SIMOULDING - Rotomoulding Simulation Software

- V-1.2

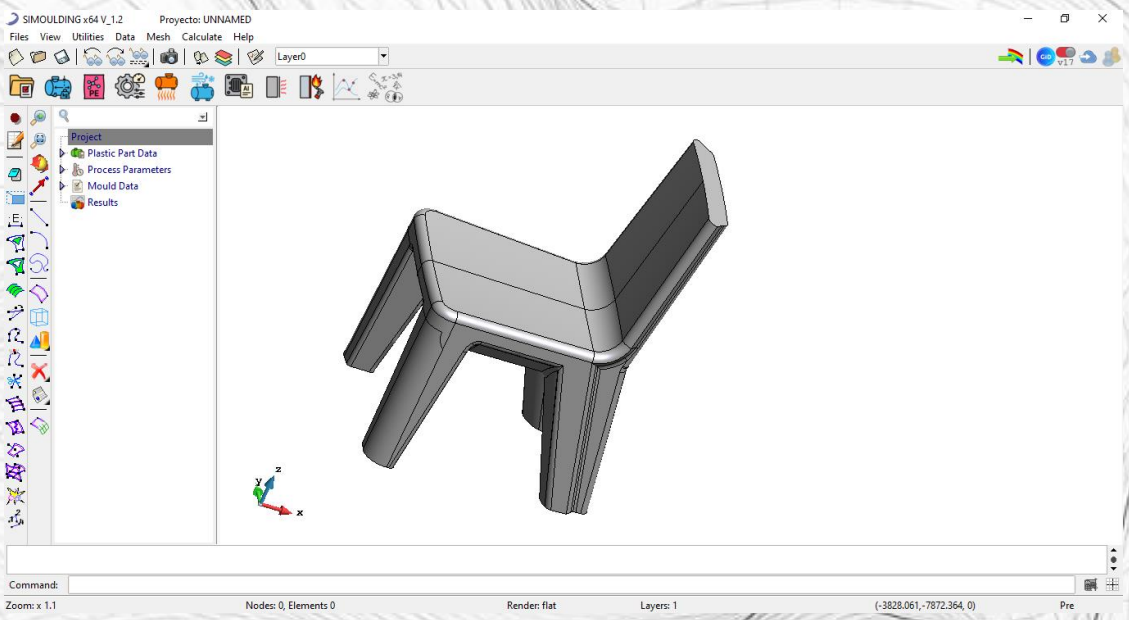
A key tool for the design and development of rotomolded products

Graphical User Interface

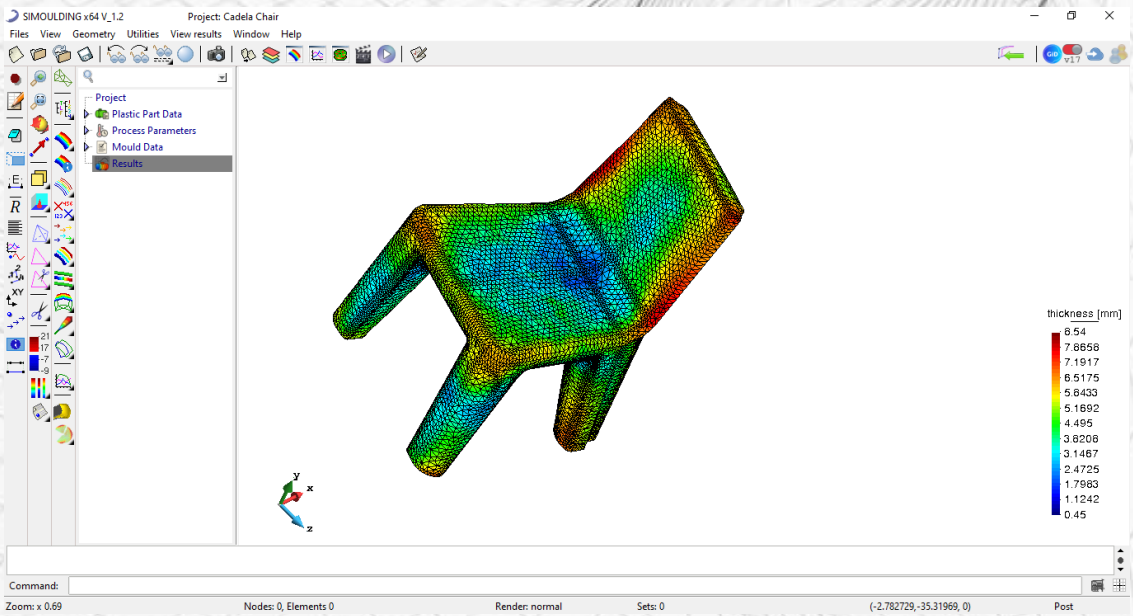
The **SIMOULDING** Graphical User Interface is developed based on a customization of the **GiD** software. For more information visit **www.gidsimulation.com**

The SIMOULDING graphical interface has two work sections:
A pre-processing section, where the user can enter the geometry of the part to be simulated, load all process parameters, part and mold data, and generate the finite element mesh.
A post-processing section, where the user can see the results obtained from the simulation.

Pre-processing section



Post-processing section



General information

SIMOULDING is a program for simulating the rotational molding process. Its interface allows you to load a 3D object or a 3D mesh and enter all the data and parameters inherent to the process and the object to simulate its behavior.

The parameters for entry are divided into 4 groups:

-  Project
-  Plastic Part Data
-  Process Parameters
-  Mould Data

The data and parameters that must be entered to perform the simulation, in addition to the geometry of the object, are the following:

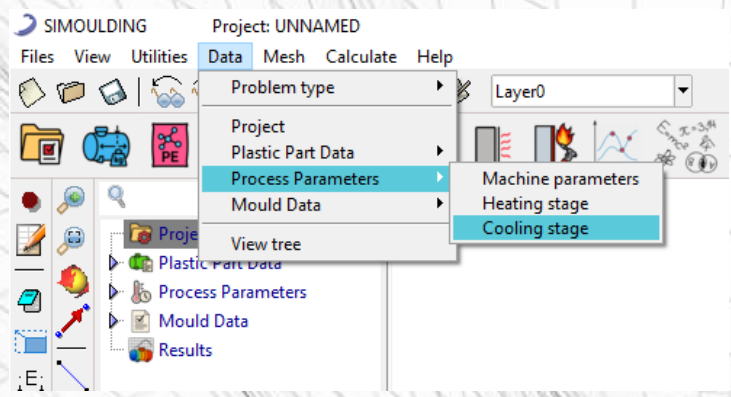
- Project: Here you must define the required results.
- Part features: The purpose of the part must be defined, by required thickness or required weight.
- Polymers: Here you must define the type of polymer of the object and the values of its characteristic properties.
- Machine parameters: The type of movement and rotation speeds must be defined.
- Heating parameters: Here you define the oven temperature and heating time.
- Cooling parameters: Here you define the type of cooling and cooling time.
- Mould features: Here the material and thickness of the mould are defined.

