Autodesk® Moldflow® Insight 2012

AMI release notes

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

Release notes 1

These release notes provide you with summaries of new features and enhancements implemented in this product. You can also find information regarding hardware requirements and supported platforms.

Autodesk Moldflow Insight 2012

This release delivers new capabilities to facilitate interoperability among Autodesk products, new solver technologies to expand the range of simulation capabilities, and solver enhancements to improve solution accuracy.

What's new in Autodesk Moldflow Insight 2012

New features and enhancements implemented in the Autodesk Moldflow Insight 2012 release are summarized.

License prerequisites

This release introduces a new method of managing product licenses with the Autodesk Network License Manager. Now, the Autodesk Moldflow Insight user interface and solvers are licensed and installed separately. To complete the Autodesk Moldflow Insight 2012 installation, you must install two applications:

- Autodesk Moldflow Synergy - User Interface
- Autodesk Moldflow Insight - Solvers

Before you run Autodesk Moldflow Insight 2012 for the first time, you must have installed the Autodesk Network License Manager.

ATTENTION: You must have a valid 2012 serial number and product key for both Autodesk Moldflow Synergy - User Interface and Autodesk Moldflow Insight - Solvers applications, and you must register your products and activate your licenses in order to use the Autodesk Moldflow Insight 2012 release.

Refer to the *Autodesk Moldflow Insight 2012 Installation Guide* for detailed instructions.

NOTE: It is no longer possible to use an Autodesk Moldflow Insight license to run Autodesk Moldflow Adviser software.

New Fill Preview technology helps minimize time to solution

The **Fill Preview** is a fast, interactive tool that provides a pre-analysis representation of how a part will fill and where weld lines are likely to form. This new technology helps you make informed decisions about gating locations, without the need of multiple, full analyses, resulting in a dramatic decrease in your time to solution.

The **Fill Preview** option appears in the **Study Tasks** pane when no analysis results are available in the study.

The **Fill Preview** can be enabled even before the part is meshed. It is available for Dual Domain and 3D analyses of Thermoplastics Injection Molding processes.

When the **Fill Preview** option is enabled, when an injection location is set on the part, the predicted filling pattern is displayed almost instantaneously. The filling pattern is overlaid with the positions of weld lines. This information aids in finding the optimal gating location and avoiding potential overpacking and warpage problems.

When you relocate or add an injection location, the filling pattern updates interactively. You can quickly assess the impact of potential gating scenarios on the filling pattern and the resulting weld lines to decide on the optimal gate location, before you have run a full analysis.

NOTE: The **Fill Preview** tool cannot detect short shorts, and it does not provide quantitative information about fill time, pressure, weld line quality, or other results that are available when you run a Fill or Fill+Pack analysis sequence.

Free import of additional CAD formats through Autodesk Moldflow Design Link

Autodesk Moldflow Design Link 2012 now enables direct import of Autodesk Inventor 2012 parts and assemblies, as well as SAT v4–v7 models.

If you are using Autodesk Inventor or Autodesk Inventor Fusion for CAD modeling, you can now take advantage of this expanded direct import capability. Simply install Autodesk Moldflow Design Link 2012 to enable direct import of IPT, IAM and SAT model formats. This new capability is available at no additional cost, and it does not require any specific license.

Leverage Autodesk Inventor Fusion capabilities to modify CAD model geometry

New in this release is the ability to send a CAD model to Autodesk Inventor Fusion to modify the geometry and import the modified CAD model into a new study automatically.

Optimizing a plastic part design for manufacturability and quality typically requires modifying the geometry. This task can be especially challenging if you do not have access to the CAD system in which the part design was originally created. Similarly, parametric (parent-child) relationships in the design could make modifying the design difficult, time-consuming, or occasionally, infeasible.

With this Autodesk Moldflow 2012 release, you now can leverage Autodesk Inventor Fusion to modify geometry without such constraints. Autodesk Inventor Fusion 2012 software is being provided at no additional cost to allow you to modify part geometry, regardless of the CAD system in which it was created or the existence of parametric relationships in the model. For example, you can change part wall thickness or adjust the position or dimensions of features such as ribs or bosses. You can even add new features to the model.

