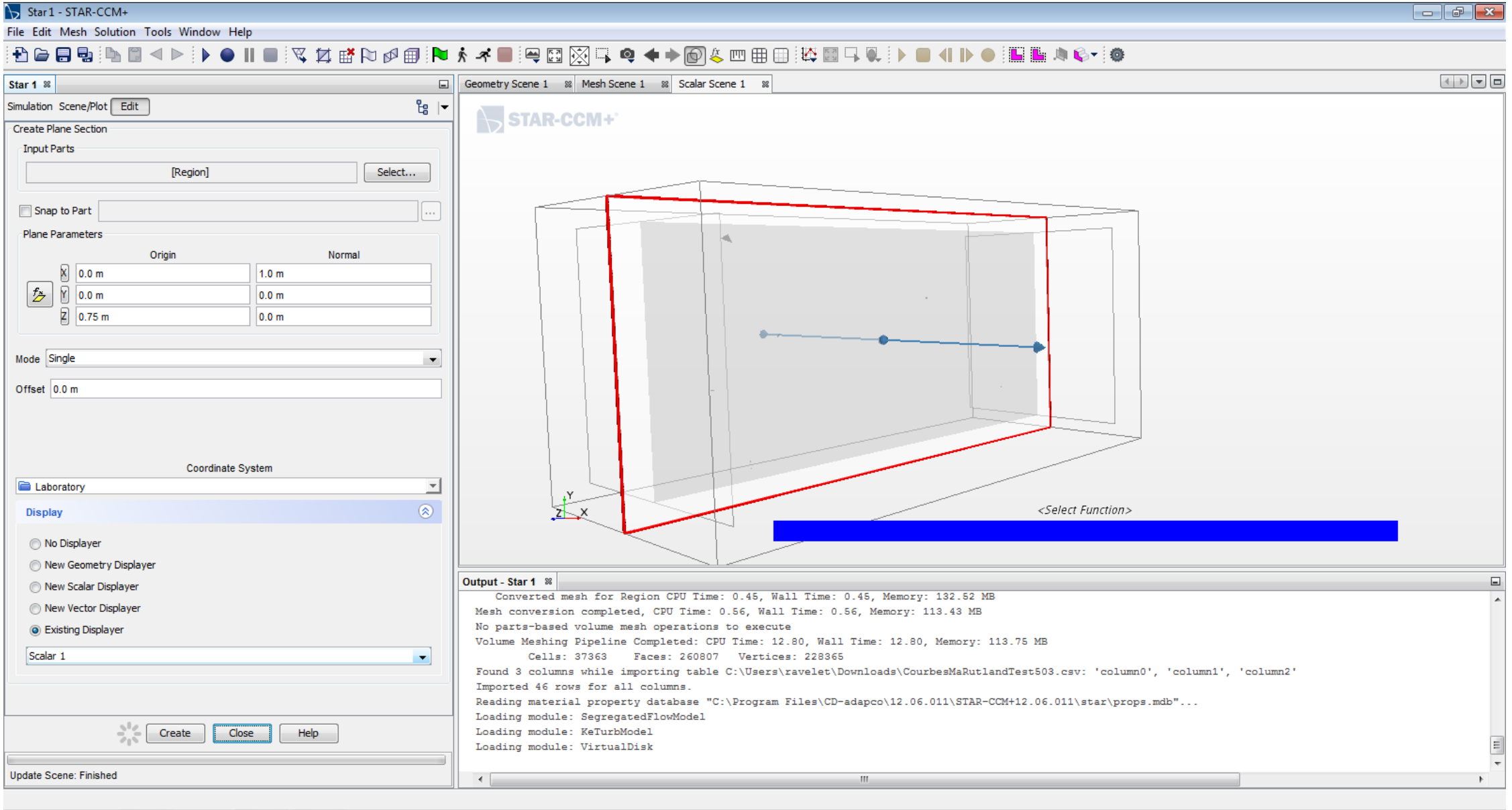
Prepare a scalar scene for visualization of the velocity field



