Autodesk® Moldflow® Insight 2012

AMI Solver Parameters

Revision 1, 18 March 2012.

This document contains Autodesk and third-party software license agreements/notices and/or additional terms and conditions for licensed third-party software components included within the product. These notices and/or additional terms and conditions are made a part of and incorporated by reference into the Autodesk Software License Agreement and/or the About included as part of the Help function within the software.

Contents

Solver parameters 1

Solver parameters allow you to configure the advanced settings in an analysis.

Solver parameters

Edit the solver settings for any given analysis to adjust tolerances and parameter defaults.

Editing solver parameters

During an analysis, solvers calculate simultaneous equations based upon the solver settings. Within the Process Settings Wizard, you can alter the numerical parameter defaults and solver tolerances for the selected molding process.

Common solver parameters and process settings can be changed with the Process settings wizard. For other parameters you need to access the advanced options page.

NOTE: Typically, the solver parameters should remain at their default values.

---

1 Open the Process Settings Wizard.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard with a button called **Advanced options...**.

TIP: If the process sequence contains Fill+Pack, it is on that page. Otherwise it is on the first page of the Process Settings Wizard.

---

3 Click **Advanced options...**

4 Click **Edit** next to the **Solver parameters** drop-down list.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Mesh/Boundary and Mesh tabs 2

For an accurate simulation, it is important that a minimum number of layers is defined through the thickness. The higher the number of laminae, the more accurate the result.

Although the accuracy of the result will improve with more layers through the thickness, up to a point, each additional layer will increase the analysis time and computer resource requirements.

The boundary conditions for the analysis defines how the heat transfer coefficients between the mold and the material are modelled. The default option allows you to enter specific values to different phases. The profile option allows you to define how the heat transfer coefficient (HTC) for the entire mold varies with time.

Mesh/Boundary and Mesh tabs

The **Mesh/Boundary** tab dialog and the Mesh tab are both used to edit the number of layers through the part and the heat transfer coefficients between the mold and the melt. The Mesh/Boundary tab is available for Midplane or Dual Domain mesh types, and the Mesh tab is available for 3D mesh types.

Editing the Mesh/Boundary or Mesh tabs

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Mesh/Boundary** or **Mesh** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Mesh/Boundary and Mesh tabs

Use this dialog to specify Mesh/Boundary and Mesh values.

The Mesh/Boundary tab is only available for Midplane and Dual Domain analysis types. The Mesh tab is available for the 3D mesh type. The inputs available for edit depend on the molding process that you have selected.

Mesh/Boundary tab

The **Mesh/Boundary** tab of the **Reactive Molding solver parameters (Midplane/Dual Domain)** dialog is used to specify values for mesh and mesh boundary related analysis inputs for a Midplane or Dual Domain Reactive Molding analysis.

Number of laminae across thickness	Specifies the number of layers across the thickness of the part to be analyzed. The number of laminae across thickness that you select determines the computational intensity of the analysis. Select the number of laminae across the part thickness. **NOTE:** For calculation speed and data storage efficiency, if symmetric thermal boundary conditions exist, Fill+Pack uses a half-gap calculation and the actual number of laminae used in the calculations is one-half of that specified.
Mold-melt Heat Transfer Coefficient (HTC)	Defines how the heat transfer coefficient between the mold and the material is modeled.
Heat transfer coefficient between pot and pellet (side)	This parameter defines the heat transfer coefficient (h) at the interface between the transfer pot and the pellet (side). If the heat transfer coefficient is zero, then there is no heat exchange between the pot and the pellet (side). Alternatively, if the heat transfer coefficient nears infinity, then there is perfect thermal contact between the pot and the pellet (side). Enter the required pot-pellet (side) heat transfer coefficient.
Heat transfer coefficient between pot and pellet (bottom)	This parameter defines the heat transfer coefficient (h) at the interface between the transfer pot and the pellet (bottom). If the heat transfer coefficient is zero,

	then there is no heat exchange between the pot and the pellet (bottom). Alternatively, if the heat transfer coefficient nears infinity, then there is perfect thermal contact between the pot and the pellet (bottom). Enter the required pot-pellet (bottom) heat transfer coefficient.

Mesh tab dialog

The **Mesh** tab of the **Solver Parameters** dialog is used to specify values for mesh related analysis inputs for a 3D mesh types.

Number of laminates across beam radius	If you require higher accuracy in shear heating calculations or in predicting gate freeze times, you should raise the number of laminates across the beam radius. To increase solution speed you may choose to lower the number of laminates across the beam radius. However, this only affects the beam elements of a 3D model and so will not significantly reduce the total 3D Flow solution time.
Mold-melt Heat Transfer Coefficient (HTC) values (3D)	The Mold-melt Heat Transfer Coefficient (HTC) values group is used to specify the heat transfer coefficient (HTC) between the mold and the material. The HTC is specified as a boundary condition on the mesh used in 3D Fill+Pack and Cool analyses.

HTC Settings dialog

This dialog is used to specify the heat transfer coefficient (HTC) between the mold and the material.

You can set different values for the HTC to apply during filling, during packing, and when the part is detached (i.e., pressure is zero). You can set a different detached HTC for both the core and the cavity, for example, to model molding processes where the part remains in the cavity after the mold separates and before ejection.

HTC Profile dialog

This dialog is used to specify the heat transfer coefficient (HTC) between the mold and the material. The HTC is used in Fill+Pack and Cool analyses by Midplane and Dual Domain solvers.

You can set different values for the HTC based on the time since the start of the cycle. You can set a different HTC profiles for both the core and the cavity, for example, to model molding processes where the core is made from a different material than the cavity, or where the part remains in the cavity after the mold separates and before ejection.

Local heat transfer coefficients dialog

The mold-melt heat transfer coefficients for a selected area can be defined for different phases of the injection cycle.

There are different heat transfer characteristics for the filling phase and the packing phase of the injection cycle.

The **Detached** value is the Heat Transfer Coefficient used from when the pressure on the mold wall equals zero until the end of the injection cycle.

Intermediate output tab 3

Intermediate output refers to how often the simulation results are updated during the analysis.

Intermediate results can be written for both the

- Filling phase, and the
- Packing phase

at constant intervals or at specified times. You can also choose to update the results display during the analysis so you can see the part filing with time. For a more accurate representation of the molding process within a critical part of the process, you may want to increase the frequency that the intermediate results are recorded.

Intermediate output tab

The **Intermediate Ouput** tab dialog is used to edit how often intermediate results are written.