Additional data. In addition to the data required for the simulation, the user can enter control points of interest and, when the project requires it, additional data related to the mold.
These data are as follows:

- Insulation areas: In cases where the mould has insulated areas to prevent heating of said zones, they can be identified on the geometry of the object. Areas where heat is not delivered efficiently due to the presence of other moulds in the machine can also be identified.
- Preheating areas: If the process requires preheating certain areas of the mould before entering the oven, these areas can be identified and their entry temperature pre-set.
- Points of interest: Possibility of monitoring the temperature evolution at any point of the mold geometry.

Note: The user can access the parameter entry from the "Data" context menu or from the specific toolbar located at the top of the graphical window.

Access from the "Data" context menu:



Access from the Toolbar:

Getting started

Geometry and mesh

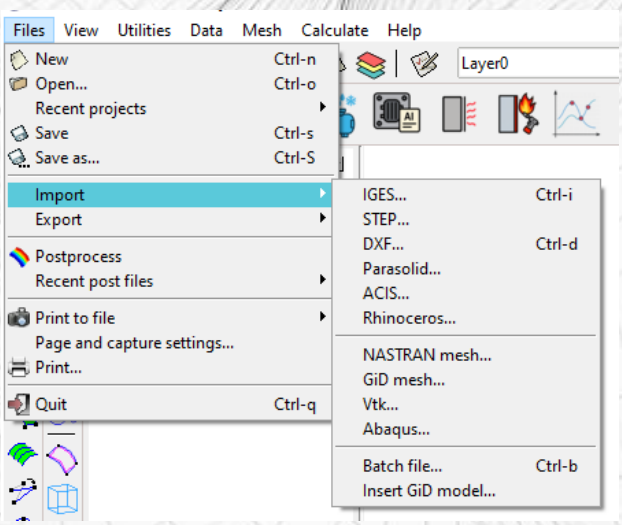
Geometry import

The geometry must be imported using the standard **GiD** commands (See **GiD** Help for details).

Several import formats are available: IGES, STEP, DXF, Parasolid, ACIS, Rhinoceros.

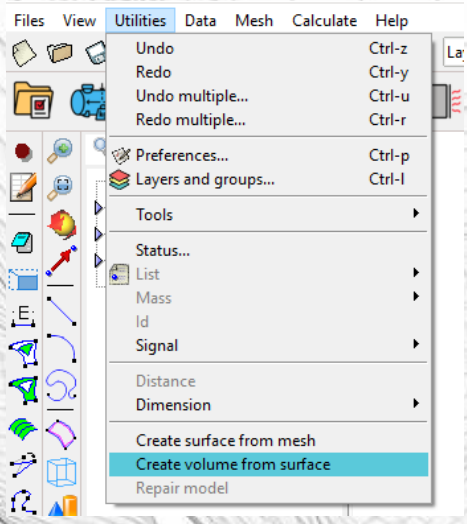
To import part, open the import window:

Files->Import



The geometry to be imported must be a closed 3D CAD part, solid or shell. In the case of a shell, there must be no thickness.
The program will use a 3D volume mesh (tetrahedral elements) to perform the simulation. The entered 3D geometry will represent the internal volume of the mold in the simulation.
If entering a shell part, the user must convert it to a solid part using the volume creation tool.

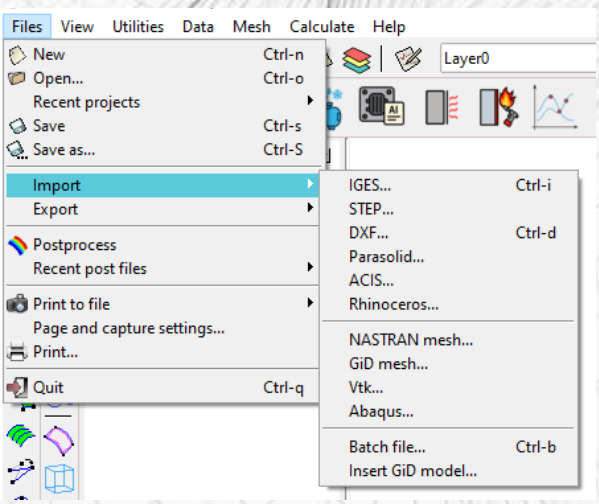
Utilities->Create volume from surface



Mesh import

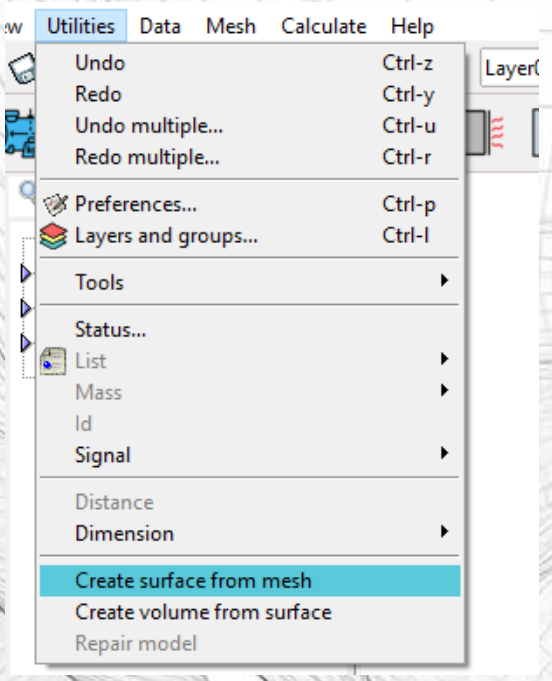
The user can directly import a mesh created in another FEM program or in a FEM pre/post-processing program.
The type of mesh to be entered, suitable for performing the simulation, must be a shell mesh of 3-node triangular elements or a volume mesh of 4-node tetrahedral elements.
Several formats are available for importing meshes: NASTRAN, GiD, Vtk, and Abaqus.

Files->Import



If the user inputs a shell mesh, they will need to convert that mesh into a surface geometry using the "Create surface from mesh" tool, and then convert that shell geometry into a volume using the "Create volume from surface" tool. Once the volume has been generated, the terahedral element mesh can be created.