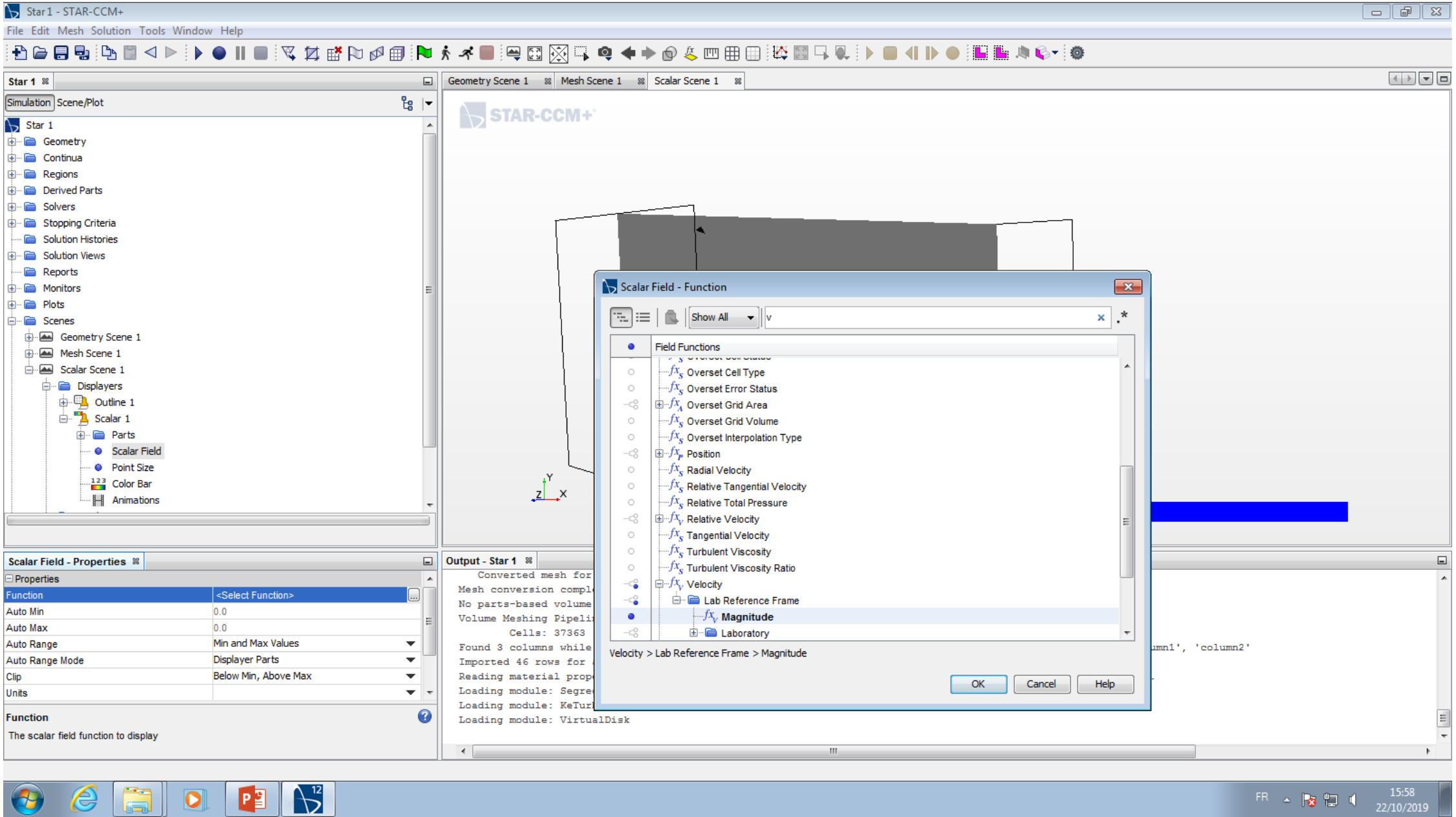
Define a « derived part » in which things are going to be displayed.
Adapt the plane to your case, explore the menus, be creative