Automatic round-trip workflow when you begin in Autodesk Moldflow Insight

The new **Modify with Autodesk Inventor Fusion** tool automatically exports an imported CAD model and opens the model in Autodesk Inventor Fusion so you can take advantage of its solid modeling capabilities to modify CAD geometry directly. To access the **Modify with Autodesk Inventor Fusion** tool, click  (**Geometry tab > Modify panel > Autodesk Inventor Fusion**).

To use this tool, both Autodesk Moldflow Design Link 2012 software and Autodesk Inventor Fusion 2012 software must be installed.

NOTE: This tool is supported for Dual Domain and 3D analysis technologies.

To use this tool, the current study must contain at least one imported CAD model (part or assembly) in a supported native geometry format. At least one layer containing a CAD Body must be visible in the graphics window. Supported CAD geometry formats include:

- Autodesk Inventor 2012, IPT
- Autodesk Inventor 2012, IAM
- SAT v4–v7
- CATIA® V5R20
- Parasolid® V22
- Pro/ENGINEER® Wildfire® 5.0
- SolidWorks® 2011

NOTE: Assemblies contain more than one CAD body but are treated as a single model when using this tool. If you select a CAD body that is a component of an assembly, the entire assembly is exported to Autodesk Inventor Fusion.

When you finish modifying the geometry, click **Return to Moldflow** (Autodesk Inventor Fusion **Home tab > Moldflow panel > Return to Moldflow**). The Autodesk Inventor Fusion program closes, and the modified CAD model is imported automatically into a new study. For the round-trip workflow to complete successfully, the current Autodesk Moldflow Insight project must remain open while you are modifying the model geometry with Autodesk Inventor Fusion.

In the new study, the following settings specified in the original study are retained:

- Material selection
- Process settings
- Mesh parameters, such as global edge length and local mesh density settings
- Injection locations
- Coolant inlets
- Beam elements (runners, cooling channels) added in Autodesk Moldflow Insight

The following are deleted in the new study:

- The original mesh on the selected CAD model
- Boundary conditions other than injection locations and coolant inlets
- Analysis results

Before you start an analysis in the new study, you must generate a mesh on any unmeshed geometry that is returned from Autodesk Inventor Fusion; ensure that all retained settings, injection locations, and coolant inlets are are correct; and reapply any required boundary conditions on the new mesh.

Export and import SAT format files manually

You also can export and import SAT format files manually to transfer model data.

To export CAD solid geometry from Autodesk Moldflow Insight to a SAT file, the study must contain one or more imported CAD models in a supported native geometry format, as listed previously. Click (**Application menu > Export > Model**), then select **SAT(*.sat)** from the **Save as type** list, and specify a file name and location in which to save the file. If more than one CAD model exists in the study, a numerical suffix will be added to the specified file name, and each CAD model will be saved to a separate file.

NOTE: Assemblies contain more than one CAD body but are treated as a single model. If your study includes an assembly, the entire assembly will be exported to a single file.

You can open the exported file in Autodesk Inventor Fusion to modify the geometry, and save the modified model in SAT format.

To import a SAT file into Autodesk Moldflow Insight, click  (**Home tab > Import panel > Import**), or right-click in the **Project View** pane and select  **Import**. Select the file you want to import, and click **Direct Import using Autodesk Moldflow Design Link** to import the model in its native format.

About Autodesk Inventor Fusion 2012

Autodesk Inventor Fusion technology unites both direct, history-free and parametric, history-based modeling workflows. The unique blend of capabilities enables users to adopt the modeling approach that is most appropriate for the task at hand. Easily express your design ideas irrespective of feature order, dependencies, or original CAD system.

Enhancements to Autodesk Moldflow CAD Doctor 2012 (licensed separately)

Autodesk Moldflow CAD Doctor 2012 (licensed separately) is now available and introduces new features and enhancements which are summarized here.

