Although Siemens NX Advanced Simulations provides good help-functions and tutorials on almost all simulation types, flow calculations form an exception. Examples on the internet are mainly based on flow through pipes or ducts.

This basic tutorial gives therefore some simple guidelines to perform a simulation of a 2D flow around an object. Key to success of the simulation are the boundary conditions. This tutorial is only meant to give a first start into CFD calculations in NX, for practical use far more knowledge than presented here is required.

It is assumed that basic knowledge of geometrical modelling in NX is present, so that modelling a geometry like in below figure is no problem. Basically, for the CFD calculation the flow volume is modelled, so the shape of the body is subtracted from an assumed flow volume (typically using Boolean operations). The problem is not modelled as an actual 2D-problem, but as 3D with dimensions in side-direction being small compared to the other dimensions.
In the advanced simulation environment, as solver “NX Thermal/Flow” has to be selected, with “flow” for analysis type, see below figure.
In the “solutions” menu that pops-up then, default settings can be used. Note that especially in the “turbulence model” more advanced options are available.

Now the model (being the flow volume) can be meshed, and that goes just the way as for a structural calculation. Note that for meshing, also the standard mesh control is available. It is advised to use a fine mesh around the object.
Using the 3D tetrahedral mesh option, below window pops up:

Choosing the appropriate mesh size, all default settings can be used. Only the destination collector needs some attention, here you define the flow medium. Clicking on the icon and choose material:
You can select from the library for instance air or another fluid or gas.

After “ok” the volume will be meshed automatically with elements representing the gas or fluid.
After “activate simulation”, boundary conditions can be applied using the “simulation object” menu.

Choosing “Flow Boundary Condition” gives below menu. Here we can choose for instance the conditions at inlet and outlet of the volume:

For the inlet flow the appropriate area is selected and a velocity is specified, as shown below:
Then the exit area can be defined by selecting “opening” and the appropriate area, see below.
All the other (in this case: four) faces are treated (constrained) in the same way and constrained as “boundary flow surface”, which can be found under “boundary flow surface”, see below.

The “slip wall” option is used to indicate that there is no friction and shear along the surfaces (i.e. no influence on the velocity). After “OK” the model including boundaries looks like in below figure, and is ready for solution.
After solution the results can be assessed:

eexample1.sim1 : flowexample1 Result:
Load Case 1, Static Step 1
Total Pressure - Element-Nodal, Unaveraged, Scalar
Min : 6.721E-003, Max : 1.138E-002, Units = N/mm²(MPa)
NX also gives the possibility to have the actual loads due to the flow on the object calculated. Under “Simulation Object Type” in simulation (see below) there is a “report” option. Of course this option has to be activated before solving.
In this menu, you can select “lift and drag”, and the contours of the object:

In your working directory, during solve, a html document is created, which contains the calculated loads:
**NX Thermal/Flow Report**

**Units**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>m</td>
<td>°C</td>
<td>kN/m²</td>
<td>N</td>
<td>m/s</td>
<td>m³/s</td>
<td>kg/s</td>
<td>W</td>
<td>W/m²</td>
<td>W/m²°C</td>
</tr>
</tbody>
</table>

**Lift Drag**

<table>
<thead>
<tr>
<th>Group</th>
<th>Time</th>
<th>LIFT X</th>
<th>LIFT Y</th>
<th>LIFT Z</th>
<th>LIFT Mag</th>
<th>DRAG X</th>
<th>DRAG Y</th>
<th>DRAG Z</th>
<th>DRAG Mag</th>
<th>SIDE X</th>
<th>SIDE Y</th>
<th>SIDE Z</th>
<th>SIDE Mag</th>
<th>PITCH X</th>
<th>PITCH Y</th>
<th>PITCH Z</th>
<th>PITCH Mag</th>
<th>ROLL X</th>
<th>ROLL Y</th>
<th>ROLL Z</th>
<th>ROLL Mag</th>
<th>ROLL Mag</th>
<th>ROLL Mag</th>
<th>ROLL Mag</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>3</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Note the dimensions! These are of course the values for the actual modelled section.