

Laboratory 5-6

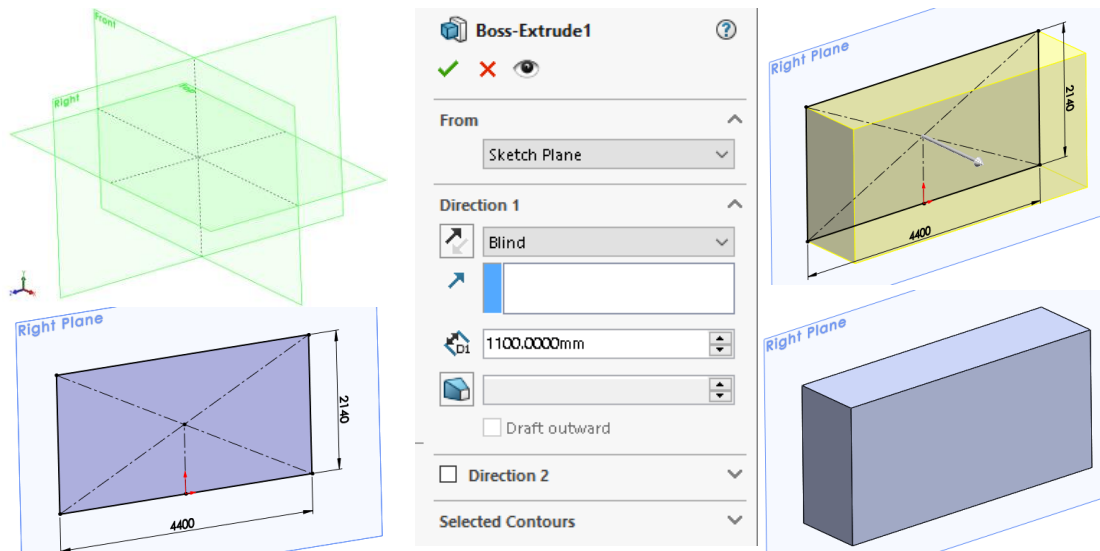
Aerodynamic study of a generic vehicle body model

This laboratory work presents the steps involved in conducting an aerodynamic study for a generic passenger car body model. The first part of the paper presents the CAD modeling of the car body, while the second part presents the simulation performed to determine the aerodynamic drag force, lift force, aerodynamic coefficient, and lift coefficient. The second part of this work also presents the methods for graphically and numerically visualizing the results obtained.

I. Generic Vehicle CAD Modeling

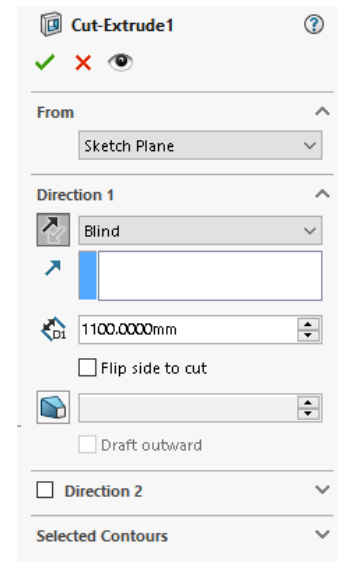
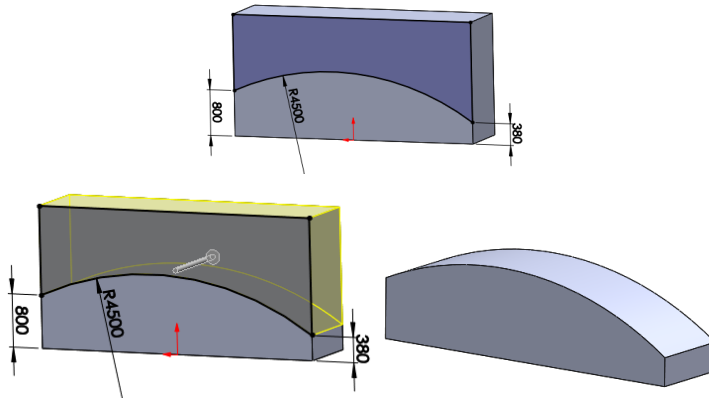
CAD modeling of the body structure is performed using the sketching and modeling commands available within the software. The steps required to model this component are presented as follows:

1. The outline is created from the **Feature Manager** toolbar, where the **Right Plane** is selected to draw the first sketch, from which a rectangle is defined, dimensioned at 4400x2140 using  command. The rectangle will be positioned at the origin of the sketch, at half its length. From the **Features**, select the **Boss-Extrude** command. The sketch is extruded 1100 mm along the OX axis. 



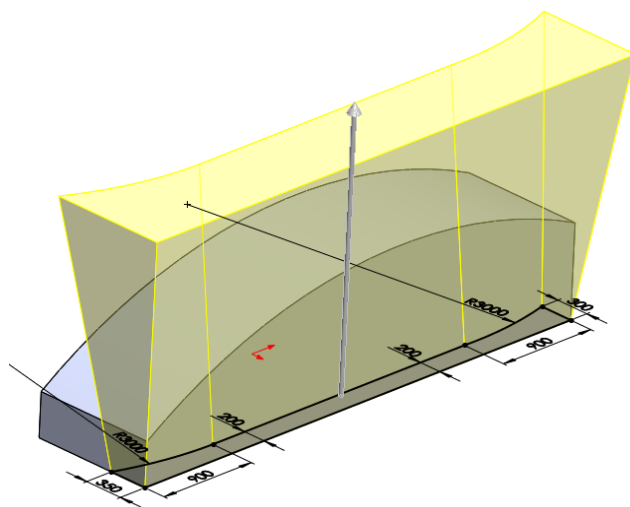
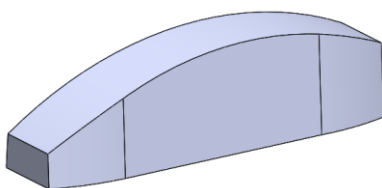
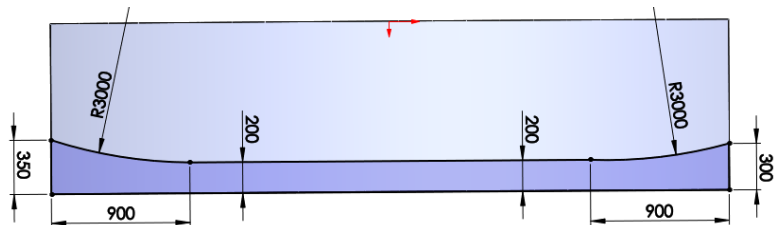
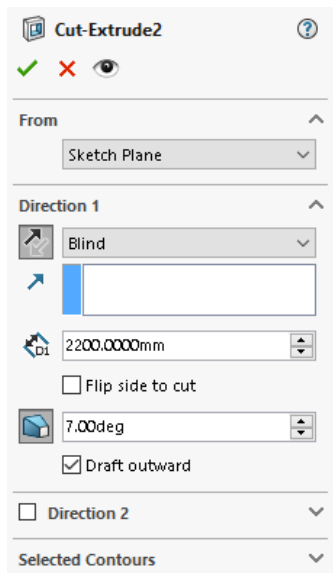
2. The upper body shape is created by cutting out a sketch drawn on the flat surface of the model. The model is cut

out using the command  .



3. The side profile of the body is created by tracing the outline shown in the adjacent image onto the flat lower surface of the model, using the dimensions indicated. The sketch is created using the **Line** , **3-Point Arc** , and **Smart Dimension**  commands.

The sketch is cut using the **Cut Extrude**  command, over a distance of 2200 mm at an angle of 7° toward the outside of the model.