Autodesk Moldflow CAD Doctor 2012 provides the following new and enhanced geometry simplification functions:

- Detection of *generic holes* in addition to *round holes* is now possible
- The target hole type (*Through hole*, *Blind hole*, or both) can be specified
- The step removal feature can be used to remove counter bores
- Fillet detection and removal has been improved

Other improvements available in Autodesk Moldflow CAD Doctor 2012 include:

- Detection of free edges by loop
- Enhanced **Cross Section View** function
- **Zebra** visualization available in **Shading** mode
- **Layers** function

If you have a license for Autodesk Moldflow CAD Doctor, you can download the 2012 version from the *Autodesk Subscription Center* or select it on the installation media.

Update for 3Dconnexion devices using the ribbon user interface

The 3Dconnexion's 3D mouse devices provide an easy way to rotate the model about different axes of rotation.

This release brings several updates for 3Dconnexion devices for the ribbon UI. Upon installation of the 3Dconnexion device, it is now necessary to select the appropriate Autodesk product for which you plan to use the device, from the drop-down menu on the 3Dconnexion Control Panel.

There are three different navigation modes, selectable from the Navigation Bar, that provide you with different views of your model.

Object mode	Also known as 'free' mode, object mode allows you rotate the object, as if you were holding it in your hand.
Walk mode	Walk mode interactively changes the navigation motion, so you appear to be walking through the model as you move the controller cap. In this mode you move about the scene as if on a moving walkway.
Fly mode	Fly mode interactively changes the navigation motion, so you appear to be flying through the model as you move the controller cap. In this mode you move about the scene as if in an airplane.

The following devices are supported in the ribbon UI, for device release 3.11.3, using drivers 6.11.1 and above:

- Navigator
- Navigator for Notebook
- Space Explorer
- SpacePilot
- SpacePilot Pro

Serial mice are no longer supported in 64-bit operating systems.

NOTE: The operation of these devices in the classic user interface is unaffected by the updates described above, and is the same as in the 2011 release.

Venting analysis option enabled for 3D analyses of thermoplastics molding processes

Previously, the Venting analysis option was available only for 3D analyses of thermoset molding processes. This release introduces the option to perform Venting analysis for analyses of Thermoplastics Injection Molding and Thermoplastics Overmolding processes using 3D analysis technology.

The Venting analysis is used to predict the effects of air pressure in the unfilled cavity on the flow of polymer. This analysis option can help to identify air venting problems and to select locations where air vents may be placed in the mold to allow trapped air to escape.

Results specific to Venting analysis include the **Air traps, including air vents** result and the **Vent region pressure** result.

The option to perform a Venting analysis is available for 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.

NOTE: The option to **Perform venting analysis** is off by default. To perform a venting analysis, ensure that you have set at least one venting location boundary condition on the model, and select the **Perform venting analysis** option on the **Venting Analysis** tab of the analysis solver parameters dialog, depending on the molding process selected.

New Fiber orientation solver options for analyses of short- and long-fiber composite materials

This release introduces the Reduced Strain Closure (RSC) model for fiber-filled composites and the Anisotropic Rotary Diffusion (ARD) model for long-fiber composites as options for calculating fiber orientation in Midplane, Dual Domain, and 3D analyses. These new fiber orientation models improve orientation prediction accuracy for a range of materials and fiber contents. More accurate fiber orientation results can improve predictions of process-induced mechanical properties and subsequent shrinkage and warpage predictions.

The Folgar-Tucker orientation equation is the basis for the default solver options used for fiber orientation calculations in Autodesk Moldflow software.

New RSC model captures slow orientation dynamics in analyses of fiber-filled composites

Recent research indicates that the Folgar-Tucker model over-estimates the change rate of the orientation tensor in concentrated suspensions. The *RSC model* has been developed to capture the slow orientation dynamics and preserve objectivity in calculating fiber orientation. Predictions of the fiber orientation distribution through the thickness of the part are improved.

A United States Patent is held on the RSC model by Delphi Technologies, Inc. (Tucker et al., 2007), and Autodesk holds an exclusive license for use of this model.

New ARD model captures fiber interactions for long-fiber composites

Fibers having an initial length longer than 1 mm are generally considered as long fibers. Usually, the fiber alignment in the flow direction is weaker in long-fiber filled materials than in short-fiber filled materials in injection molding applications. The Folgar-Tucker and RSC models assume isotropic fiber diffusion and cannot predict the fiber interactions accurately for long-fiber filled materials. The *ARD model* assumes anisotropy of fiber diffusion, and this model more accurately predicts fiber orientation when using long-fiber composites.