Editing the Intermediate Output tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Intermediate Output** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Intermediate output tab

This dialog is used to specify how often intermediate result files are written.

This tab is only available for Midplane and Dual Domain analysis types. The inputs available for edit depend on the molding process that you have selected.

Intermediate Output tab

Dynamically update results display during analysis

You can view intermediate results while an analysis is running simply by selecting the result in the Study Tasks pane as you would for a completed analysis.

If this option is selected, the results display will update every time the solver delivers a new set of intermediate results.

Clear this option if you do not want this automatic refresh during the analysis to occur.

Filling/packing phase intermediate results

These settings are used to specify the times during the filling phase, or packing phase, at which you want the solver to output regular and/or profiled intermediate results.

NOTE: For a reactive analysis, the "packing phase"settings relating to the curing phase of the analysis.

Regular intermediate results are used in the animation of contour results over time and in the display of XY plots of a variable against time. Profiled intermediate results are used in the display or animation of certain results as a function of normalized thickness, that is, through the thickness of the part.

The following three options are provided for the regular intermediate results and the profiled intermediate results respectively:

None	Select this option if you do not want the solver to output this type of intermediate result. Note that this will restrict the range of post-processing options that will be available to you.
Write at	Select this option to instruct the solver to output a fixed

constant intervals	number of sets of intermediate results at regularly spaced intervals. Increasing the value will increase the number of animation frames available when animating certain results through time, but will also increase the disk space requirements for the analysis.
Write at specified times	Select this option if you want to enter specific times at which you want the solver to output sets of intermediate results. For example, you may know the time at which the gate freezes and want the solver to output more animation frames at times close to that event.

Results Output Settings dialog

This dialog is used to specify the times at which intermediate results should be written.

This dialog is accessed from the **Intermediate Output** tab of the **Solver Parameters** dialog. Select **Write at specified times** from any of the drop-down menus, then click **Edit times** to open this dialog.

Convergence tab 4

Calculating results is an iterative process, performed at sequential time steps. With each iteration the results converge to a more accurate answer. The more iterations you perform, the more accurate the result, but the longer it takes to get a result.

In order to calculate a result, there are a number of factors that must be taken into account, such as the melt temperature, the flow rate, the viscosity of the material and the pressure. As the melt flows through the mold, each of these numbers will change from what they were in the previous time step. In order to simulate the flow accurately, these actual numbers, which change from one time step to the next, are be used in a series of iterative calculations.

The default settings provide an accurate result within a reasonable timeframe. It is possible to change the default settings to obtain a more accurate result at the expense of a longer analysis time. The settings that can be changed depend on the molding process and analysis sequence chosen, and will be a subset of the following:

- Pressure tolerance as a percentage change from one iteration to the next,
- Flow rate tolerance as a percentage change from one iteration to the next,
- Melt temperature tolerance as a percentage change from one iteration to the next,
- Conversion as a percentage change from one iteration to the next,
- Maximum melt temperature iterations,
- Maximum flow rate iterations,
- Maximum melt temperature and conversion iterations,
- Nodal growth mechanism,
- Viscosity treatment at high shear rates.

Convergence tab

The **Convergence** tab dialog is used to edit the number of iterations and accuracy of the result.

Editing the convergence tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Double-click  **Process Settings** to open the **Process Settings Wizard**.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.

3 Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **Convergence** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Convergence tab

Use this dialog to specify solver convergence values.

This tab is only available for Midplane and Dual Domain analysis types. The inputs available for edit depend on the molding process that you have selected.

Convergence Tab

The **Convergence** tab of this dialog is used to specify values for solver convergence related analysis inputs for Midplane or Dual Domain mesh types.

Pressure convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged.
Flow rate convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged.
Melt temperature convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged.
Conversion convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged.

	As soon as this % change value falls below the convergence tolerance, the solution has converged.
Maximum number of melt temperature iterations	Specifies the maximum number of iterations that the program will perform in order to solve the melt temperature equations. The program will continue until either the maximum number of iterations have been surpassed, or the error limit is less than the specified value.
Maximum number of flow rate iterations	Specifies the maximum number of iterations that the program will perform in order to solve the flow rate equations. The program will continue until either the maximum number of iterations have been surpassed, or the error limit is less than the specified value.
Maximum number of melt temperature and conversion iterations	Specifies the maximum number of iterations that the program will perform in order to solve the simultaneous equations for melt temperature and conversion.
Nodal growth mechanism	The nodal growth scheme controls the size of each flow front advancement step taken in a Dual Domain Flow analysis. The **Multiple** option (default) allows multiple bands of elements to be filled in a single time step. The **Single** option allows only a single band of elements to fill each time step. Select Single if you wish to increase the accuracy of the fill pattern prediction but note that the computation time will be increased.
Viscosity treatment at high shear rates	For certain materials with high temperature or shear sensitivity, extrapolation of viscosity values to high shear rates may introduce instability in the flow calculations. This option relates to a corrective algorithm that eliminates the instability at high shear rates.

Restart tab 5

Restart files provide a snapshot of the simulation during an analysis and are used to safeguard against system failure, to pause long analyses, and to extend analyses.

Every time a new restart file is created, the previous one is deleted to minimize the impact on disk space requirements.

The number of restart steps in both the filling and packing phase can be edited. The more steps you save, the more up to date the data file will be, but the slower the analysis time, since the solver will need to ouput more data to file.

Restart tab

The **Restart** tab dialog is used to specify the number of restart steps in the filling phase and the packing phase.

Editing the Restart tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Restart** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Restart tab

Use this dialog to specify settings for the writing of restart files.

Restart tab

The **Restart** tab of this dialog is used to specify settings for the writing of restart files.

Number of restart steps in filling phase	Specifies the number of times, at regularly spaced intervals, that restart files will be written during the filling phase
Number of restart steps in packing phase	Specifies the number of times, at regularly spaced intervals, that restart files will be written during the packing phase

Interface tab 6

The **Interface** tab provides a means to interface with 3rd party programs, such as Abaqus, ANSYS, NASTRAN and LS-DYNA.

By default,

- Include part insert is set to YES
- Mesh model only interface is set to NO

If you plan to interface to a 3rd party program, you can specify additional setting for the following structural interace programs:

- Abaqus
- ANSYS
- NASTRAN
- LS-DYNA

Interface tab

The **Interface** tab dialog is used to set-up interace settings to 3rd party structural interface programs.

Editing the Interface tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Interface** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Interface tab

Use this dialog to specify solver parameters related to 3rd party program interfaces.

Interface tab dialog

The **Interface** tab dialog is used to specify values for inputs related to third-party program interfaces.