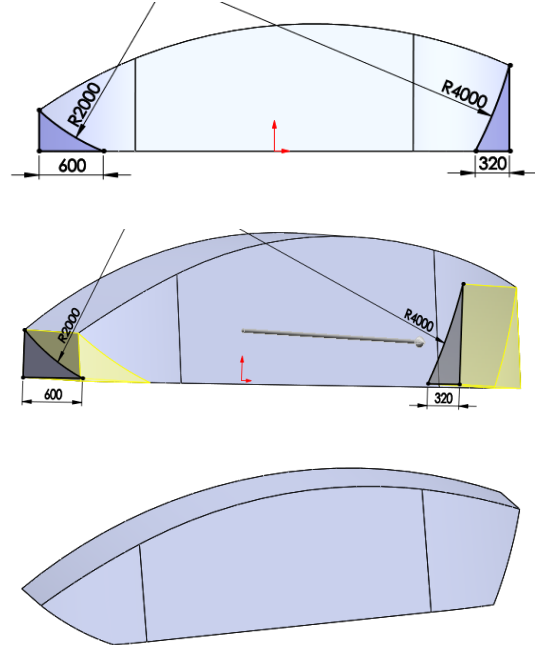
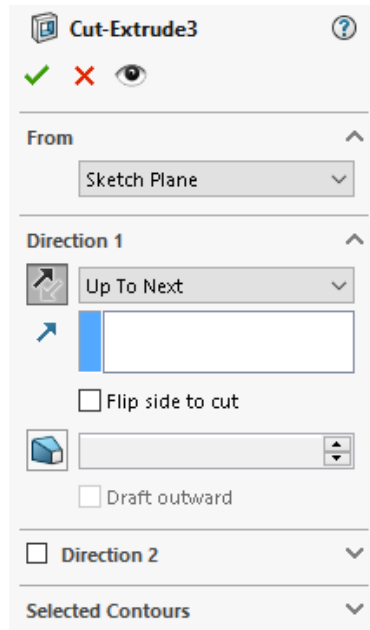


4. The approach angle and departure angle of the body are created by drawing two sketches on the flat, vertical surface of the model.

In the sketch environment, the commands used are: **Line**, **3-Point Arc**, and **Smart Dimension**.

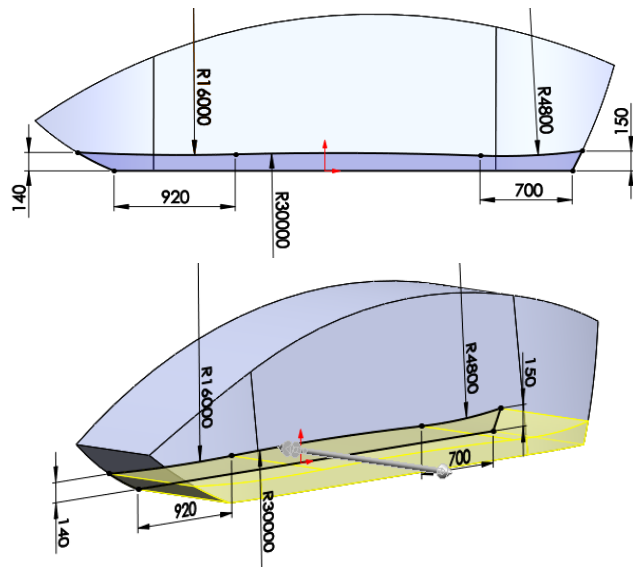
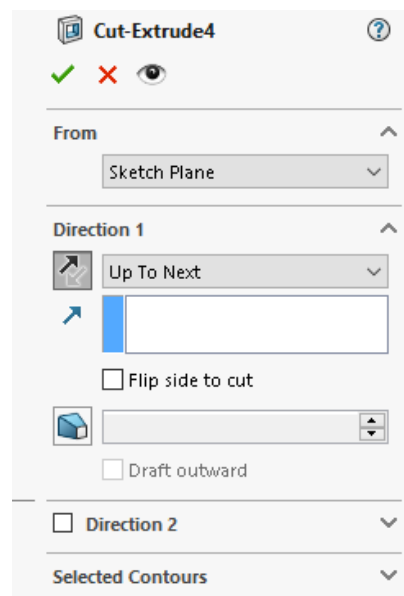
The outline inside the sketches is removed using the **Cut Extrude**

command.



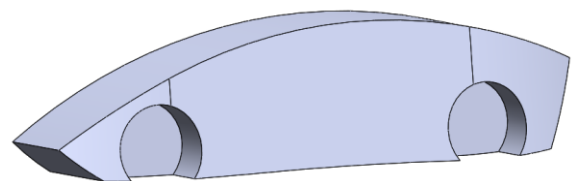
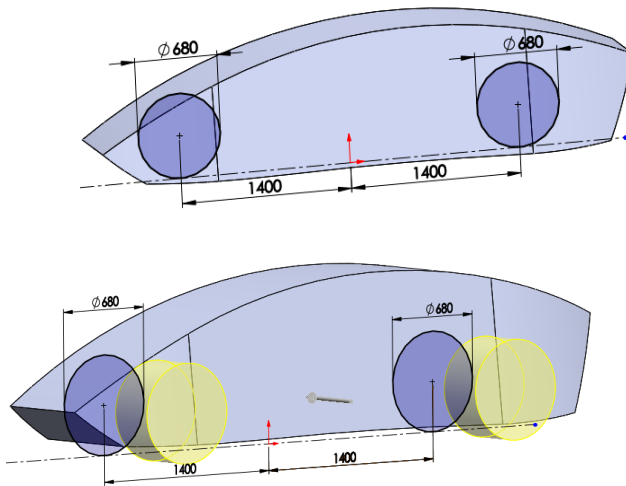
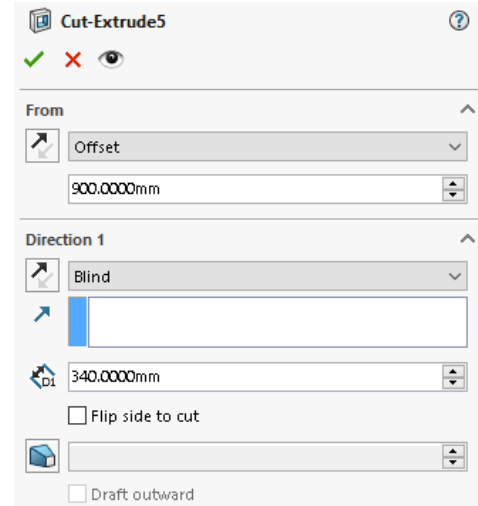
5. The shape of the lower part of the body is given by cutting out the sketch drawn on the vertical flat surface of the model. The sketch is cut across its entire width using the

Extruded Cut command.

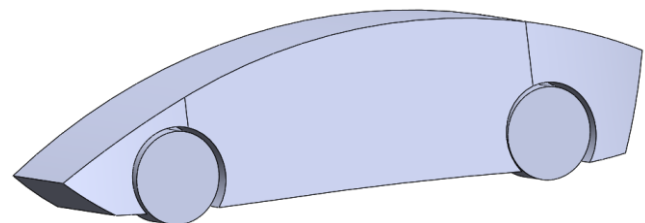
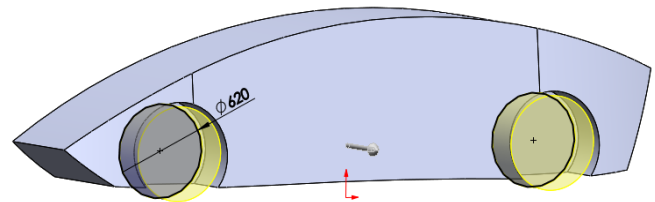
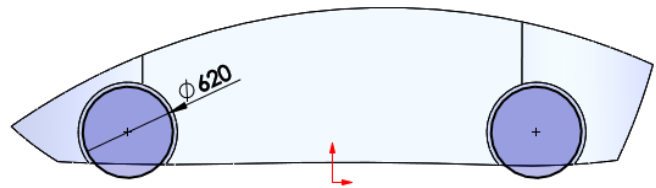
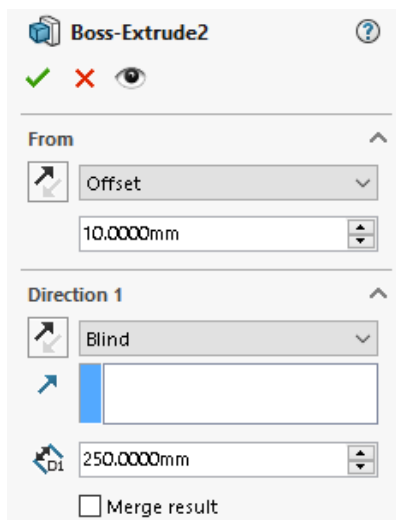


6. The wheel wells are cut out from the outline formed by two circles with a diameter of 680 mm drawn on the flat surface of the body. The circles are cut