To support the ARD fiber orientation model option, **Filler length information** has been added to the **Filler Properties** data in the material database.

The RSC and ARD model options have been added to the **Calculate fiber orientation using** model selection list in the **Fiber Orientation Solver Parameters** dialog or on the **Fiber Analysis** tab of the analysis solver parameters dialog, depending on the mesh type and molding process selected.

NOTE: For Midplane and Dual Domain analyses, the RSC and ARD model options are available for analyses using either fiber-filled thermoplastics or thermoset materials. For 3D analyses, the RSC and ARD model options are available only for analyses using thermoplastics materials.

Fiber orientation calculation extended to runners for Midplane and Dual Domain analyses

For Midplane and Dual Domain analyses, the Fiber orientation solver now can account for the fiber orientation in a runner system modeled by beam elements, if one exists in the model. Extending the fiber orientation analysis to include the runner system better represents the actual molding process in the simulation and improves the predicted fiber orientation distribution in the part. In previous releases, the fiber orientation calculation began at the entrance to the part cavity.

NOTE: The extension of the fiber orientation calculation to beam elements was already implemented for 3D analyses.

To take advantage of this new capability for Midplane and Dual Domain analyses, make sure your model includes a runner system modeled by beam elements, set up an analysis sequence that includes **Fill+Pack** and ensure that the option to perform **Fiber orientation analysis if fiber material** is selected, and change the solver parameters **Apply fiber inlet condition at** option to **Injection location**. Calculating the fiber orientation in beam elements is expected to represent more accurately the initial fiber orientation at the cavity entrance. You may notice differences compared to previous versions of Autodesk Moldflow Insight in the predicted fiber orientation in the cavity, and these differences may affect the results of subsequent Warp analyses.

New inlet condition options for Midplane and Dual Domain analyses

New Apply fiber inlet condition at location option	This release adds a new option to specify the location at which the fiber orientation calculation begins for Midplane and Dual Domain analyses using fiber-filled thermoplastics or thermoset materials. The new **Apply fiber inlet condition at** location option allows you to choose whether to apply the selected inlet condition at the **Gate** (the default setting) or at the **Injection location**. Choosing to begin the fiber orientation calculation at the injection location accounts for

the fiber orientation in the runner system, if one exists in the model.

The **Apply fiber inlet condition at** location option has been added to the **Fiber Orientation Solver Parameters** dialog or the **Fiber Analysis** tab of the analysis solver parameters dialog, depending on the mesh type and molding process selected.

New Fiber inlet condition options

New User-supplied inlet orientation option

This release adds a new option to define a specific fiber orientation profile as the inlet boundary condition for Midplane and Dual Domain analyses using fiber-filled thermoplastics or thermoset materials. The new **User-supplied inlet orientation** option allows you to specify values for the in-plane fiber orientation components in the flow direction and cross-flow direction vs. normalized thickness to define the initial fiber orientation profile at the inlet location.

The **User-supplied inlet orientation** option has been added to the **Fiber inlet condition** selection list in the **Fiber Orientation Solver Parameters** dialog or on the **Fiber Analysis** tab of the analysis solver parameters dialog, depending on the mesh type and molding process selected.

New default setting for Fiber inlet condition

The default **Fiber inlet condition** setting has been updated in this release to **Fibers aligned at skin / random at core**.

Now you can choose whether the inlet orientation state is **Fibers aligned at skin / random at core**, **Fibers aligned at skin / transverse at core** (previously the default setting), or defined by a **User-supplied inlet orientation**.