Include part insert	This option is set to **YES** by default. If the model includes a part insert, the insert will be accounted for in the model and result data passed to the interface.
Mesh model only interface	This option is set to **NO** by default. If this option is set to **YES**, only the mesh model will be passed to the interface, and no Autodesk Moldflow Insight result data will be included.
ANSYS options	Opens a dialog to specify additional settings for interface to ANSYS structural analysis program.
Abaqus options	Opens a dialog to specify additional settings for interface to Abaqus structural analysis program.
LS-DYNA options	Opens a dialog to specify additional settings for interface to LS-DYNA structural analysis program.
NASTRAN options	Opens a dialog to specify additional settings for interface to NASTRAN structural analysis program.

ANSYS Options dialog

The **ANSYS Options** dialog is used to specify values for settings related to the Autodesk Moldflow Insight interface to ANSYS structural analysis program for 3D thermoplastics analyses.

The **Separate finite element mesh for mold** option is set to **No** by default. This means the part mesh will be used as input to the interface.

You can set this option to **Specify it**. You will be prompted to specify the file containing the 3D mold mesh you wish to use as input to the interface, and to choose what part data (pressure history, temperature, or both pressure and temperature data) will be passed to the mold mesh.

To access this dialog and specify settings for the current study, ensure that you have selected an analysis sequence that includes Fill+Pack, click  (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, click **Advanced options**, click **Edit** in the Solver parameters group, click **Interface** (tab), and click **ANSYS options**.

Specify filename dialog

This dialog is used to specify the mold mesh that you would like to use as input to the ANSYS interface.

To access this dialog, select the **Interface** tab of the **Solver Parameters** dialog, click **ANSYS options**, and choose **Specify it** from the drop-down menu. Click **Specify filename** to open the dialog.

Specify the

- **Mold mesh model filename**,
- **Select part data to pass onto the mold mesh**, and
- **Solution Method**; One step steady, or Multi-step transient.

Abaqus Options dialog

The **Abaqus Options** dialog is used to specify values for settings related to the Autodesk Moldflow Insight interface to Abaqus structural analysis program for Midplane thermoplastics analyses.

To access this dialog and specify settings for the current study, ensure that you have selected an analysis sequence that includes Fill+Pack, click  (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, click **Advanced options**, click **Edit** in the Solver parameters group, click **Interface** (tab), and click **Abaqus options**.

Select the relevant Abaqus interface file option, depending on the Abaqus license that you have. This will generate an output file(s) that can be read by Abaqus for analysis.

Options include:

None	No Abaqus interface file(s) will be produced.
C-MOLD - Abaqus 6.2	Is for Abaqus / C-MOLD license holders. The Flow solver will produce a *.fem file and *.osp file, which are ASCII files in the old C-MOLD format. You must change the extension **.osp** to **.lsp** for the file to be read in Abaqus 6.2.
Autodesk Moldflow Insight - Abaqus 6.2	Is for Abaqus / Autodesk license holders. The Flow solver will produce a *.mab file, which is an ASCII file of the same format as in MPI 2.0.
Autodesk Moldflow Insight - Abaqus 6.3	Is for Abaqus / Autodesk license holders using the Abaqus 6.3 (or later) release. The Flow solver will produce an *.osp file. You must export the mesh model in *.pat file format (Click **File > Export** and choose **Patran File (*.pat)** in the **Save as type** list).

NASTRAN options dialog

This option opens a dialog to specify additional settings for interface to the NASTRAN structural analysis program.

LS-DYNA Options dialog

The **LS-DYNA Options** dialog is used to specify values for settings related to the Autodesk Moldflow Insight interface to LS-DYNA structural analysis program.

The LS-DYNA Interface is available for Midplane thermoplastic injection molding analyses only.

Options include:

- Material model option
- Solution method
- Memory limit option
- Element formulation option
- Parallel process option

To access this dialog and specify settings for the current study, ensure that you have selected an analysis sequence that includes Fill+Pack, click  (**Home tab > Molding Process Setup panel > Process Settings**), if necessary click **Next** one or more times to navigate to the **Fill+Pack Settings** page of the Wizard, click **Advanced options**, click **Edit** in the Solver parameters group, click **Interface** (tab), and click **LS-Dyna options**.

Cool Analysis tab 7

The **Cool Analysis** tab of the **Solver Parameters** dialog is available to optimize cool analysis calculations according to your requirements.

The factors that are taken into consideration and the accuracy imposed on an analysis will impact on the speed and accuracy of an analysis. Parameters available for edit include:

Mold temperature convergence tolerance The default entry is 0.1

Maximum number of mold temperature iterations The default entry is 50

Include runners in automatic cooling time calculations The default entry is no.

Use aggregated mesh solver The default entry is yes.

Cool Analysis tab

The **Cool Analysis** tab dialog is used to specify what to include, and the accuracy of, the cool analysis calculations.

Editing the Cool Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1 Double-click  **Process Settings** to open the **Process Settings Wizard**.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.

3 Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **Cool Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Cool Analysis tab

Use this dialog to specify solver parameters for a Cool analysis.

This tab is only available for thermoplastic injection molding processes.

Cool Analysis tab dialog

The **Cool Analysis** tab of the **Solver Parameters** dialog is used to specify values for Cool analysis related inputs for a thermoplastics analysis using a Midplane or 3D mesh type.

The parameters available for edit depend upon the mesh type you have chosen.

Mold temperature convergence tolerance	Convergence tolerances apply to the % change in a function value from one iteration to the next, and are used to identify when a solution has converged. As soon as this % change value falls below the convergence tolerance, the solution has converged
Maximum number of mold temperature iterations	The amount of iterations that the program will perform in order to solve the simultaneous equations for mold-temperature. The program will continue until either the maximum number of iterations have been surpassed, or the error limit is less than the specified value.
Number of time steps for 3D cooling flux calculation	The 3D Cool analysis uses a full transient finite element method to calculate the temperature distribution in the part, from which the fluxes are derived. A larger number of time steps will lead to more accurate results, but will also require a longer solution time.
Include runners in automatic cooling time calculations	Specifies that the runner system is included when automatic cooling time is calculated.
Use aggregated mesh solver	Mesh aggregation is a feature that reduces the number of elements of the model internal to the solver, allowing for much faster Cool analysis times.
Calculate internal mold temperatures	When selected, this option enables the creation of data for the Temperature, internal mold result.

Cool Analysis Advanced Options dialog

This dialog is used to specify the Cool analysis related advanced options for the analysis sequence.