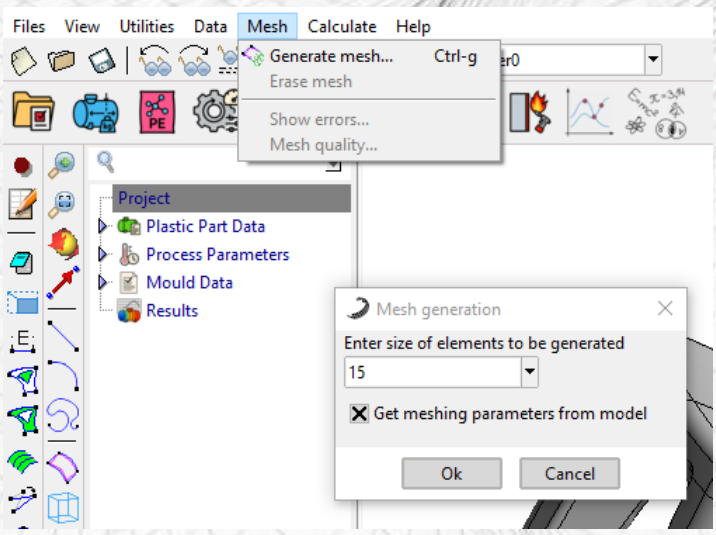
Utilities->Create surface from mesh



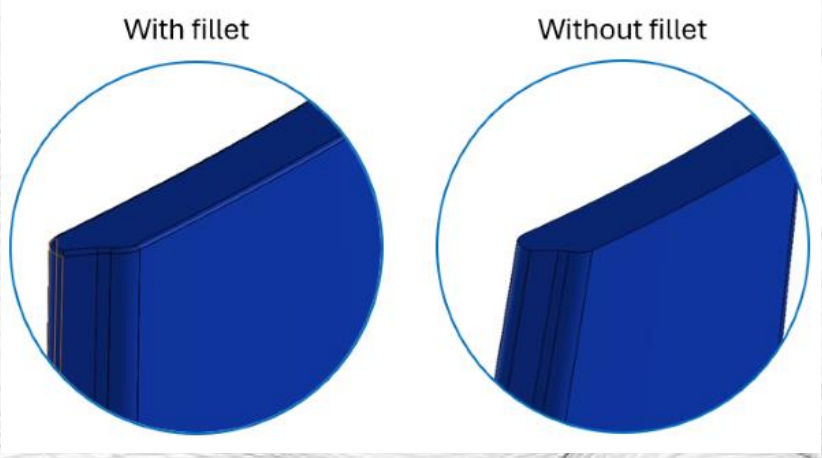
Generate mesh

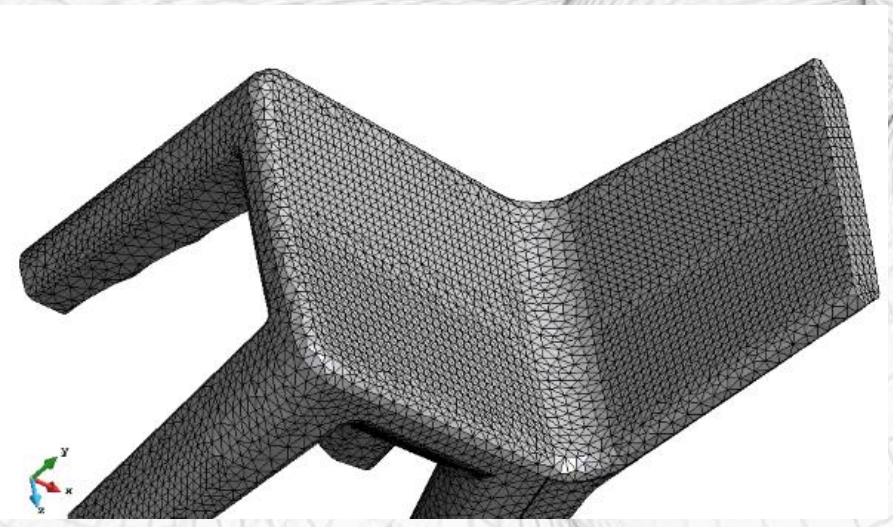
On the geometry entered or converted to a solid part, the tetrahedral element mesh must be generated by choosing the average element size with which the mesh will be generated.

Mesh->Generate mesh...



When generating the finite element mesh, it is important to carefully choose the element size. A mesh that is too coarse can compromise the quality of the results, while an excessively fine mesh can generate an unnecessary number of elements, increasing computational costs without providing significant improvements.
It is important that the mesh be as homogeneous as possible, with elements of similar size and smooth transitions between areas of different densities.
Abrupt variations in element size or shape can negatively affect the stability and accuracy of the solution.
Therefore, before generating the mesh, it is recommended to review and simplify the model geometry, eliminating small details that do not influence the results.
Areas such as rounded corners, very fine fillets, small cavities, or surface inscriptions can unnecessarily increase the number of elements without providing significant improvements in the simulation.
Maintaining clean and simplified geometry not only makes meshing easier, but also improves calculation stability and efficiency.





Define project

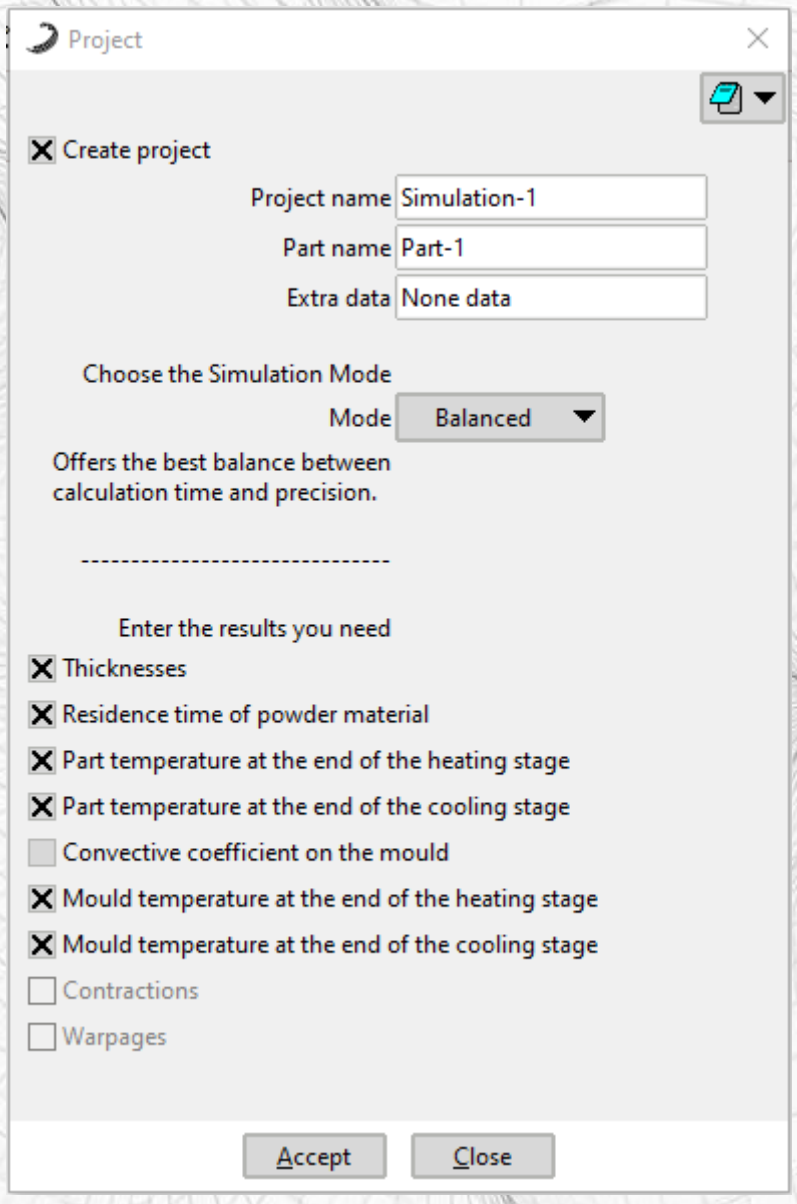
After entering the geometry, a "Project" must be created to define the parameters for the simulation.