The following table summarizes the availability of the new Fiber orientation solver options for different molding processes and mesh types:

Molding Process	Midplane	Dual Domain	3D
Thermoplastics Injection Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option	RSC and ARD fiber orientation model options Apply inlet condition at location option	RSC and ARD fiber orientation model options

Molding Process	Midplane	Dual Domain	3D
	User-supplied inlet orientation option	User-supplied inlet orientation option	
Thermoplastics Overmolding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	RSC and ARD fiber orientation model options
Gas-Assisted Injection Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option		RSC and ARD fiber orientation model options
Co-injection Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option		
Injection-Compression Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option		
Microcellular Injection Molding	RSC and ARD fiber orientation model options	RSC and ARD fiber orientation model options	

Molding Process	Midplane	Dual Domain	3D
	Apply inlet condition at location option User-supplied inlet orientation option	Apply inlet condition at location option User-supplied inlet orientation option	
Reactive Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	
Microchip Encapsulation	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option	
Multiple-Barrel Reactive Molding	RSC and ARD fiber orientation model options Apply inlet condition at location option User-supplied inlet orientation option		

Transient cool mold temperature analysis options predict mold temperature variations over time

The new transient mold temperature analysis options use the Cool (FEM) analysis sequence to investigate the change in temperature of the mold during an injection cycle. With the inclusion of this feature, you can now investigate the effects of your design on the temperature of the mold during an injection cycle; you can also predict how many cycles would be required from production start-up before the mold reaches a stable temperature

cycle. This feature is available for 3D models only, using the Thermoplastics Injection Molding process.

During an injection molding cycle, the mold temperature will vary around the average temperature. Certain areas of the mold, for instance, areas that are very close to the part, may heat up and cool down significantly during the cycle. The transient mold temperature analysis options enable you to see how the mold temperature varies with time. The Cool (FEM) analysis solver uses the finite element method to calculate results and requires a 3D finite element mesh of both the part and the mold.

There are three different **Mold temperature options** which can be selected for the Cool (FEM) analysis sequence.

Transient within cycle	When an injection molding machine has been working continuously for a period of time, the mold temperature eventually settles down to a consistent, cyclical temperature variation about a cycle-averaged mean temperature. The **Transient within cycle analysis** option can simulate the cyclical nature of the mold temperature within a cycle and enables you to see the deviations from the cycle-averaged mean temperature.
Transient from production start-up	When an injection molding machine is commissioned to manufacture a product, it may require many cycles before the mold reaches its stable operating conditions. The **Transient from production start-up** analysis option will show the evolution of temperature cycles until the mold reaches a stable temperature cycle.
Averaged within cycle	While not a transient analysis, the **Averaged within cycle** analysis option uses the Cool (FEM) solver to determine the average mold temperature during the molding cycle for each location in the mold. This solution is the default selection, and it is equivalent to the existing boundary element method (BEM) Cool analysis sequence.

In a Cool (FEM) analysis sequence, part temperatures and the heat flux from the part into the mold can be determined by either of two finite element method solvers, which are specified in the **Cool (FEM) Solver Parameters**:

- As with the boundary element solution used in the Cool analysis sequence, the **Conduction solver** in the Cool (FEM) analysis considers only the thermal effect of melt contained in the molding cavity and ignores shear heating and fill pattern effects. The conduction solver therefore provides similar information to the boundary element solver used for Cool analyses, but, in most cases, requires less computation time than the boundary element solver. This is the default selection.
- The **Flow analysis on every iteration** option can be used to capture the shear heating and fill pattern effects from flow, but takes longer to run than the conduction solver.

New 3D Mold Mesh tool added to facilitate model preparation

The Cool (FEM) analysis sequence is available only for 3D models and requires that the mold as well as the part be represented by a 3D tetrahedral mesh. You can choose to import mold geometry from a CAD model or STL model, create mold surfaces directly, or use the **Mold Surface Wizard** to generate an initial mold surface. After the mold geometry has been added to the model, you can use the **3D Mold Mesh** tool to generate the required mesh.

NOTE: You must select an analysis sequence that includes Cool (FEM) before you use the **3D Mold Mesh** tool to generate the mold mesh.

Wire Sweep analysis enhancements deliver more detailed information for 3D analyses of Microchip Encapsulation processes

This release introduces two new solver options and five new result plots for Wire Sweep analyses of Microchip Encapsulation processes using 3D analysis technology. These enhancements make it easier to model, analyze, and interpret results of microchip encapsulation cases, especially those that include large numbers of wires.