To access this dialog, ensure that you have selected an analysis sequence that includes only Cool, click  (**Home tab > Molding Process Setup panel > Process Settings**), then click **Advanced options**.

Molding material	Select and edit the material to analyze.
Process controller	Allows you to select and edit a process controller to control the injection molding process during the analysis. You can control the filling phase, velocity/pressure switch-over point, pack/holding phase, mold temperature and mold-open time.
Mold material	Allows you to select and edit the mold material to be used during the analysis. You can specify the density, specific heat, and thermal conductivity of the mold material.
Solver parameters	Allows you to select and edit the solver parameters to be used during the analysis.

Grid resolution dialog

The **Grid Resolution** dialog is used to specify the number of data points in the X, Y and Z direction to use when computing the temperature of the mold in the **Temperature, internal mold** result.

NOTE: To access this dialog, click **Cool Solver parameters** on the **Cool Settings** page of the **Process Settings Wizard**, and select the **Calculate internal mold temperatures** check box.

Cool (FEM) Analysis tab 8

The **Cool (FEM) Analysis** tab of the **Thermoplastics injection molding solver parameters (3D)** dialog is only available for models with 3D mesh types.

Here, you can select whether to run a steady-state cool analysis (**Averaged within cycle**) or a transient cool analysis (**Transient within cycle, Transient from production start-up**), using the Finite Element Method (FEM) solver. Once you have selected the type of cool analysis you would like to run, you can edit the process settings by clicking on the **Cool (FEM) Solver Parameters** button.

Cool (FEM) Analysis tab

The Cool (FEM) Analysis tab dialog is used to specify which solver to use for the heat flux calculation, and what tolerances to set.

Editing the Cool (FEM) Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Cool (FEM) Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Cool (FEM) Analysis tab

The **Cool (FEM) Analysis tab** dialog is used to specify whether to run a steady-state cool analysis using the Finite Element Method (FEM) solver, or whether to run a transient cool analysis, and if transient, what type of transient cool analysis to run. From this dialog, you can also access the Cool (FEM) solver parameters dialog, so you can change solver parameters.

Cool (FEM) Analysis tab dialog

This analysis sequence is only available for thermoplastic injection molding processes, with a 3D mesh type.

Mold temperature options	Select the type of analysis you would like; Averaged within cycle, Transient within cycle, or Transient from production start-up.
Cool (FEM) Solver Parameters	Specify whether to use the **Conduction solver** or whether to run a **Flow analysis on every iteration**. The conduction solver will be faster, but the flow solver may provide a more accurate simulation. The solver parameters are set here.

Warp Analysis tab 9

The Warp tab is used to define the type of warp analysis, the stress results to output and what to consider in the calculation.

For Midplane mesh types, there are 4 warp analysis types from which to select:

Automatic	This is the default setting, and is a good choice if you have no prior knowledge or expectation regarding your part.
Small deflection	Select this analysis type if you expect the warpage of the part to be stable. This analysis can be performed on all mesh types.
Large deflection	This analysis is suitable for Midplane and 3D meshed models and can be selected if you expect the warpage of the part ot be unstable.
Buckling	This analysis is used to determine whether the warpage of the part is stable or unstable.

You can decide whether to include cold runners, thermal expansion and corner effects during the calculation, and which matrix solver to use.

For 3D mesh types, you can also select whether to output the initial stress tensor before performing the analysis, so it can be used in another warpage analysis program.

Warp Analysis tab

The **Warp Analysis** tab dialog is used to specify what to include, and the accuracy of, the warp analysis calculations.

Editing the Warp Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Click **Tools tab > Databases > New** to open the **New Database** dialog.

2 Select **Parameters** from the **Category** drop-down menu, and choose the appropriate molding process and mesh type combination from the **Property type** menu. Click **OK**.

3 Click **New** in the **Properties** dialog to open the **Solver Parameters** dialog.

4 Select the **Warp Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Warp Analysis tab

Use this dialog to specify solver parameters for a Warp analysis.

Warp Analysis tab dialog

The **Warp Analysis** tab dialog is used to specify input values for a Warp analysis.

NOTE: Different inputs are available depending upon the molding process and mesh type you have chosen.

RESTRICTION: A Warp analysis can be performed only when you have selected an analysis sequence that includes the Compressible solver and a material that includes pvT properties data.

NOTE: Warp analysis inputs can be specified on the **Warp Settings** page of the **Process Settings Wizard**.

Warpage analysis type	Allows you to select the type of Warp analysis to run.
Stress result(s) to output	Specify which laminate-based stress-related results the solver will output.
Consider gate surface and cold runners?	Specifies whether cold runners and/or gate surface elements, if present in your model, are taken into consideration during Warp analysis.
Consider mold thermal expansion	Select this check box if you want Warp analysis to consider the effect of mold thermal expansion on the warpage and/or molded-in stress levels in the part.
Consider corner effects	Select this option if you want the Warp analysis to calculate and account for deformations due to mold-restraint induced differential shrinkage.
Matrix solver	Select the equation solver to be used in the Warp analysis.
Include tetrahedral cold runners	Specifies whether tetrahedral cold runners, if present in your model, are taken into consideration during Warp analysis.

Use mesh aggregation and 2nd-order tetrahedral elements	Controls whether 3D Warp analysis should employ mesh aggregation. This makes little difference for Warp analysis but produces a result more quickly.
Interface to other structural analysis	This option controls whether 3D Warp analysis outputs the initial stress tensor before performing the analysis. This initial stress tensor can then be used in another warpage analysis program.
Number of threads for parallelization	Specify the number of threads to be used for parallel solution.The default setting is Single thread (No parallelization).
Use AMG matrix solver	The AMG solver improves the speed of the analysis by using progressively coarser mesh grids for computations.

Stress Analysis tab 10

The Stress Analysis tab is used to specify the type of stress analysis, the parameters to include in the calculation and the matrix solver to use.

The Stress Analysis tab is only accessible for thermoplastic injection molding processes using a Midplane mesh type.

Small deflection	Small deflection analysis is the most common type of analysis and is the basis of both large deflection and buckling analysis. A thorough understanding of small deflection analysis is an important prerequisite for understanding the other analysis types.
Large deflection	The large deflection analysis provides the final deformed shape of the part, allowing for nonlinear stress-strain behavior within the part. In practice, most structures will exhibit a linear or approximately linear response only over a restricted range of load intensities. At higher loads the stiffness of the structure can alter significantly, leading to a non-linear response.
Buckling	A buckling analysis is used to determine whether the deformation of the part will be stable or unstable under the applied load(s). Whilst the full non-linear incremental/iterative method of following the response of a structure is completely general and relatively precise, it can also involve a great deal of computational effort.
Modal frequency	Select this analysis type if you want to determine the natural, undamped frequency response of the part. Theoretically, this analysis type is similar to the buckling analysis, however the physical interpretation of the results is different.
Creep	The creep behavior of the part is its time dependent deformation under the applied load(s).

