**1. New Project**: File > New

**2. Importing the Geometry**

* File > Import > Import Surface Mesh
* Expand the Geometry > Parts to see the body

**3. Defining Boundary Surfaces**

* Expand the Geometry > Parts > Body > Surface node
* Right-click Face node and select Split by Patch
* Type (e.g. Inlet) in the Part Surface Name selecting the corresponding face
* Click create
* Repeat for the required faces, and click close

**4. Assigning Parts to Regions**

* Right-click Geometry > Parts > Body and select Assign Parts to Regions
* Select Create a Region for Each Part and Create a Boundary for Each Part Surface
* Click Apply and close

**5. Setting Boundary Types**

* Regions > Body > Boundaries node, select the e.g. Inlet node and set Type to e.g. Stagnation Inlet.

**6. Generating the Volume Mesh**

* Right-click Geometry > Operations and select New > Mesh > Automated Mesh
* Select Surface Remesher, Polyhedral Mesher, Prism Layer Mesher
* Expand the Operations > Automated Mesh > Meshers node.
* Within the Geometry > Operations > Automated Mesh node, right-click the Default Controls node and select Edit....
* To disable the prism layer generation on the slip wall: a) Right-click the Geometry > Operations > Automated Mesh > Custom Controls node and select New > Surface Control. b) Right-click the Custom Controls > Surface Control node and select Edit
* To reduce the mesh size on the blunt body surface: a) Right-click the Automated Mesh > Custom Controls node and select New > Surface Control. b) Right-click the Custom Controls > Surface Control 2 node and select Edit
* Click Generate Volume Mesh in the toolbar or select Generate Volume Mesh in the Mesh menu.
* Click Create/Open Scenes and select Mesh

**7. Selecting the Physics Models**

E.G. TURBULENT AND COMPRESSIBLE:

* Right-click the Continua > Physics 1 node and choose Select models....
* Coupled flow
* Ideal gas / coupled energy
* Steady
* Turbulent (RANS – K-Epsilon Two Layer, Wall distance and two-layer All y+ wall treatment
* Close

**8. Specifying the Initial Conditions for the Simulation**

* Expand the Continua > Physics 1 > Initial Conditions node
* Select Velocity and set Value

**9. Setting Boundary Conditions**

* Expand the Regions > subdomain-1 > Boundaries node.
* Expand the e.g. Inlet > Physics Values node, and select the e.g. Total Pressure node
* To set slip wall condition: Select Slip\_wall > Physics Conditions > Shear Stress Specification node and set Method to Slip.

**10. Visualizing the Solution**

* Right-click the Scenes node and select New > Scalar
* At the top of the simulation explorer panel, click Scene/Plot
* Select Scene 1 > Scalar 1 > Parts node
* In the Properties window, click (Custom Editor) in the right half of the Parts property.
* In the Parts dialog, expand Regions > subdomain-1 > Boundaries, select Inner\_wall and Symmetry\_plane1, and click OK.
* To define Mach number as the scalar shown: a) Within the Scalar 1 node, select the Scalar Field node. b) In the Properties window, click to the right of Function. c) In the Scalar Field - Function dialog, within the Filter by Path entry, enter Mach. The list of Field Function updates to show only those with, "Mach", in their names. d) Select Mach Number > Lab Reference Frame and click OK.
* At the top of the simulation explorer panel, click Simulation

**11. Monitoring the Drag Force Coefficient**

* Right-click the Reports node and select New > Flow / Energy > Force Coefficient.
* To set the part on which the report operates, open the Regions > subdomain-1 > Boundaries node and then drag the Inner\_wall node onto the Force Coefficient 1 node
* Right-click the Force Coefficient 1 node and select Create Monitor and Plot from Report.

**12. Setting Stopping Criteria**

* Select the Stopping Criteria > Maximum Steps node

**13. Running the Simulation**

* Click Run and visualise results