The new solver options and new result plots are available when an analysis sequence that includes Wire Sweep is selected, and only when the wire sweep calculation is performed in Autodesk Moldflow Insight.

New 3D Wire Sweep solver options

Effects of wires on flow option

When this option is set to **Consider**, the Wire Sweep analysis includes the effects of wires on flow and effects of adjacent wires on the drag force.

Previously, only the Wire Sweep Detail analysis included the effects of wires on flow. However, the Wire Sweep Detail analysis also required special modifications to the 3D mesh, to represent the wire regions as holes in the solid model. This method made it impractical to analyze cases that included large numbers of wires, for example, more than 100.

The **Effects of wires on flow** option allows the Wire Sweep analysis sequence to account empirically for the effects of large numbers of wires on the flow of encapsulant, as well as the effects of adjacent wires on the drag force. Wires are modeled as 1D (beam) elements, and no special modification of the part represented by tetrahedral elements is required.

NOTE: When the **Effects of wires on flow** option is enabled, the solution time may increase significantly with the number of wires included in the model. This option is not enabled by default.

Critical clearance between wires option	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 clearance value is 0, which indicates that adjacent wires can touch. You can specify a value greater than zero if you want more clearance between adjacent wires.

Access both of these options on the **Microchip Options** tab of the **Reactive molding solver parameters (3D)** dialog.

New 3D Wire Sweep result plots make data interpretation easier

Five new result plots are introduced in this release. These plots are available by default upon completion of an analysis sequence that includes Wire Sweep.

TIP: To display results of a 3D Wire Sweep analysis effectively, it is important to enable the results visualization option to **Display 1D elements as segments**. If this option is not enabled, result display could be very slow, especially on models that contain many wires.

Click **Application menu > Options > Results tab**, then in the **Optimize memory for results display** area, click in the **Display 1D elements as segments** checkbox to enable this option.

Wire number result	This result graphically shows the identification number assigned to each wire. Use the animation controls to more easily identify the locations of individual wires in the graphical display.
Maximum wire deflection magnitude - Wire number: XY Plot result	This result shows the maximum deflection value of each wire in the model. Use this information to identify individual wires that undergo the most deflection.
Maximum wire sweep index - Wire number: XY Plot result	This result shows the maximum wire sweep index value of each wire in the model. Use this information to identify individual wires that undergo the greatest drag force during cavity filling.
Wire pairs within critical clearance result	This result shows number of wire pairs for which the predicted clearance after wire deformation is equal to or less than the specified critical clearance value. Use this information to identify pairs of wires that are too close together after wire deformation occurs during cavity filling.

Distance to closest wire result	This result shows, for each wire, the distance to the nearest other wire in the model. Ideally, wire spacing should be uniform. Use this information to identify wires that are too close or too far from adjacent wires.

Solver enhancements address accuracy and speed

Enhancements have been implemented to improve the performance of analysis solvers.

Improved results and solution time for the Gate Location analysis

The Advanced gate locator algorithm has been improved in order to provide a more accurate prediction of the best gate location. In addition to this, the solver automatically takes advantage of available parallel solution technology to speed up the calculation of gate locations.

These improvements result in more accurate Gate Location results within a shorter analysis time:

- When the model geometry is symmetrical and the Advanced gate locator algorithm is specified, the analysis more accurately identifies appropriate, symmetrically placed gate locations.
- Automatic parallelization enables the solver to read the CPU usage at each time step and assign an appropriate number of threads to use at that time step of the analysis. Therefore, the number of threads used in the analysis may vary if the machine load (CPU usage) varies during the analysis.

Improvement of the automatic injection time prediction

Solver improvements have been made in order to obtain a more accurate calculation of the automatic injection time prediction.

Although such improvements don't have a direct impact on any other result than the automatic injection time calculation, Flow results or Molding Window results will benefit from a better prediction of the automatic injection time.

Displacement boundary conditions are available for Dual Domain and 3D Warp analyses

The ability to apply constraints on a part and use them in a Midplane Warp analysis was extended to Dual Domain and 3D Warp analyses.