Stress Analysis tab

The **Stress Analysis** tab dialog is used to specify the type of analysis and what parameters to include in a stress analysis.

Editing the Stress Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Click **Tools tab > Databases > New** to open the **New Database** dialog.
2 Select **Parameters** from the **Category** drop-down menu, and choose **Thermoplastic injection molding solver parameters (Midplane)** from the **Property type** menu. Click **OK**.
3 Click **New** in the **Properties** dialog to open the **Solver Parameters** dialog.
4 Select the **Stress Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Stress Analysis tab

Use this dialog to specify solver parameters for a Stress analysis.

Stress Analysis tab dialog

The **Stress Analysis** tab of this dialog is used to specify values for stress analysis related inputs for a thermoplastics analysis of a Midplane model.

Stress analysis type	Allows you to select the type of stress analysis to run.
Stress result(s) to output	Specify which laminate-based stress-related results the solver will output.
Consider gate surface and cold runners?	Specifies whether cold runners and/or gate surface elements, if present in your model, are taken into consideration during the stress analysis.
Consider mold thermal expansion	Select this check box if you want stress analysis to consider the effect of mold thermal expansion on the molded-in stress levels in the part.
Matrix solver	Select the equation solver to be used in the stress analysis.

Fiber Analysis tab 11

Fiber filled materials can display anisotropic behaviour. The direction of the fiber in the final part can affect the mechanical properties of the part in that direction.

The Fiber Analysis tab of the Solver Parameters dialog is used to determine which values are used in the calculation, the models used in the calculation and the fiber alignment at different points.

Fiber Analysis tab

The **Fiber Analysis** tab dialog is used to specify the values and models used in the calculation and the fiber orientation and various points.

Editing the Fiber Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Click **Tools tab > Databases > New** to open the **New Database** dialog.
2 Select **Parameters** from the **Category** drop-down menu, and choose the appropriate combination of molding process and mesh type from the **Property type** menu. Click **OK**.
3 Click **New** in the **Properties** dialog to open the **Solver Parameters** dialog.
4 Select the **Fiber Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Fiber Analysis tab

Use this dialog to specify solver parameters for a Fiber Orientation analysis.

Fiber Analysis tab dialog

The **Fiber Analysis** tab of the **Solver Parameters** dialog is used to specify values for Fiber orientation analysis.

NOTE: The parameters available to edit will depend on the molding process and mesh type selected.

NOTE: If the shrinkage model for the selected material is set to CRIMS, and the **Use CRIMS** option on the **CRIMS Shrinkage Model Coefficients** dialog is set to the default ("change solver parameters to be consistent with the CRIMS model"), any changes that you make to the solver parameters above will be overwritten in the analysis. If you want to use non-default settings for these solver parameters, either change the **Use CRIMS** setting, or select a different shrinkage model.

Calculate fiber orientation using	This option specifies the model used by the Fiber analysis solver to calculate fiber orientation.
Apply fiber inlet condition at	This option specifies whether the fiber orientation calculation begins at the part gate or at the injection location.
Fiber inlet condition	Allows you to specify the inlet boundary condition of the fiber orientation state.
Edge boundary condition	This option affects the way that fiber alignment is calculated in elements along the edge of the part.
Composite property calculation options	This dialog is used to edit options relating to the prediction of the mechanical properties of the composite, that is, fibers plus polymer matrix.
Coefficient of interaction (Ci)	Specifies the coefficient of interaction (Ci) value that will be used in the calculation of fiber interactions.
Reduced Strain Closure factor	Specifies the Reduced Strain Closure factor, which is used to model slow fiber orientation dynamics.

Miscellaneous tab 12

The **Miscellaneous tab** of the **Solver Parameters** dialog is used to set fiber orientation analysis settings

Fiber filled materials can display anisotropic behaviour. The direction of the fiber in the final part can affect the mechanical properties of the part in that direction. Modeling the fiber correctly allows these characteristics to be taken into account.

NOTE: This tab is only available for thermoplastic injection molding processes using a Midplane mesh type.

Miscellaneous tab

The **Miscellaneous** tab of the **Solver Parameters** dialog is used to set up parameters for a fiber orientation analysis.

Editing the Miscellaneous tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Click **Tools tab > Databases > New** to open the **New Database** dialog.
2. Select **Parameters** from the **Category** drop-down menu, and choose the **Thermoplastic injection molding solver parameters (Midplane)** option from the **Property type** menu. Click **OK**.
3. Click **New** in the **Properties** dialog to open the **Solver Parameters** dialog.
4. Select the **Miscellaneous** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Miscellaneous tab

Use this dialog to specify miscellaneous settings when running an analysis.

Thermoplastics Injection Molding Solver Parameters (Midplane) dialog—Miscellaneous

The **Miscellaneous** tab of this dialog is used to specify a selection of additional settings for a thermoplastics analysis of a Midplane model.

MuCell Analysis tab 13

The incorporation of a gas into the melt gives the final polymer an aerated structure. The bubble growth within the polymer needs to be defined.

The MuCell Analysis tab is used to specify the number of iterations that are used for the bubble growth calculation.

MuCell Analysis tab

Editing the MuCell Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Double-click **Process Settings** to open the **Process Settings Wizard**.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.

3 Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **MuCell Analysis** tab and edit as necessary.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

MuCell Analysis tab

Use this dialog to specify solver parameters for a Microcellular Injection Molding analysis.

MuCell Analysis tab dialog

The **MuCell Analysis** tab of this dialog is used to specify values for the inputs of a Microcellular injection molding analysis of a Midplane model.

Core Shift tab

14

Under the pressure of the injection process, the core half of the mold can be deflected. This will result in thickness variation of the part, and is a frequent problem with long, slender product such as vials, test tubes and pen barrels..

The Core shift simulation provides detailed information about the movement of the mold core and its interaction with the polymer flow process as the plastic is being injected.

The Core Shift tab provides an opportunity to decide whether or not to run an core shift analysis. If selected, you can set the frequency of core shift analysis, the matrix solver used and the accuracy of the calculation.

Core Shift tab

The **Core Shift** tab of the **Solver Parameters** dialog can be accessed from the **Process Settings Wizard**.