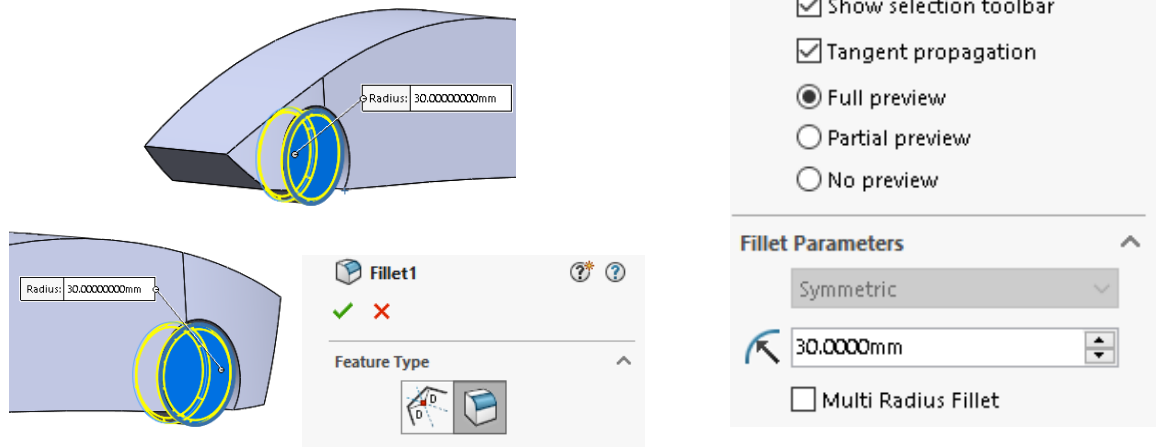
using the **Extruded Cut** command



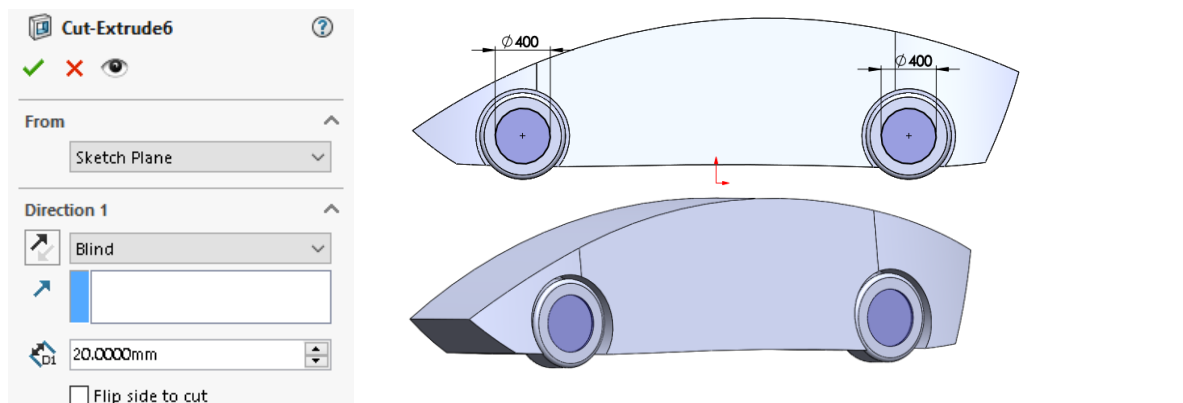
7. The wheels are modeled by extruding the sketch consisting of the circles that define the wheel diameter. The wheel sketch is extruded 10 mm away from the selected flat surface.



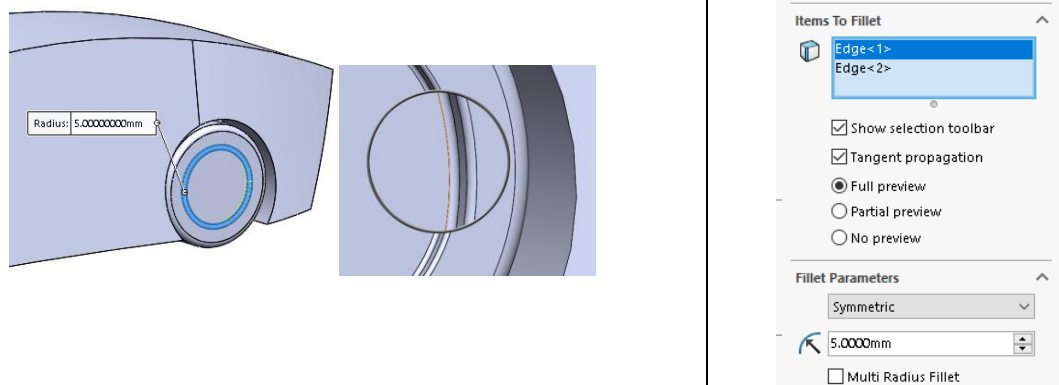
8. The modeled wheels are rounded by applying the **Fillet** command at 30 mm, for each wheel separately.



9. To create the recessed shape of the rim on the front surface of the wheel, a sketch consisting of two circles is drawn, which will be cut out using the **Extruded Cut** command.

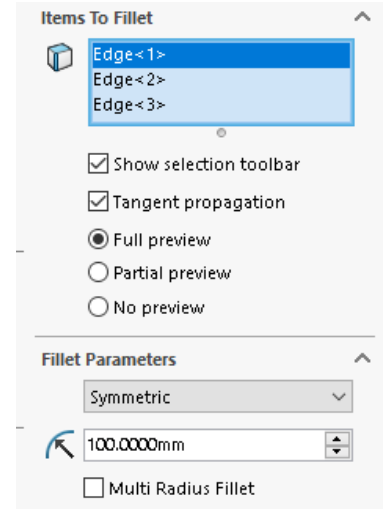
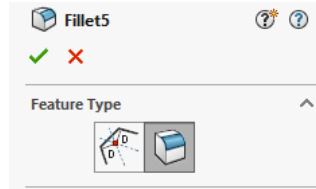
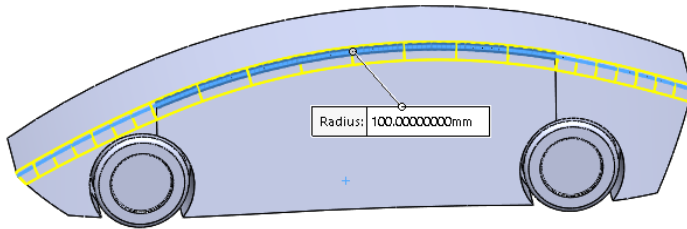


10. The edges resulting from the cutout are rounded by applying the **Fillet** command with a radius of 5 mm, for each wheel separately.



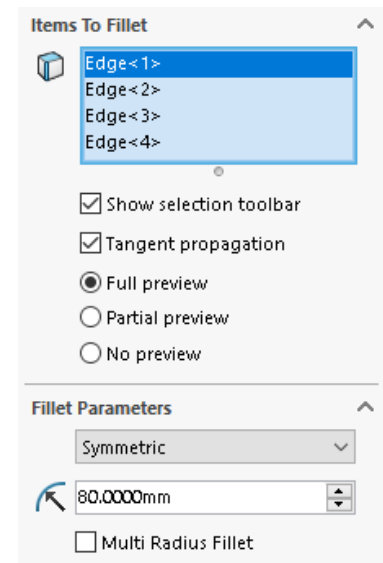
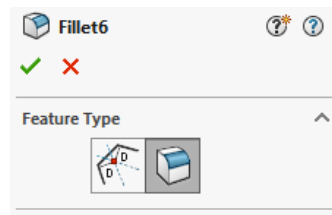
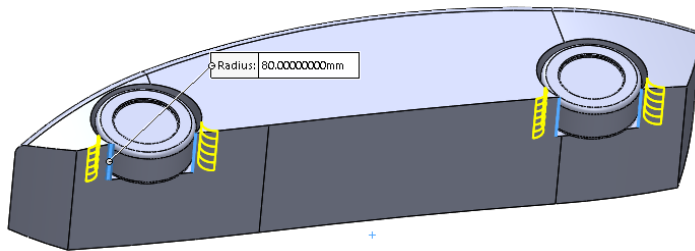
10. The upper edge of the body is rounded with

the **Fillet** command.