To create project, open the "Project" window

Data->Project

Or use the following icon of the **SIMOULDING** Toolbar: 

In the "Project" window, the user must select the results they wish to obtain from the simulation, in addition to entering a specific name if desired.



Plastic Part Data

Part features

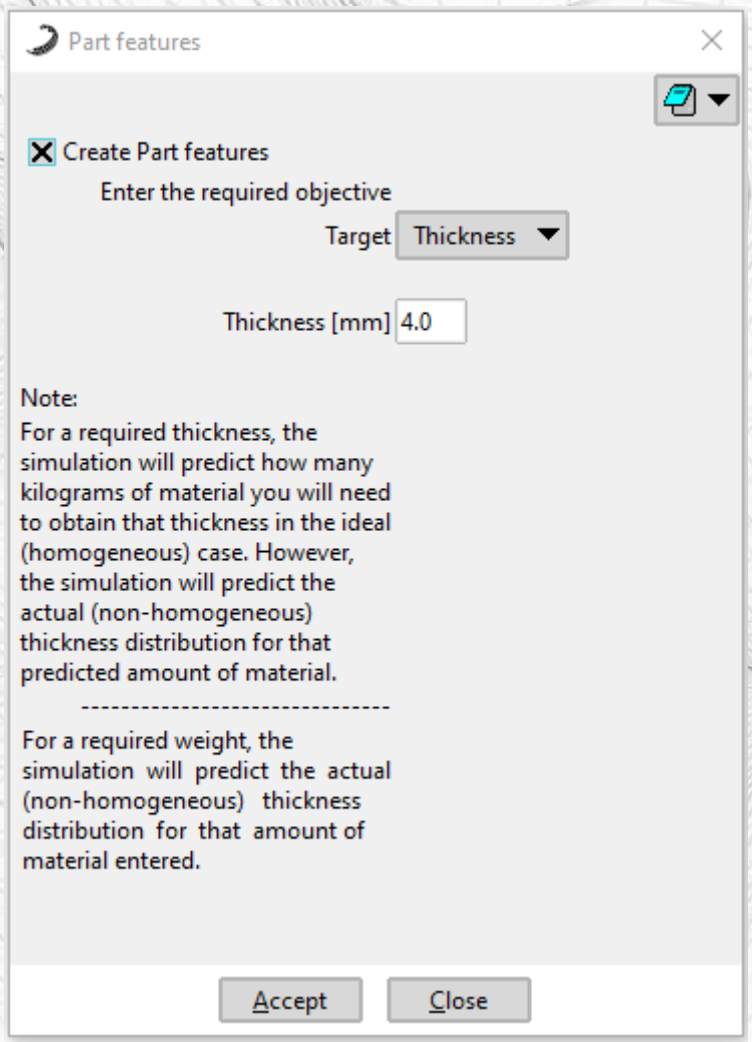
A "Part features" must be created with the desired objectives for the part.

To create part features, open the "Part features" window

Data->Plastic Part Data->Part features

Or use the following icon of the **SIMOULDING** Toolbar: 

In the "Part features" window, the user can choose a specific thickness in mm or a specific quantity of material in kg.



Polymers

The user must select a polymer for the simulation to be performed, and then assign it to the geometry or mesh.

To choose and assign a polymer, open the "Polymers" window

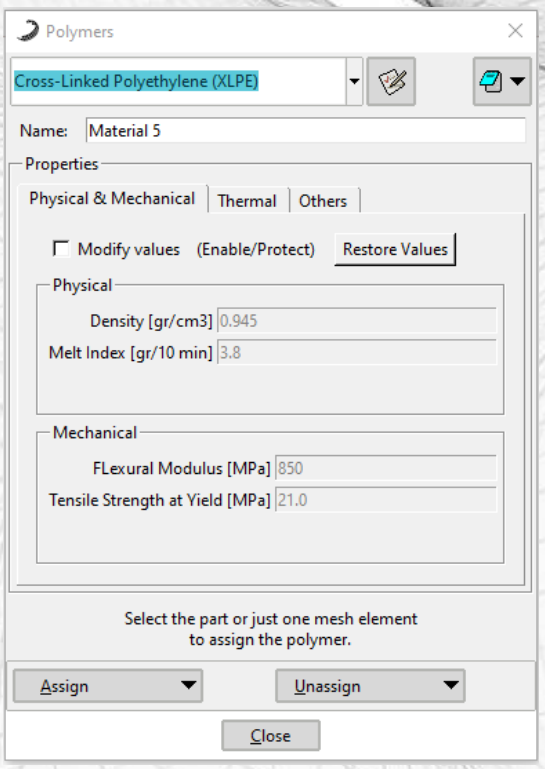
Data->Plastic Part Data->Polymers

Or use the following icon of the **SIMOULDING** Toolbar: 

SIMOULDING comes with 9 default polymers loaded.
The user can choose to modify, if desired, each of the values of the properties of the chosen polymer,
according to how best they fit the values of the material provided by their material supplier.

The polymers loaded by default are the following:

- **MDPE** Medium-Density Polyethylene
- **LDPE** Low-Density Polyethylene
- **HDPE** High-Density Polyethylene
- **LLDPE** Linear Low-Density Polyethylene
- **XLPE** Cross-Linked Polyethylene
- **PP** Polypropylene
- **NY** Polyamide PA12 / Nylon
- **EVA** Ethylene Vinyl Acetate
- **PC** Polycarbonate



Once the polymer and its appropriate property values have been chosen, the polymer must be assigned to the geometry.
The assignment can be made to the part, provided it is a solid part, or on the mesh by selecting at least one element.
The user can unassign the assigned polymer at any time or assign another polymer. The last assigned polymer will be validated for the simulation.
If the assignment is made to one or more mesh elements, to reassign another polymer so that it prevails over the previous one, it must be assigned to the same element(s) to which the first assignment was made. Otherwise, two or more polymers will remain assigned; however, the simulation will consider the last assigned polymer as valid.
For added security, if the user assigns the polymer to the mesh, to unassign it use the unassign tool in the Polymers window.