This tab is only accessible for Fill+Pack analyses using thermoplastic injection molding processes.

Editing the Core Shift tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Double-click  **Process Settings** to open the **Process Settings Wizard**.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.

3 Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **Core Shift** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Core Shift tab

Use this dialog to specify solver parameters for core shift prediction related inputs.

Core Shift tab dialog

The **Core Shift** tab of this dialog is used to specify values for core shift prediction related inputs for an injection molding analysis of a model and core (mold insert).

Perform Core shift analysis	If this option is selected, and the study contains elements of type Core (3D), the Fill+Pack analysis will run a number of Stress analyses to predict the deflections of the core.
Frequency of Core shift analysis	Pressures at different locations within the cavity vary greatly during the filling and packing phases, therefore, the Core shift analysis needs to be repeated several times to account for these pressure variations. These options determine at what interval the Core shift analyses are performed during the Fill+Pack analysis:
Maximum volume increment between analyses	Specifies the maximum change in %cavity filled during the filling phase of the analysis before another Core shift analysis is performed.
Maximum time step between analyses	Specifies the maximum change in filling or packing time before another Core shift analysis is performed.
Insert temperature	Specifies whether the part insert and core temperatures are set to a fixed value, or calculated by the Fill+Pack analysis.
Perform Core shift analysis during pressure iteration	During the iterations of the pressure solution in the Fill+Pack analysis, the pressure value changes which affects the core shift prediction. This option specifies whether additional Core shift analyses should be performed during the pressure solution iteration.
Surface matching tolerance	Specifies the maximum allowed distance between a core element and a neighboring part surface element for them to be associated with one another.
Percentage frozen layer that makes node constrained	Specifies the % frozen layer through the cavity thickness at which no further core deflection is expected to occur.

Update mesh during Core shift analysis	When the predicted core shift is large, the resulting mesh distortion may make the Fill+Pack analysis unstable, and the analysis will stop. If this is the case, turning this option off will allow the analysis to complete, but the solution will be somewhat less accurate.
Calculate Core shift after filling phase	If the **Calculate core shift after filling phase** option is not selected, then Core shift is only calculated during the filling phase. Whatever core displacement existed at the end of the filling analysis is assumed to stay unchanged during packing.
Analyze core using	Specifies which type of tetrahedral element to use for the core elements in the stress analysis of the core.
Use AMG matrix solver	The AMG solver improves the speed of warp analysis by using progressively coarser mesh grids for computations. Select **Yes** to enable the Algebraic Multigrid (AMG) matrix solver when performing a Core shift analysis.

Wire Sweep/ Paddle Shift and Dynamic Paddle Shift tabs 15

This setting is available for microchip encapsulation analyses using Midplane or Dual Domain mesh types.

The Wire Sweep analysis calculates the deformation of the bonding wires (connecting the chip to the leadframe) that occurs during encapsulation. This calculation enables you to improve the mold design and process conditions to prevent wire-sweep from occurring during encapsulation.

The Paddle Shift analysis calculates the deformation of the paddle due to the pressure difference in the two sub-cavities separated by the leadframe. Microchip Encapsulation calculates the pressure in the cavity. .

The deformation can be calculated either internally in Autodesk Moldflow Insightusing the Warp module, or externally using Abaqus

3D Microchip Encapsulation also supports a Dynamic Paddle Shift simulation where the paddle shift is recalculated several times during filling. This analysis can provide a more accurate prediction of the final paddle shift when large deformations occur. The dynamic paddle shift analysis includes an option to perform Core shift analysis during pressure iteration, which is valid if the paddle has been modeled using 3D elements. However, if the paddle has been modeled using shell elements, the additional Core shift analysis cannot be performed.

Wire Sweep/Paddle Shift and Dynamic Paddle Shift tabs

The Wire Sweep/Paddle Shift tab can be accessed from the Process Settings Wizard, if you are running a microchip encapsulation analysis with a Midplane or Dual Domain mesh type. The Dynamic Paddle Shift tab is accessible if you are modeling a 3D mesh type.

Wire Sweep/Paddle Shift and Dynamic Paddle Shift tab dialogs

NOTE: Typically, the solver parameters should remain at their default values.

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **Wire Sweep/Paddle Shift** tab or the **Dynamic Paddle Shift** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Wire Sweep/Paddle Shift and Dynamic Paddle Shift tabs

This Wire Sweep/Paddle Shift dialog is used to select the stress analysis type when setting up a Wire Sweep/ Paddle Shift analysis and to specify in which product to perform the simulation. The Dynamic Paddle Shift dialog is used to specify core shift inputs for a 3D mesh type.

This tab is only accessible for microchip encapsulation analyses, using a Midplane or Dual Domain mesh type.

Wire Sweep/Paddle Shift tab

The **Wire Sweep/Paddle Shift** tab of this dialog is used to specify imput parameters for wire sweep and paddle shift prediction for a Microchip Encapsulation analysis with a Midplane or Dual Domain mesh type.

NOTE: Wire deformation during the Microchip Encapsulation process can be quite large so we recommend that a Large Deflection Stress analysis be run in either Autodesk Moldflow Insight or Abaqus.

Stress analysis type	This option is used to select the type of Stress analysis to run to predict paddle shift/wire sweep.
Perform wire sweep/paddle shift simulation in	If the analysis sequence includes wire sweep and/or paddle shift simulation, this option is used to specify which solver will be used to perform this simulation.

Dynamic Paddle Shift tab

The **Dynamic Paddle Shift** tab of the **Reactive Molding solver parameters (3D)** dialog is used to specify values for paddle shift prediction related inputs for a 3D Reactive Molding analysis.

Frequency of Core shift analysis	Pressures at different locations within the cavity vary greatly during the filling and packing phases, therefore, the Core shift analysis needs to be repeated

	several times to account for these pressure variations.
Perform Core shift analysis during pressure iteration	During the iterations of the pressure solution in the Fill+Pack analysis, the pressure value changes which affects the core shift prediction. This option specifies whether additional Core shift analyses should be performed during the pressure solution iteration.
Update mesh during Core shift analysis	When the predicted core shift is large, the resulting mesh distortion may make the Fill+Pack analysis unstable, and the analysis will stop. If this is the case, turning this option off will allow the analysis to complete, but the solution will be somewhat less accurate.
Calculate Core shift after filling phase	If the **Calculate core shift after filling phase option** is not selected, then Core shift is only calculated during the filling phase. Whatever core displacement existed at the end of the filling analysis is assumed to stay unchanged during packing. If the **Calculate core shift after filling phase** option is selected, then Core shift calculations continue during packing also.
Analyze core using	Specifies which type of tetrahedral element to use for the core elements in the stress analysis of the core.
Use AMG matrix solver	The AMG solver improves the speed of warp analysis by using progressively coarser mesh grids for computations.