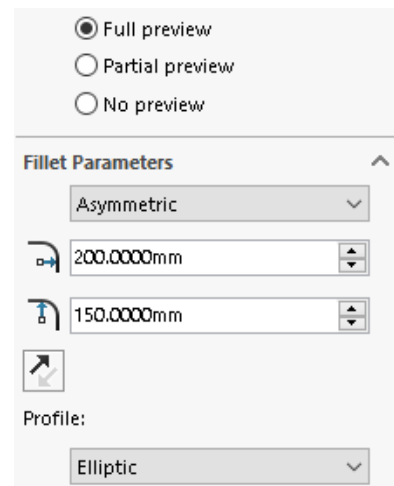
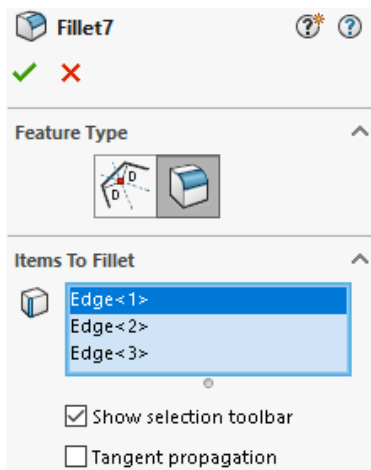
11. The edges indicated from the bottom of the body are rounded with

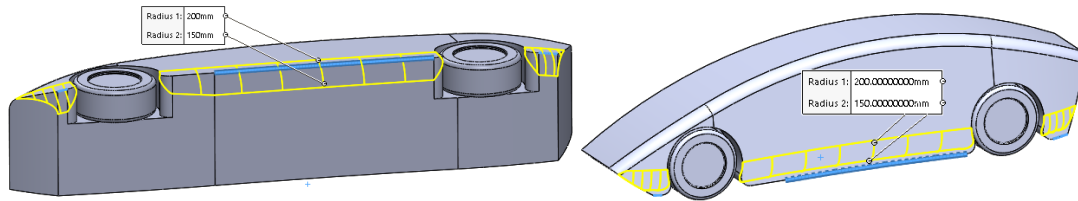
the **Fillet** command to a radius of 80 mm.



12. The lower side edge of the body is rounded asymmetrically using

the **Fillet** command with a radius of 200 mm and 150 mm.

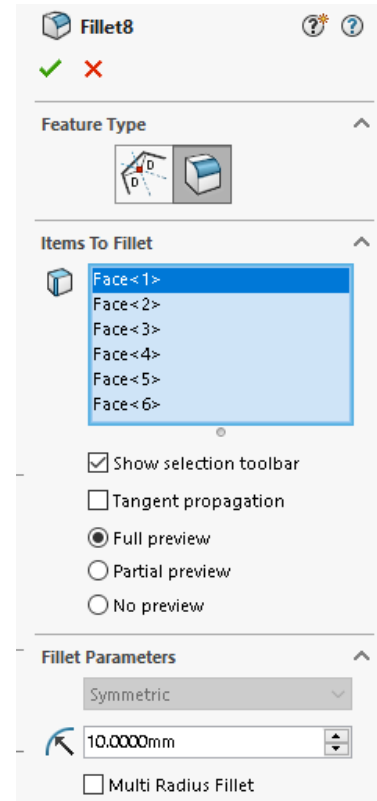
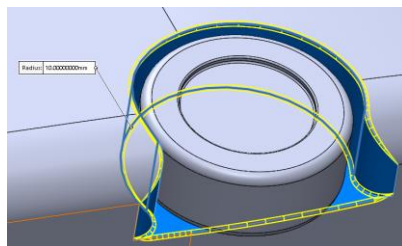
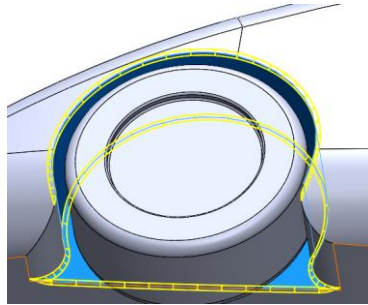




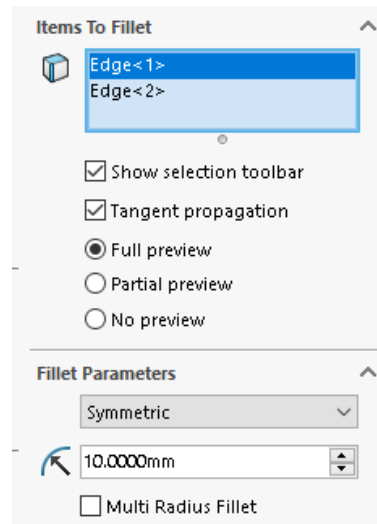
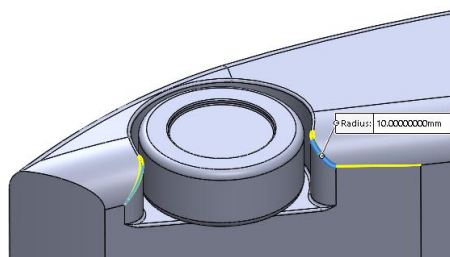
13. The wheel fairing edges are rounded with the **Fillet**



command to a radius of 10 mm. The order is used separately for each fairing.



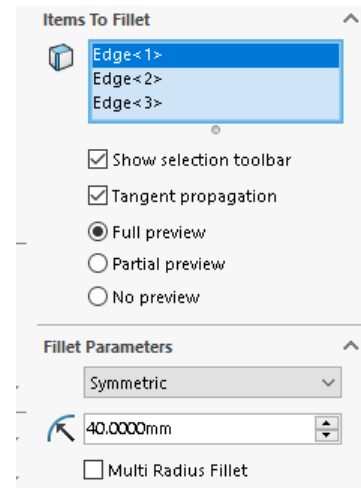
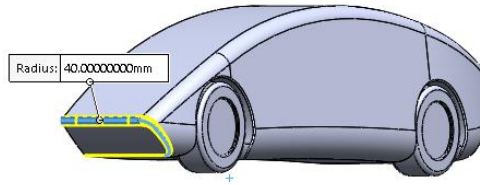
14. Following the rounding of the edges of the wheel fairings to the front wheel fairing, two edges are rounded separately with a radius of 10 mm due to the impossibility of rounding them in the previous operation.



15. The front part of the body is rounded with the **Fillet**



command to a radius of 40 mm.

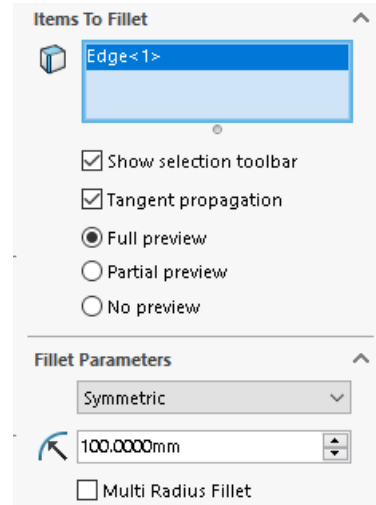
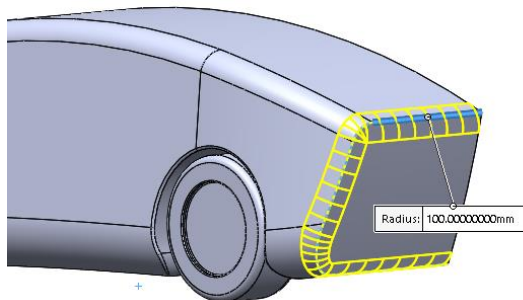


16. Partea din spate a caroseriei este rotunjită cu comanda **Fillet** la o rază de 100 mm.

17. The rear of the body is rounded with the



Fillet command at a radius of 100 mm.



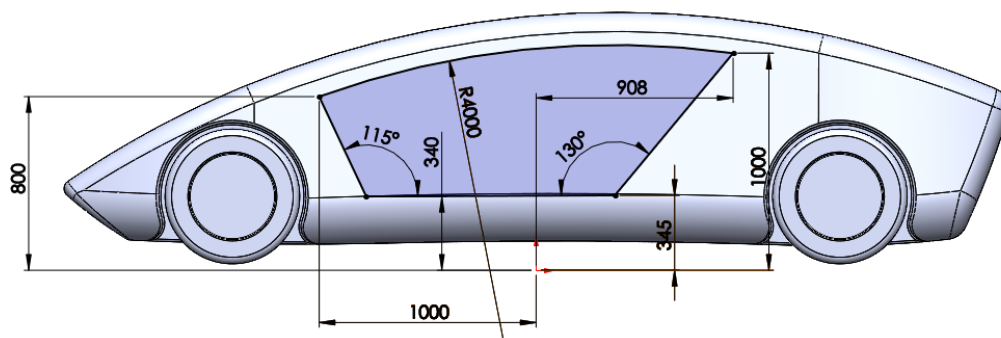
18. The general shape of the door is determined by the outline drawn on the vertical flat surface of the car body model. Select the **Sketch** toolbar, where the **Line**