Process Parameters

Machine parameters

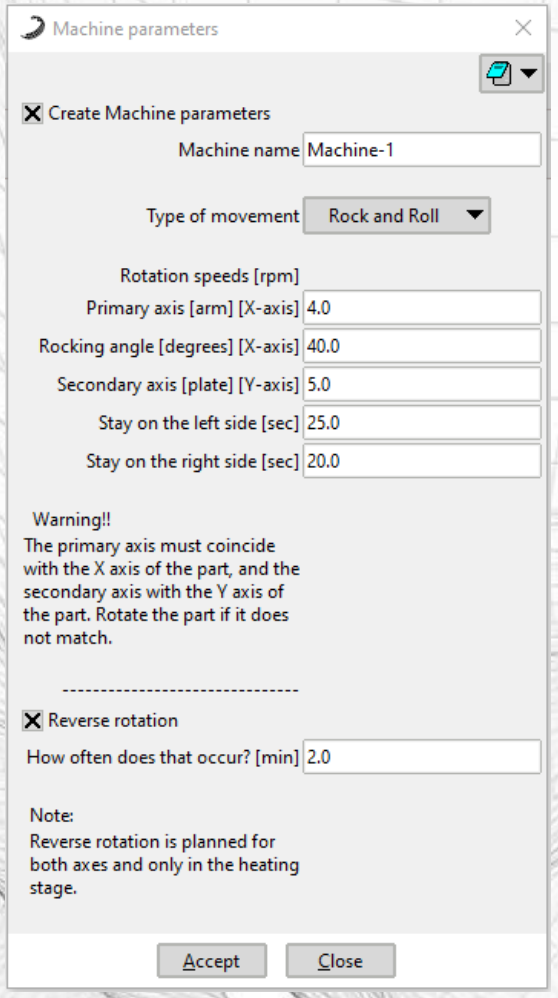
In the "Machine parameters" window, the data related to the type of machine and rotation speeds must be entered.

To enter the machine parameters, open the "Machine parameters" window

Data->Process Parameters->Machine parameters

Or use the following icon of the SIMOULDING Toolbar: 

SIMOULDING offers the option of choosing between a machine with Biaxial 360° or Rock and Roll movement.
The rotational speeds of the main arm and the platters must be entered.
In the case of Rock and Roll motion, the rocking angle and the dwell time at both ends must also be entered.
The reverse rotation option is available for both movements, allowing you to enter how often the rotation is reversed.
In the case of Biaxial 360° movement, the reverse rotation is performed for both the main axis and the secondary axis (plates). For Rock and Roll movement, the reverse rotation is only performed for the secondary axis.



Heating stage

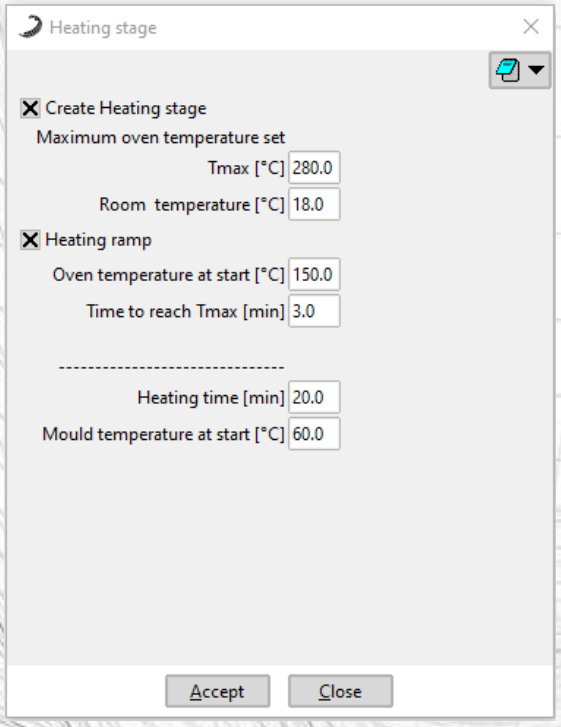
In the "Heating Stage" window, the data related to the heating of the part must be entered.

To enter the heating stage parameters, open the "Heating stage" window

Data->Process Parameters->Heating stage

Or use the following icon of the **SIMOULDING** Toolbar: 

In the "Heating Stage" window, the user can enter the maximum temperature to which the part will be heated and the time it will remain in the oven, in addition to the room temperature.
The option to enter data to generate a linear heating ramp for the oven is available, from the moment the part enters until the oven reaches maximum temperature.
The user can also enter the mold temperature at the start of the heating stage, which is very important when the mold is in production. If the user leaves this field at 0°C, the room temperature will be used as the value.



Cooling stage

In the "Cooling Stage" window, the data related to the cooling of the part must be entered.

To enter the cooling stage parameters, open the "Cooling stage" window

Data->Process Parameters->Cooling stage

Or use the following icon of the **SIMOULDING** Toolbar:

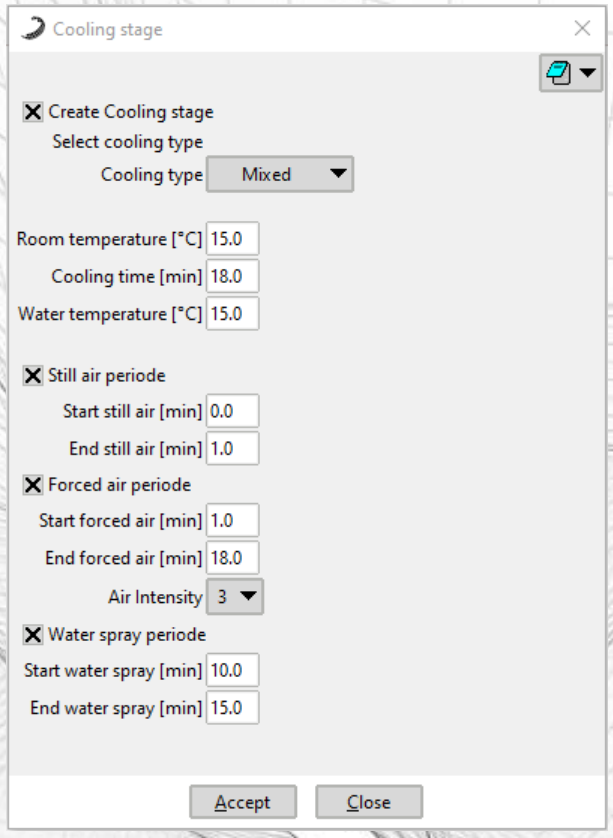
In the "Cooling sateg" window, the user can choose between 4 possible cooling types:

- Still air
- Forced air
- Water spray
- Mixed

For air-only cooling options, the room temperature and desired cooling time must be entered.
Note that the room temperature is also requested during the heating stage, but the two may not be the same,
especially in plants where there are significant variations between the moulds cooling zone and the oven entry zone.