Fill+Pack and Flow Analysis tabs

16

The Fill+Pack Analysis and Flow Analysis tabs are used to specify the solver settings you would like to use during a Fill+Pack analysis, using a 3D mesh type.

In addition to the solver set-up, you can specify how frequently intermediate results and recovery data are written.

Fill+Pack and Flow Analysis tabs

These tabs are used to set up the solver parameters for a 3D Fill+Pack, fast fill, or 3D Flow analysis.

Fill+Pack or Flow Analysis tab dialogs

NOTE: Typically, the solver parameters should remain at their default values.

1. Click **Tools tab > Databases > New** to open the **New Database** dialog.
2. Select **Parameters** from the **Category** drop-down menu, and choose **Thermoplastic injection molding solver parameters (3D)** from the **Property type** menu. Click **OK**.
3. Click **New** in the **Properties** dialog to open the **Solver Parameters** dialog.
4. Select the **Fill+Pack Analysis** or **Flow Analysis** tab and edit the appropriate inputs.

   NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Fill+Pack and Flow Analysis tabs

Use these dialogs to specify solver parameters for a Fill+Pack (3D) or Flow (3D) analysis.

Fill+Pack Analysis tab dialog

The **Fill+Pack Analysis** tab of the **Solver Parameters** dialog is used to specify input values for Fill+Pack analyses of a 3D model.

Solver parameters	Specify solver parameters related to simulating inertia and gravity effects, GPU options, parallel solution, gate contact diameter, and filling and packing phase parameters.
Intermediate results	Intermediate results enable you to view certain analysis results (for example, pressure, temperature etc.) at various times during the filling and/or packing phase.
Recovery data	Recovery data is used to safeguard against system failure, by providing a snapshot of the simulation data at the time of writing.

Flow Analysis tab dialog

The **Flow Analysis** tab on this dialog is used to specify values for an Underfill Encapsulation analysis of a 3D model.

Solver parameters	Specify solver parameters related to simulating inertia and gravity effects, GPU options, parallel solution, gate contact diameter, and filling and packing phase parameters.
Intermediate results	Intermediate results enable you to view certain analysis results (for example, pressure, temperature etc.) at various times during the filling and/or packing phase.
Recovery data	Recovery data is used to safeguard against system failure, by providing a snapshot of the simulation data at the time of writing.

Solver Parameters (3D) dialog

This dialog is used to specify filling phase, packing phase, gate diameter and general solver parameters for the 3D flow solver used in a Fill+Pack or Flow analysis of a 3D model.

To access this dialog, choose an analysis sequence which includes Fill or Fill+Pack. Click **Home tab > Molding Process Setup panel > Process Settings**. Click **Next**, if necessary, until the Fill or Fill+Pack Settings page appears. Click **Advanced Options**, then click **Edit** in the Solver Parameters group. On the **Thermoplastics injection molding solver parameters (3D)** dialog, select the **Fill+Pack Analysis** tab. In the Solver setup group, select **Use coupled solver**, then click **Solver Parameters**.

All solver parameters have a default value that will be suitable for most analyses.

Simulate inertia effect	Specify whether you want the effects of inertia to be included in the Navier-Stokes calculation. Selecting this option will improve analysis accuracy for thick parts, but is the most compute-intensive option.
Simulate gravity effect	Specify whether you want the effects of gravity to be included in the Navier-Stokes 3D flow simulation calculation.
GPU options	Specify whether you want numerical calculations to be performed on a GPU (Graphics Processing Unit) card when running an analysis. The default setting is to **Use GPU if available**.
Number of threads for parallelization	Specify the number of threads to be used for parallel solution.
Gate contact diameter	Set an automatic or specific gate diameter, for use when analyzing your part without a gate(s) or runner system.
Solver parameters	Access the filling phase and packing/curing phase parameters for the 3D flow solver.

Filling Phase Solver Parameters dialog

The **Filling Phase Solver Parameters** dialog is used to specify the settings for the filling phase related solver parameters for a Fill+Pack or Flow analysis of a 3D model.

To access this dialog, choose an analysis sequence which includes Fill, Fill+Pack, or Flow. Click  **Home tab > Molding Process Setup panel > Process Settings**. Click **Next** if necessary until the Fill or Fill+Pack page appears. Click **Advanced Options**, then click **Edit** in the Solver Parameters group. On the **Thermoplastics injection molding solver parameters (3D)** dialog, select the **Fill+Pack Analysis** tab. Click **Solver Parameters**, then click **Filling Parameters**.

NOTE: Default solver parameters are set by experience or heuristically to suit most models and analyses. It is not advisable to change these settings unless modification is justified.

Packing/Curing Phase Solver Parameters dialog

The **Packing/Curing Phase Solver Parameters** dialog is used to specify the settings for the packing phase (thermoplastics materials) or curing phase (thermoset materials) related solver parameters for a Fill+Pack or Flow analysis of a 3D model.

To access this dialog, choose an analysis sequence which includes Fill+Pack or Flow. Click  **Home tab > Molding Process Setup panel > Process Settings**. Click **Next** if necessary until the Fill or Fill+Pack page appears. Click **Advanced Options**, then click **Edit** in the Solver Parameters group. On the **Thermoplastics injection molding solver parameters (3D)** dialog, select the **Fill+Pack Analysis** tab. Click **Solver Parameters**, then click **Packing Parameters**.

NOTE: All solver parameters have a default value that will be suitable for most analyses.

Microchip Options tab 17

The Microchip Encapsulation analysis simulates the encapsulation of semiconductor chips with reactive resins, which provides protection from hostile environments, facilitates heat dissipation, and enables electrical interconnection of the chips.

The **Microchip Options** tab of the **Solver Parameters** dialog is only available for microchip encapsulation molding processes, using a 3D mesh type. Here, you can select the stress analysis type, define the viscosity cut-off temperature, specify the insert temperature and choose in which product to perform the wire sweep/ paddle shift simulation.

Microchip Options tab

This tab is only available for microchip encapsulation molding processes, using a 3D mesh type.

Editing the Microchip Options tab

NOTE: Typically, the solver parameters should remain at their default values.

1 Double-click  **Process Settings** to open the **Process Settings Wizard**.

2 Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.