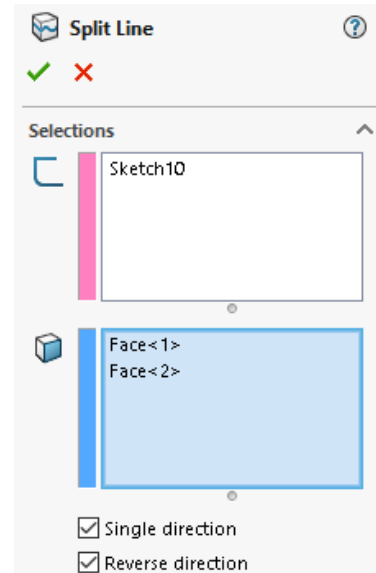
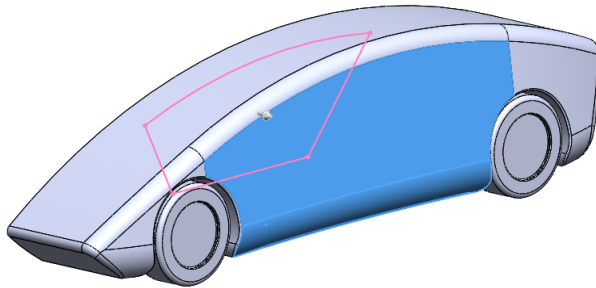
, and **Smart Dimension**



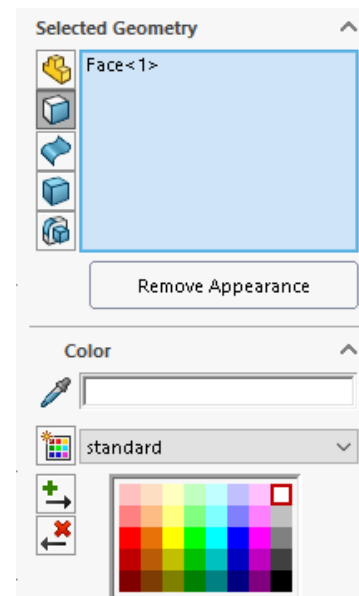
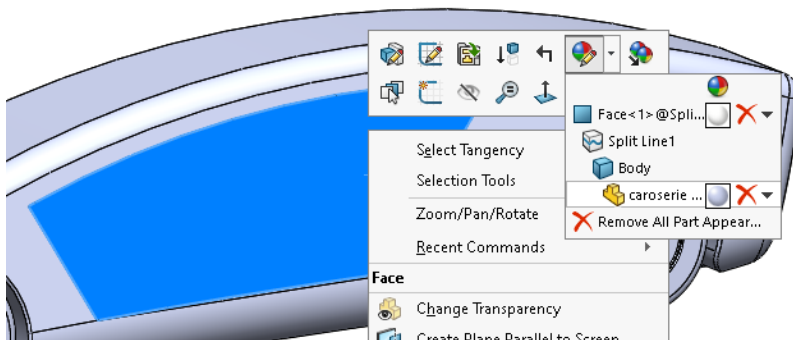
commands will be used to create the following sketch according to the dimensions.



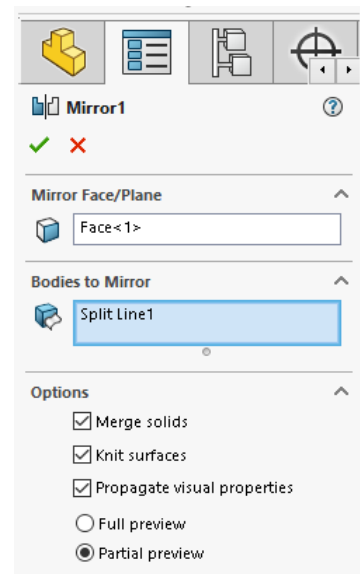
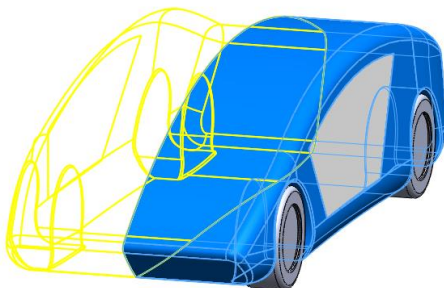
19. To outline the door on the body model from the toolbar, select the **Curves**  command, then the **Split Line**  command, where you will select the sketch and the surfaces onto which it will be projected.



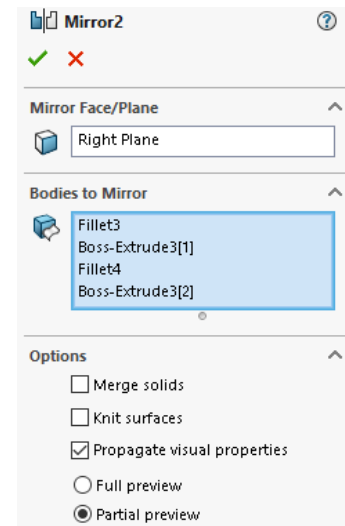
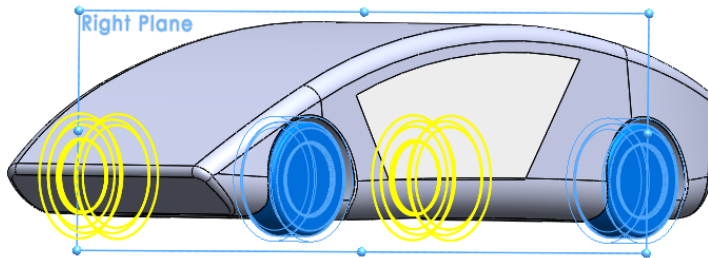
20. For better visual observation of the surface that defines the door, it is colored white. This operation is performed by right-clicking on the surface that defines the door and choosing to make the door surface colorable.



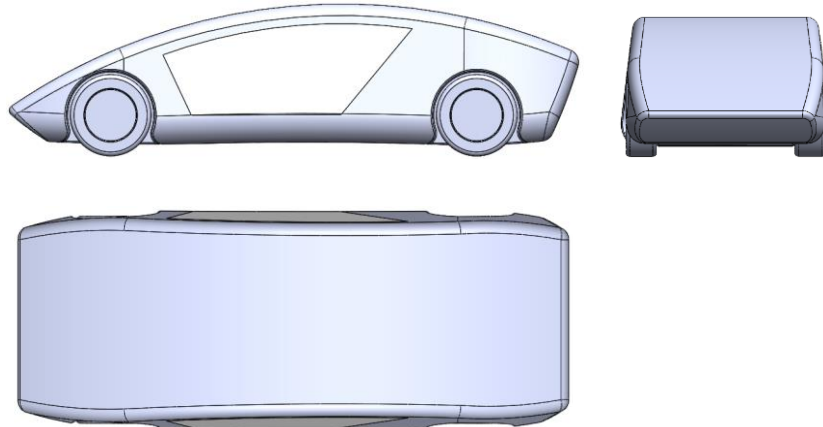
21. The body model will be mirrored using the **Mirror**  command in the toolbar. Mirroring the body model will be done by selecting "**Bodies to Mirror,**" and in the "**Mirror Face/Plane**" section, you will choose the flat face of the body model.



22. Mirroring the wheels for the right side of the body is done using the Mirror command relative to the Right Plane, which is located in the median area of the body.

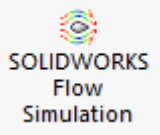


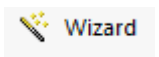
The resulting body model is presented in triple orthogonal projection in the adjacent image.



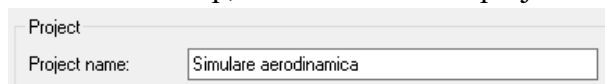
II. Aerodynamic Study of a Generic Vehicle Structure

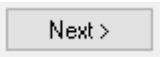
1. The study of airflow around the car body model is conducted using the **Flow Simulation** module integrated into the SolidWorks software package. To activate this module from the toolbar, select the tab, **SOLIDWORKS Add-Ins** then activate the fluid

simulation module by left-clicking on the icon . After the module is activated, the tools tab for that module appears in the toolbar.