For options where water spray will be used during cooling, the water temperature is required.
In the case of a mixed cooling, the user can choose any of the 3 types of cooling above, indicating the time period in which each occurs, within the total cooling

interval.
Water cooling may overlap with either of the two types of air cooling. The user must be careful when loading the periods,
ensuring that the total cooling time is not exceeded, nor that "Still air" and "Forced air" are overlapped.



Mould Data

Mould features

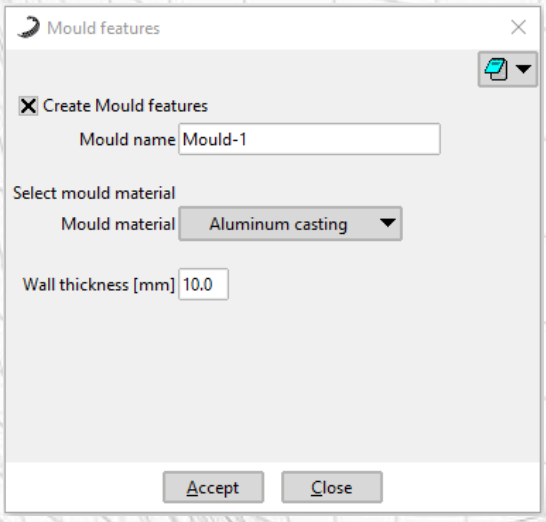
A "Mould features" must be created with the mold parameters for simulation.

To create mould features, open the "Mould features" window

Data->Mould Data->Mould features

Or use the following icon of the **SIMOULDING** Toolbar: 

In the "Mould features" window, the user can choose one of the available materials for the mold:
Aluminum casting, Aluminum alloy, Sheet steel and Sheet stainless steel.
The user will also need to enter the mold wall thickness and a name if you wish.



Insulation areas

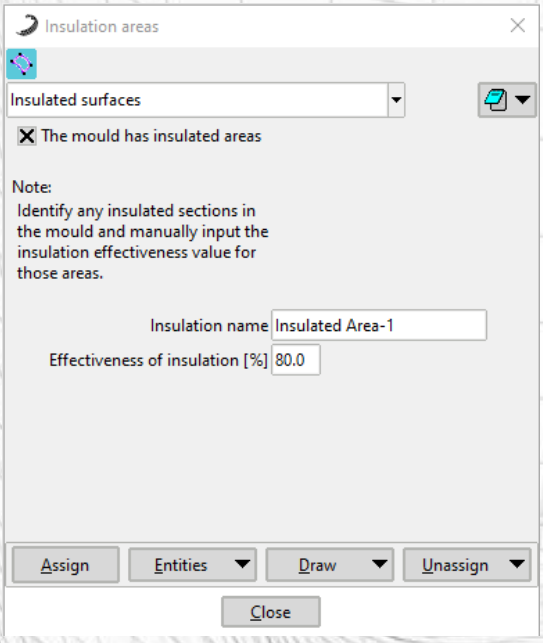
One of the optional parameters that **SIMOULDING** has is the ability to identify areas on the mould where the mold has insulation to prevent overheating and, consequently, polymer adhesion.
The user can use standard **GiD** tools to edit the geometry and create partitions on the faces and generate new faces that fit the size and shape of the insulation patch.

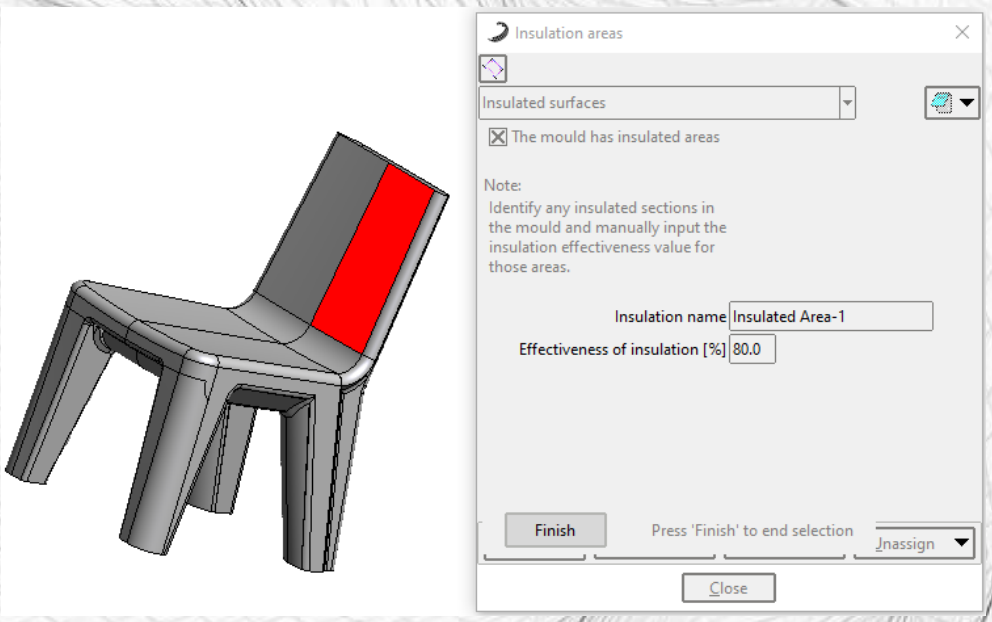
To add insulation areas, open the "Insulation areas" window

Data->Mould Data->Insulation areas

Or use the following icon of the **SIMOULDING** Toolbar:

In the "Insulation areas" window, the user can enter the insulation effectiveness and select the zone(s) that belong to that group of insulated areas.
The desired number of areas and groups of areas can be selected.
The insulation effectiveness will affect the temperature the mould receives in those areas.





Preheating areas

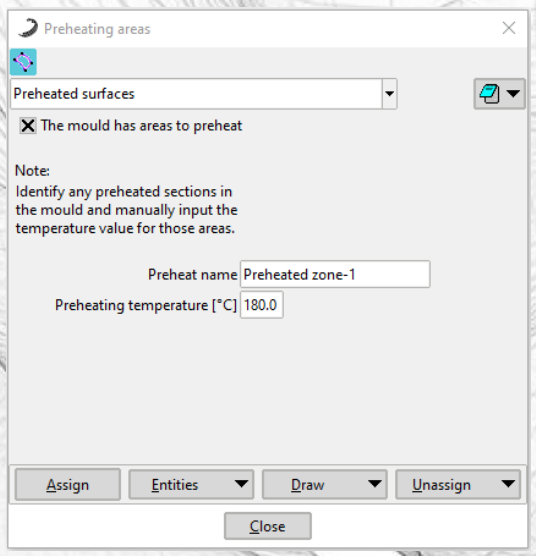
Preheating zones can be identified in areas of the mold where the heat does not fully act due to the geometry,
and therefore the convective coefficient is very low.
If the rotomolder chooses to preheat these areas before the mold enters the oven to promote material adhesion,
this parameter can be entered into the simulation as one of the optional **SIMOULDING** parameters.
The user can use standard **GiD** tools to edit the geometry and create partitions on the faces and generate new faces that fit the size and shape of the preheated zones.