3 Click **Advanced options.**

4 Click the **Edit** button associated with the **Solver parameters** drop-down list.

5 Select the **Microchip Options** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Microchip Options tab

Use this dialog to specify solver parameters for a Microchip Encapsulation analysis.

Microchip Options tab dialog

The **Microchip Options** tab of the **Reactive Molding solver parameters (3D)** dialog is used to specify values for analysis related inputs for a 3D Microchip Encapsulation analysis.

Stress analysis type	This option is used to select the type of Stress analysis to run to predict paddle shift/wire sweep.
Perform wire sweep/paddle shift simulation in	If the analysis sequence includes wire sweep and/or paddle shift simulation, this option is used to specify which solver will be used to perform this simulation.
Viscosity cut-off temperature	This option is used to define a lower bound for the temperature value when calculating the viscosity of the material. When the predicted temperature of the melt is lower than the cut-off value, then the viscosity is calculated at this viscosity cut-off temperature. **NOTE:** This option is specifically intended for 3D reactive analyses that exhibit convergence problems, typically for materials with a low melt temperature. These convergence problems may be able to be resolved by setting the viscosity cut-off temperature to somewhat higher than the melt temperature.
Effects of wires on flow	When the selected analysis sequence includes Wire Sweep, this option is used to specify whether the analysis will account for the effects of wires on flow. ■ The default setting is **Do not consider**. ■ Change the setting to **Consider** to include the effects of wires on flow and effects of adjacent wires on the drag force. **NOTE:** This option is available only when the wire sweep calculation is performed in Autodesk Moldflow Insight. **NOTE:** The Wire Sweep Detail analysis always includes the effects of wires on flow. Therefore, the **Effects of wires on**

	flow option setting has no effect on results when the analysis sequence includes Wire Sweep Detail.
Critical clearance between wires	When the selected analysis sequence includes Wire Sweep, this option is used to define the distance between pairs of adjacent wires (after wire deformation) at which the wires are considered to be too close together. The clearance distance is measured between the surfaces of adjacent wires. ■ The default value is **0**, which indicates that adjacent wires can touch. ■ Specify a value greater than zero if you want more clearance between adjacent wires. **NOTE:** This option is available only when the wire sweep calculation is performed in Autodesk Moldflow Insight.
Insert temperature	This option specifies whether the part insert and core temperatures are set to a fixed value, or calculated by the Fill+Pack analysis.

To specify values for these options to be used in the current study, double-click **Process Settings** in the Study Tasks pane, or click (**Home tab > Molding Process Setup panel > Process Settings**) to open the **Process Settings Wizard**. Click **Next** to display the **Profile Settings** page; click **Advanced options**; click **Edit** in the **Solver parameters** area; and click the **Microchip Options** tab.

To view or change default settings for these options, click (**Tools tab > Databases panel > Edit Default Properties**); from the **Properties** list, select **Parameters**, and then from the **Description** table, select **Reactive Molding solver parameters defaults (3D)**; click **Edit**; and click the **Microchip Options** tab.

Thermoset Options tab 18

The Thermoset Options tab of the Reactive molding solver parameters (3D) dialog is used to specify a lower bound for the temperature value, when calculating the viscosity of the material.

This option is specifically intended for 3D reactive analyses that exhibit convergence problems, typically for materials with a low melt temperature. These convergence problems may be able to be resolved by setting the viscosity cut-off temperature to somewhat higher than the melt temperature.

Thermoset Options tab

This dialog is only available for reactive molding processes using a 3D mesh type and allows you to set the viscosity cut-off temperature.

Editing the Thermoset Options tab

NOTE: Typically, the solver parameters should remain at their default values.

---

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Thermoset Options** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Thermoset Options tab

Use this dialog to specify values for thermoset related inputs for a 3D Reactive Molding analysis.

Thermoset Options tab dialog

The **Thermoset Options** tab of **Reactive Molding solver parameters (3D)** dialog is used to specify values for thermoset analysis related inputs for a 3D Reactive Molding analysis.

Viscosity cut-off temperature	Defines a lower bound for the temperature value when calculating the viscosity of the material. When the predicted temperature of the melt is lower than the cut-off value, then the viscosity is calculated at this viscosity cut-off temperature. **NOTE:** This option is specifically intended for 3D reactive analyses that exhibit convergence problems, typically for materials with a low melt temperature. These convergence problems may be able to be resolved by setting the viscosity cut-off temperature to somewhat higher than the melt temperature.

Venting Analysis tab 19

Venting analysis is used to predict the effects of air pressure on the flow of polymer to aid in selecting suitable locations where vents may be placed in the mold to allow trapped air to escape the cavity.

The option to perform a venting analysis is available for 3D Thermoplastics Injection Molding, Thermoplastics Overmolding, Reactive Molding and Microchip Encapsulation molding processes when an analysis sequence that includes Fill or Fill+Pack (using the Coupled 3D Flow solver) is selected.

The **Venting Analysis** tab of the solver parameters dialog allows you to set up the details of the analysis, including:

- the vent size
- vent exit pressure
- minimum void volume option, and
- a venting analysis locations grouping option

Venting Analysis tab

The **Venting Analysis** tab of the **Solver Parameters** dialog is available for Thermoplastics Injection Molding, Thermoplastics Overmolding, Reactive Molding and Microchip Encapsulation molding processes, using a 3D mesh type.

Editing the Venting Analysis tab

NOTE: Typically, the solver parameters should remain at their default values.

1. Double-click  **Process Settings** to open the **Process Settings Wizard**.
2. Using the **Next** button if necessary, locate the page of the Process Settings Wizard that has a button called **Advanced options**.
3. Click **Advanced options.**
4. Click the **Edit** button associated with the **Solver parameters** drop-down list.
5. Select the **Venting Analysis** tab and edit the appropriate inputs.

NOTE: You can also edit the material, process conditions and machine parameters for the current study.

Venting Analysis tab

Use this dialog to specify solver parameters for a Venting Analysis.

Venting Analysis tab dialog

The **Venting Analysis** tab dialog is used to specify input values for a venting analysis, when using a 3D mesh type and an analysis sequence that includes Fill or Fill+Pack using the Coupled 3D Flow solver.

Venting analysis option	This option determines if a venting analysis will be performed.
Vent size	Specify the size of the vent associated with a venting analysis location.
Vent exit pressure (Gauge)	Specify the exit pressure of the vent associated with a venting analysis location.
Minimum void volume option	Set the minimum volume of entrapped air that will be considered as a void (air trap).
Venting analysis locations grouping	Specify whether to allow multiple nodes (venting analysis locations) to be defined as a single air vent.