2. To create a new fluid simulation project from the toolbar, select the command , which opens a dialog box where you can name the project and choose its configuration. In the first step, we name the project by typing “**Simulare**

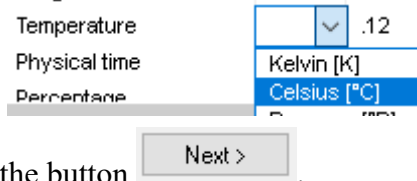
aerodinamica”



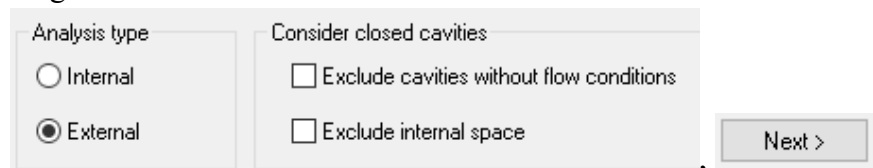
from the keyboard, then left-clicking the mouse on the button  to proceed to the next step.

3. From this dialog box, you can select the standard used in the current project and choose the units of measurement for the parameters according to the project requirements. For example, the image on the right shows how to convert the unit of

temperature from Kelvin (°K) to Celsius (°C) . To move to the next project settings page, left-click the button



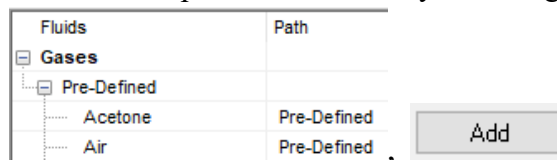
4. From this dialog box, you select the type of fluid simulation to be performed within the project. In this case, the external simulation type will be chosen, as can be seen in the adjacent image.



5. Un aspect important în cadrul simulării este constituit de alegerea fluidului de simulare. În acest proiect se va alege ca și fluid predefinit **Aerul**, , prin selecția acestuia cu click stânga mouse și butonul. Pentru a trece la următoarea pagină de setări ale proiectului se apasă butonul

6. An important aspect within the simulation is the choice of the simulation fluid. In this project, **Air** will be chosen as the predefined fluid by selecting it with a left mouse

click and the button



7. In this dialog box, you can make settings for simulating fluids under heat transfer conditions and choose the roughness of that surface.

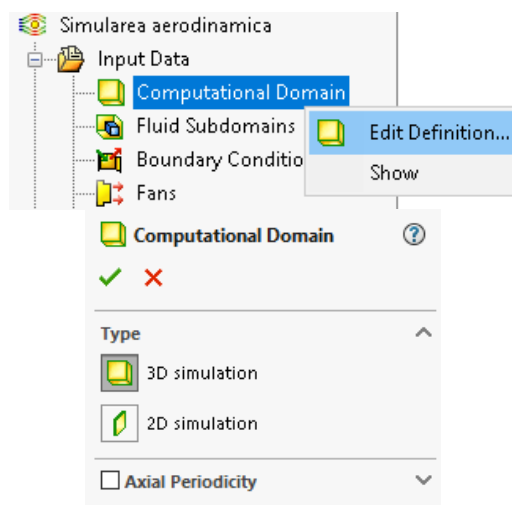
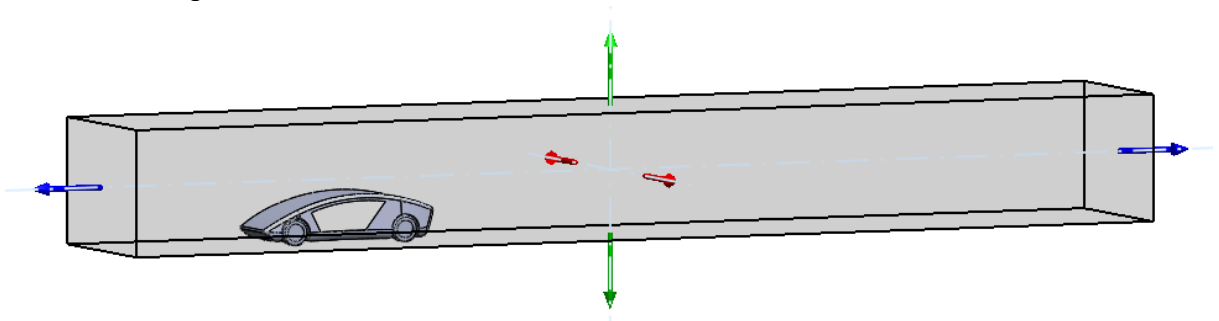


8. In this section, an air velocity of 33.34 m/s is applied along the OZ axis, as can be seen in the adjacent image. The minus sign in front of the numerical value of the velocity indicates that the air velocity is oriented along the negative axis.

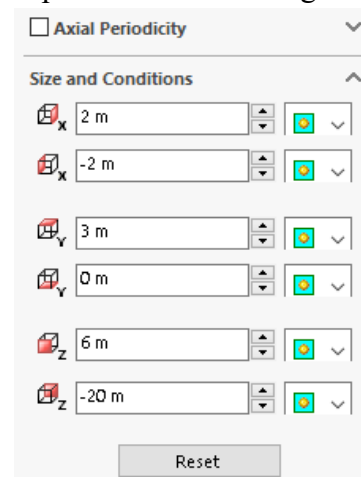
Finish

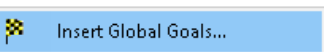
Velocity Parameters	
Parameter	Velocity
Defined by	3D Vector
Velocity in X direction	0 m/s
Velocity in Y direction	0 m/s
Velocity in Z direction	-33.34 m/s

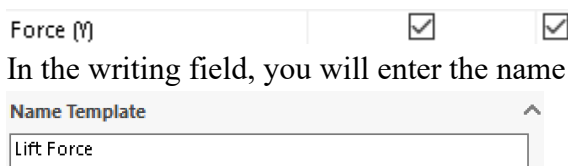
9. From the **Feature Manager**, right-click to select the dimensional parameters of the calculation domain in which the external air flow simulation around the vehicle body will be performed.



The parameters of the chosen computational domain are presented in the image below.



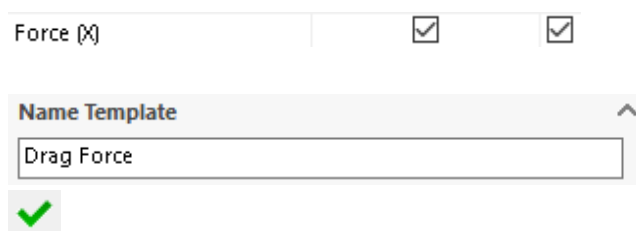
10. To determine the numerical value of the aerodynamic lift force from the **Feature Manager**, right-click on **Goals** and insert a new global goal , from the displayed list, by checking the force on the OY axis.



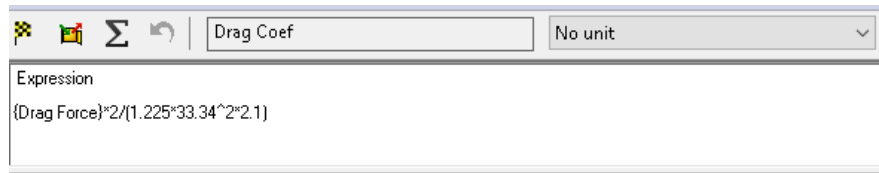
In the writing field, you will enter the name of this force, "Lift Force."

This work order can also be selected from the toolbar under the Goals button .

11. The aerodynamic force is also determined by right-clicking on "Goals" and adding a new global objective, selecting "Force on OX axis," and naming this force "Drag Force."



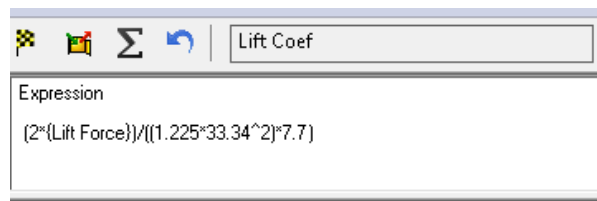
12. The formula for determining the value of the aerodynamic coefficient is introduced from **Goals**  by selecting  **Insert Equation Goal...**. The formula expression is written in the field allocated for keyboard input, and the size is called "**Drag Coef**" as you can see in the image below.



The dialog box shows the name 'Drag Coef' and the unit 'No unit'. The expression field contains the formula: $\{ \text{Drag Force} \}^2 / (1.225 \cdot 33.34^2 \cdot 2.1)$.

The variables in the formula represent: {Drag Force} – Aerodynamic force [N], 1.225 – air density at 15°C – [kg/ m³], 33.34 – air velocity [m/s], 2.1 – maximum cross-sectional area [m²]. .

13. To introduce the formula for determining the lift coefficient, the same procedure as in the previous operation is followed .