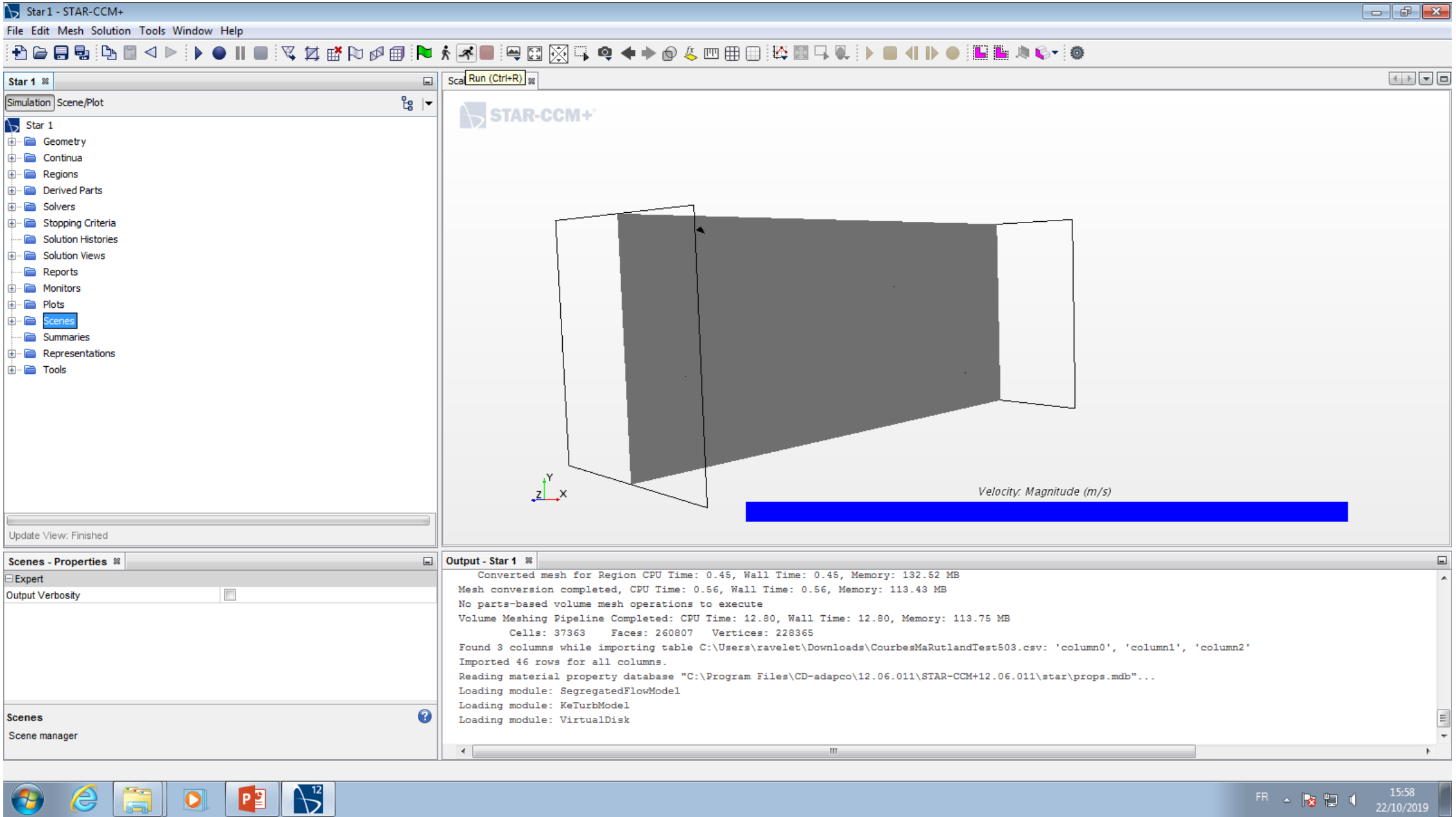
Here, the new plane will be automatically assigned to the scalar scene by choosing the option « diplay in existing displayer »



Choose the scalar field to be displayed: here the velocity magnitude



Launch the simulation



Analyse the results, please call me