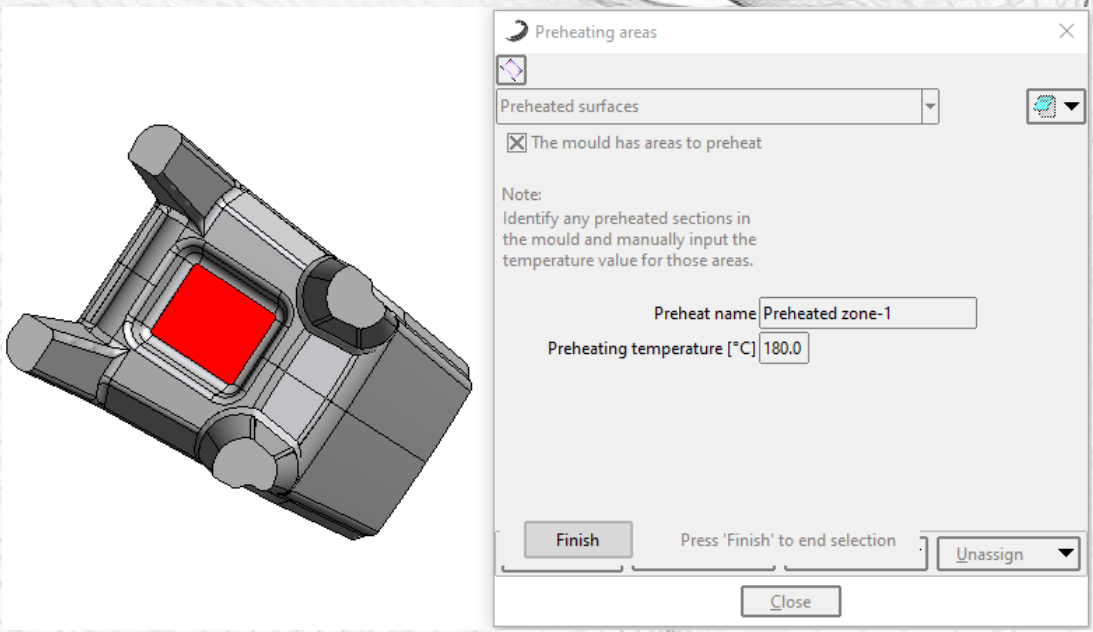
To add preheated areas, open the "Preheating areas" window

Data->Mould Data->Preheating areas

Or use the following icon of the **SIMOULDING** Toolbar: 

In the "Preheating areas" window, the user can enter the preheated temperature and select the zone(s) that belong to that group of preheated areas.
The desired number of areas and groups of areas can be selected.
The preheating temperature will be taken as the initial temperature of that zone at the beginning of the heating stage.





Points of interest

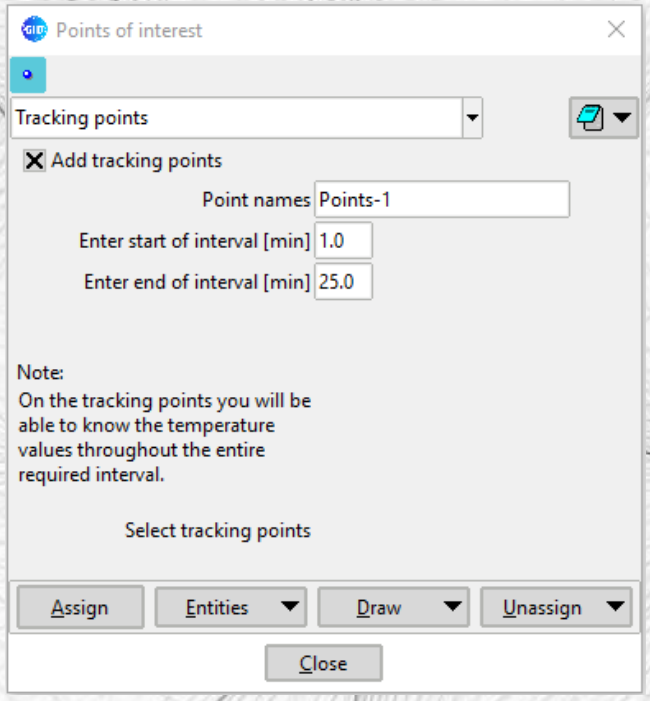
SIMOULDING offers the ability to monitor temperature evolution at any point in the mould geometry. The user will be able to select the points of interest they wish to track at the desired interval.

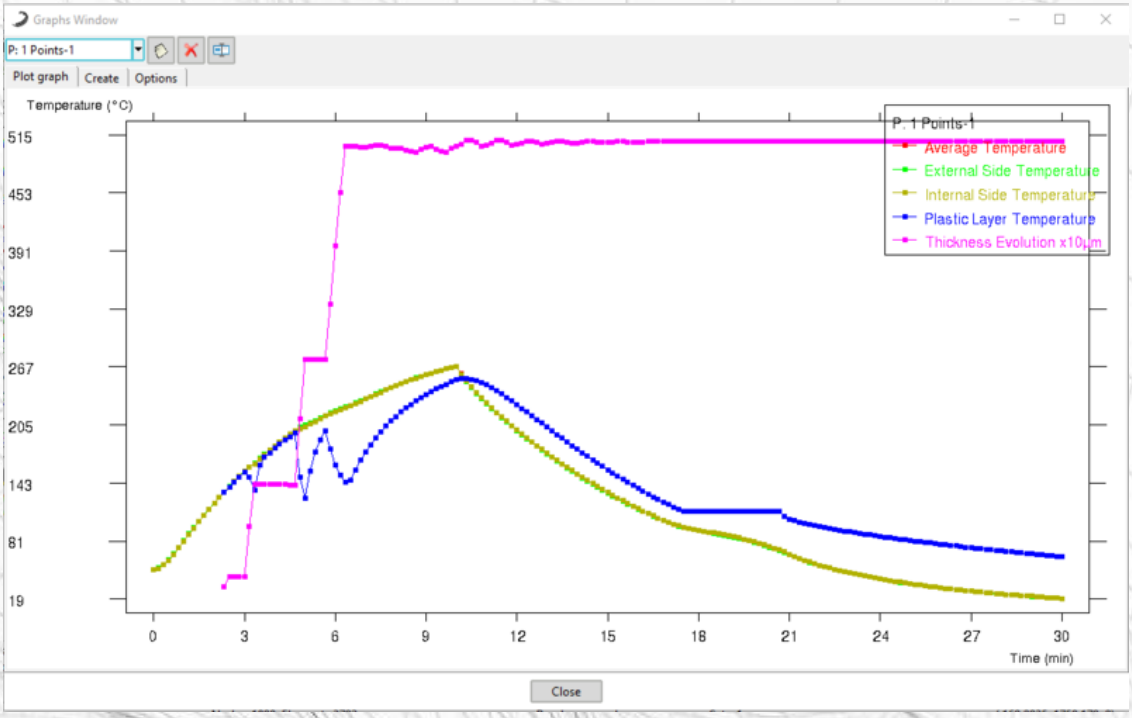
To add points of interest, open the "Points of interest" window

Data->Mould Data->Points of interest

Or use the following icon of the SIMOULDING Toolbar: 

In the “Points of interest” window, the user can enter the interval at which to track and select the point(s) that belong to that tracking point group.
The desired number of points and groups of points can be selected.