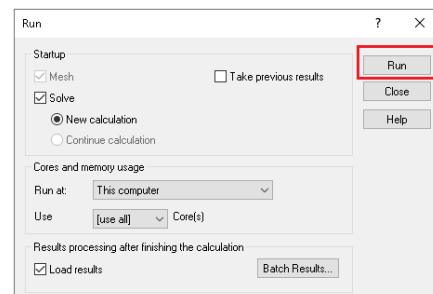


The dialog box shows the name 'Lift Coef'. The expression field contains the formula: $(2 \cdot \{ \text{Lift Force} \}) / ((1.225 \cdot 33.34^2) \cdot 7.7)$.

14. The calculation process is started by right-clicking the aerodynamic simulation project name in the **Feature Manager**  or from the toolbar  **Run...**



15. In the open dialog box, you can choose the number of processors used for the simulation calculation or the computer on which the simulation can be performed.



The 'Run' dialog box shows options for 'Startup' (Mesh, Solve), 'Cores and memory usage' (Run at: This computer, Use: [use all] Core(s)), and 'Results processing after finishing the calculation' (Load results). The 'Run' button is highlighted with a red box.

16. After the calculations are completed, the results obtained for each requested parameter are displayed in the dialog box, showing the current value and the average value of the results.

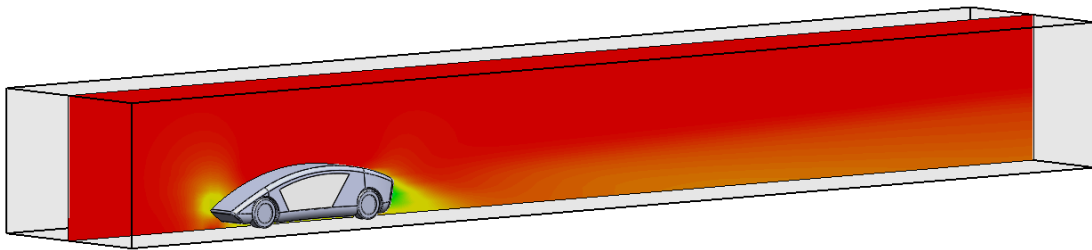
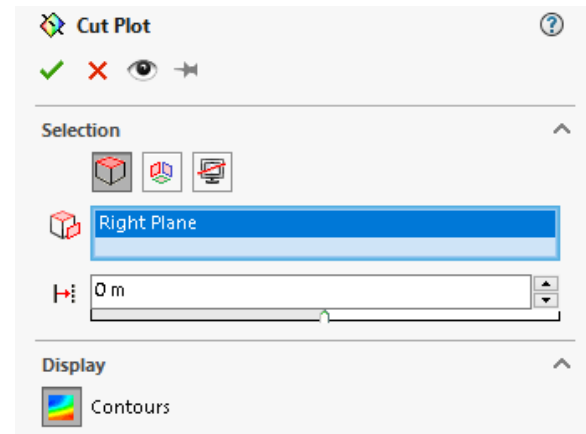
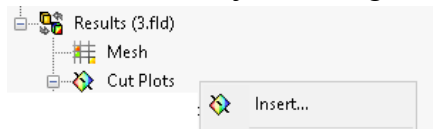
Solver: Simularea aerodinamica [Default] (caroserie SPC.SLDPRJT) - [List of Goals]

Name	Current Value	Progress	Criterion	Averaged Value
Drag Coef	-0.266371	Achieved (IT = 62)	0.0107305	-0.267699
Drag Force	-380.84 N	Achieved (IT = 62)	15.3418 N	-382.74 N
Lift Coef	0.112005	Achieved (IT = 65)	0.00169301	0.111948
Lift Force	587.17 N	Achieved (IT = 65)	8.87538 N	586.872 N

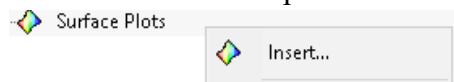
Info Log Pressure(Right Plan) List of Goals

Ready Solver is finished.

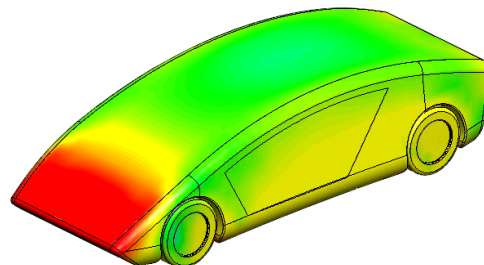
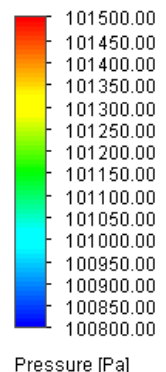
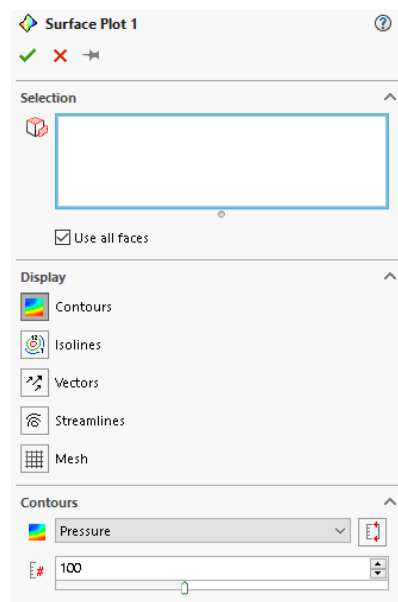
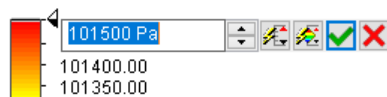
17. The graphical visualization of the results obtained on a plane is achieved by right-clicking on from the **Design Tree, Flow Simulation Analysis**, from where the desired visualization plane is inserted, as shown in the adjacent image.



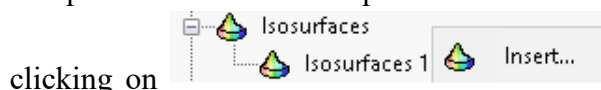
18. To visualize the pressure distribution on the model surface, select and insert a surface plot from the **Design Tree, Flow Simulation Analysis**. To visualize the air pressure distribution on all surfaces of the model, check the "Use all faces" option.



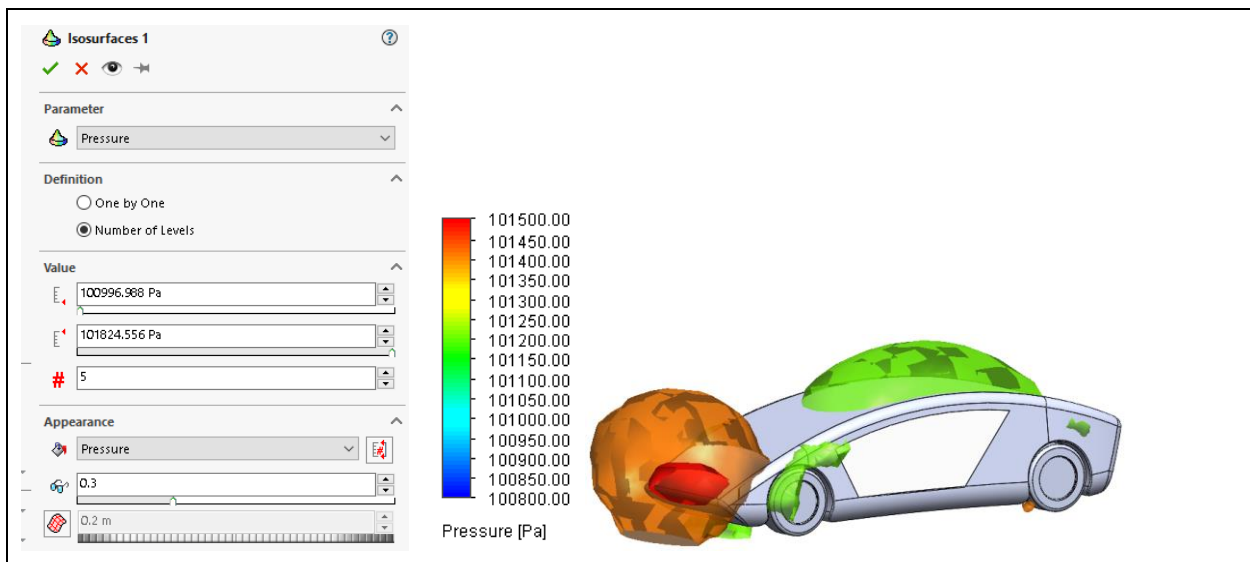
To create a numerical representation range for the color model by left-clicking on the upper numerical value in the legend, it becomes editable, as does the lower value. The model and the legend for the pressure distribution representation are shown in the adjacent images.



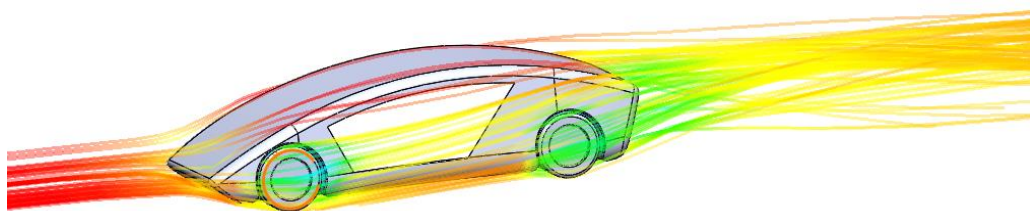
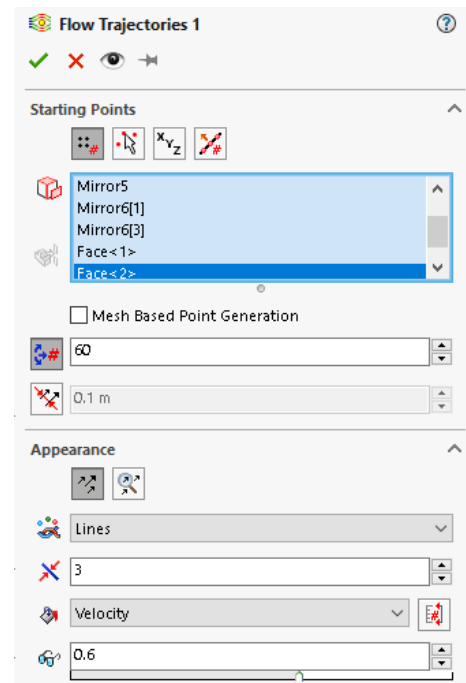
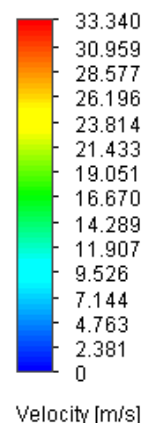
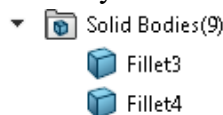
19. The pressure distribution representation bounded by surfaces can be achieved by right-



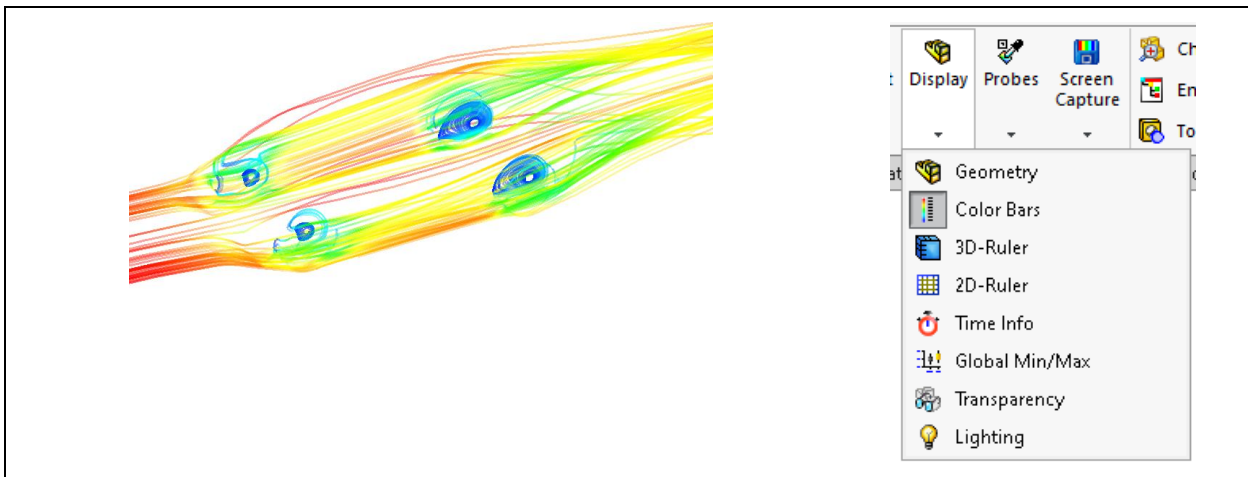
clicking on in the **Design Tree, Flow Simulation Analysis**. This representation can be achieved based on the number of chromatic levels; in this case, a number of 5 chromatic representation levels are chosen.

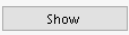


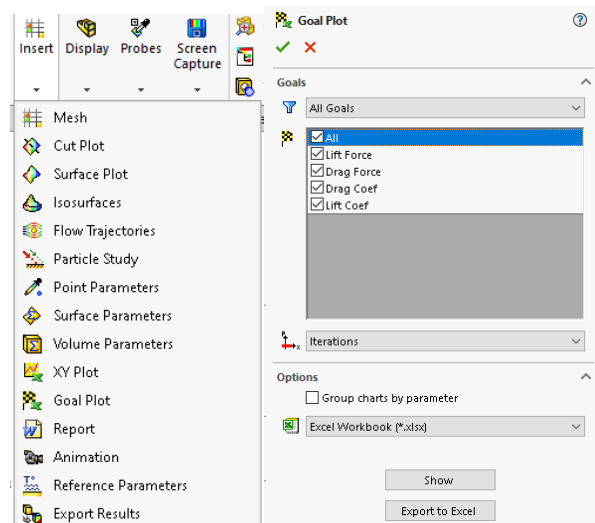
20. The graphical visualization of the results obtained in the air flow line mode can be achieved by right-clicking on  Flow Trajectories  Insert... in the **Design Tree, Flow Simulation Analysis**. In this case, the representation of the air velocity distribution of 33.34 m/s around the car body model is presented. Selecting the solid components of the model can be done from the work tree by mouse selection.



21. To view the model and the results obtained in different ways from the **Flow Simulation** mode toolbar, the model can be hidden, leaving only the flow lines visible, as shown in the image below. The model can also be represented at a transparency level chosen by the operator using the command  Transparency .



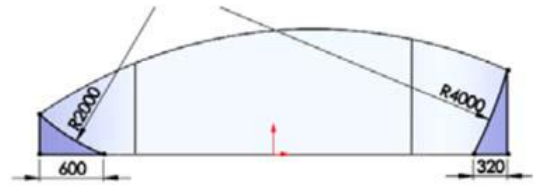
22. The numerical visualization of the results can be viewed by right-clicking on , then checking all the quantities for visualization and clicking the button . After performing this operation, a dialog box appears showing the names of the calculated quantities, their units of measurement, the minimum and maximum values, the average value, etc.



Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Lift Force	[N]	587.170	586.872	585.016	591.749	100	Yes	6.734	8.875
Drag Force	[N]	-380.840	-382.740	-390.471	-380.499	100	Yes	9.972	15.342
Drag Coef	[]	-0.2663707	-0.2676993	-0.2731066	-0.2661317	100	Yes	0.0069749	0.0107305
Lift Coef	[]	0.1120045	0.1119478	0.1115936	0.1128781	100	Yes	0.0012845	0.0016930

III. Self- evaluation test

1. In the example above, change the two dimensions related to the front shape ($R=2000$ mm) and rear shape ($R=4000$ mm) of the vehicle and perform a simulation process for each change, plus a simulation process that includes both changes (at different air flow speeds). Enter the data into a comparative table.



	Base model	The model with the front end changed	The model with rear end changed	The model with front and rear end changed	Obs.
Drag force [N]					
Lift Force [N]					
Drag coef.					
Lift coef.					

2. Add a new three simulation cases at speeds of 90 and 120 km/h of the air: in first case add a front spoiler to the generic vehicle model, in second case add a rear spoiler and in third case run the simulation with both spoilers. Enter the data into a comparative table.

	Base model	The model with the front spoiler	The model with rear spoiler	The model with front and rear spoiler	Obs.
Air speed	90 km/h				
Drag force [N]					
Lift Force [N]					
Drag coef.					
Lift coef.					

	Base model	The model with the front spoiler	The model with rear spoiler	The model with front and rear spoiler	Obs.
Air speed	120 km/h				
Drag force [N]					
Lift Force [N]					
Drag coef.					
Lift coef.					