Simulate

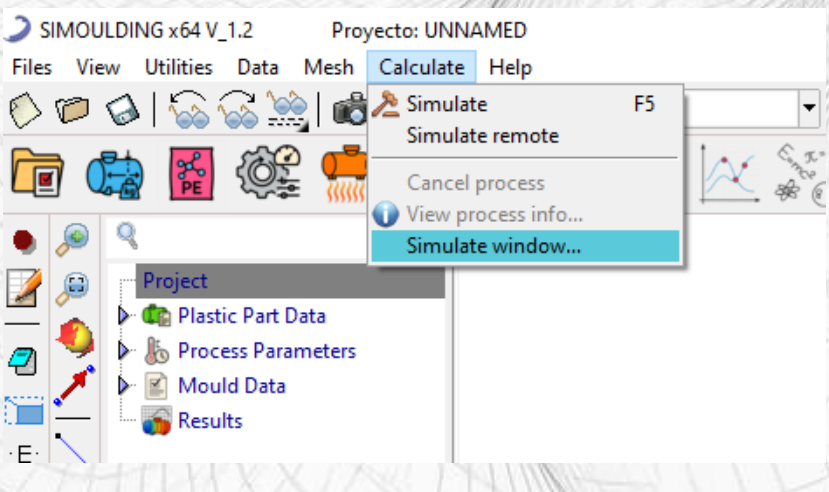
Once the user has generated the volume mesh and entered all the parameters required for the rotomoulding process, the simulation can be started.

To start the simulation you can do it from the context menu

Calculate->Simulate

Or use the following icon of the **SIMOULDING** Toolbar:

The user can follow the progress of the simulation in the Simulate window...

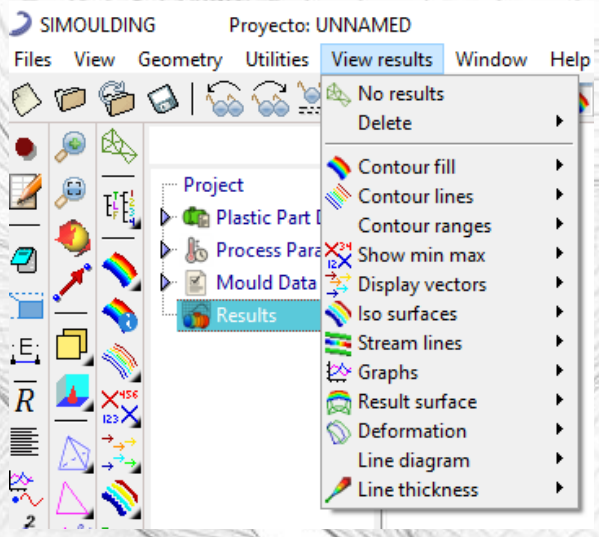


Results file

The simulation results will be printed in a text file that **SIMOULDING** will use to display them in the post-processing section.
To access the results, the user must use this icon located on the top right of the graphical interface's toolbar: 

The user will be able to select the result they wish to view from the menu, mapping its distribution on the surface of the part.

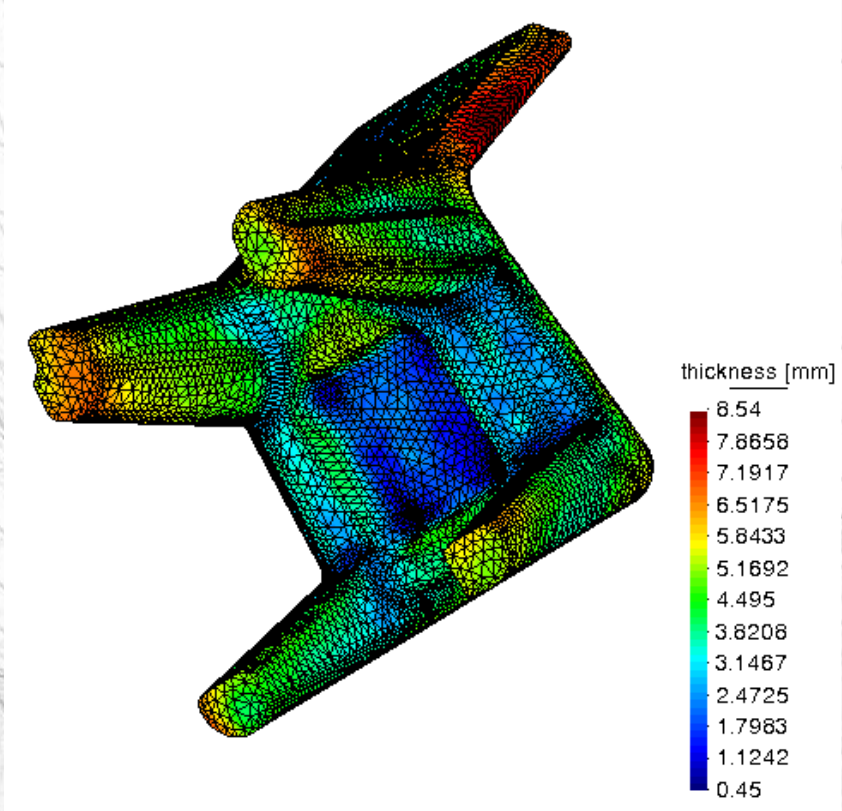
View results



The results that can be viewed will be those chosen in the "Project" window, in addition to the graphs of the added tracking points.

The available results are as follows:

- Thicknesses
- Residence time of powder material
- Part temperature at the end of the heating stage
- Part temperature at the end of the cooling stage
- Part temperature 5 minutes after demolding
- Convective coefficient on the mould
- Melt fraction
- Solidification frantion
- Mould temperature at the end of the heating stage
- Mould temperature at the end of the cooling stage
- Tracking points graphicals


thickness [mm]
8.54
7.8658
7.1917
6.5175
5.8433
5.1692
4.495
3.8208
3.1467
2.4725
1.7983
1.1242
0.45