This enhancement enables consideration displacements a part may experience when being put into a larger assembly. This capability allows for the simulation of the shape and stresses in a warped part, as part of an assembly. For example, constraints can be set on specific nodes, in order to consider assembly-induced deformations and stresses.

Elements can be excluded from Warp analysis

It is now possible to exclude elements from the warpage calculation.

This applies to any part elements, cold runners and inserts, as well as whole parts themselves when in multi-cavity mode, and is available for Midplane and 3D analysis technologies.

This new capability is particularly useful when trying to isolate a cause of warpage in a part, or simply to reduce the analysis time by focusing only on the area of interest.

Increase in the maximum number of elements that can be handled by the 3D Warp analysis solver

The current model mesh size limit for a 3D warpage analysis using the default settings (6-layer option during tetrahedral meshing, and mesh aggregation option for warpage) was extended from 5 to about 15 million tetrahedral elements.

With the continuous increase of computing power and the emergence of cloud computing, simulation models have become larger and more complex.

In order to be able to perform 3D warpage analyses on these models, the default model mesh size limit – i.e. 6-layer option during tetrahedral meshing, and mesh aggregation option for warpage – was extended to reach about 15 million tetrahedral elements. In the meantime, the memory requirement for a 3D warpage analysis of an overmolded part has been reduced by about 20%.

Large Deflection analysis option is available for 3D Warp analysis

The Large Deflection analysis option, already supported by the Midplane Warp analysis, is now available for the 3D Warp analysis as well.

This option provides more accuracy in predicting the deformation of a 3D part, for example with very thin parts where deflection is expected to be high or buckling is likely to occur.

NOTE: For 3D models, the Large Deflection option can require large amounts of CPU time.

Improved volumetric shrinkage calculation at rib junctions for Midplane models

The Midplane flow solver has been improved to more accurately calculate temperature at the base of ribs, where they join the part wall. This enhancement is available in Midplane Fill or Fill+Pack analyses of Thermoplastics Injection Molding, Thermoplastics Overmolding, Gas-Assisted Injection Molding, Co-injection Molding, and Injection-Compression Molding processes.

The solver improvement leads to more accurate predictions of results at rib junctions, including Volumetric shrinkage; Volumetric shrinkage at ejection;

Sink marks, index; Sink marks, depth; and Time to reach ejection temperature results.

Legacy solver option removed from 3D Fill+Pack and Flow analysis solver parameters

The legacy Segregated 3D Flow solver option is no longer available.

The current Coupled 3D Flow solver has been the default solver since its initial implementation; however, the legacy Segregated 3D Flow solver remained as an option in the solver parameters. In this release, the legacy Segregated 3D Flow solver option has been removed.

If studies containing analysis results obtained from the legacy Segregated 3D Flow solver option are imported into the current release, results may still be examined. However, if the analysis is re-run, the current Coupled 3D Flow solver with default solver parameters settings will be used, and a warning message will be displayed to notify you of the change.

To check the solver parameters used in the current study: double-click the **Process Settings** icon in the **Study Tasks** pane; if necessary, click **Next** one or more times until the **Advanced options** button appears; click **Advanced options**; in the **Solver parameters** area, click **Edit**; and examine the settings on the **Fill+Pack tab** or **Flow tab** of the solver parameters dialog (depending on the specified Molding Process).

Enhanced support for GPU technology

Support for GPU (Graphics Processing Unit) technology is extended to 3D Warp analyses in this release. GPU technology allows numerical intensive calculations in a 3D Warp analysis to be performed on the GPU card itself, thus resulting in a faster analysis time.

The software now enables users who have a compatible GPU card installed to benefit from speed improvements when running a 3D Warp analysis.

The **Use GPU card if available** option needs to be selected on **Warp Analysis** tab of the **Thermoplastics injection molding solver parameters (3D)** or **Reactive molding solver parameters (3D)** dialog, depending on the specified molding process. The GPU option for 3D Warp analyses is off by default.

For more information about supported analyses and supported GPU cards, see *GPU card* on page 25.

Automation capabilities

Command-line tools have been added that assist with the integration of third party optimization tools.

Design of Experiments

Understanding how process parameters and design variations impact part production and quality is key when producing high quality tool designs that are easy to run and stable. Design of Experiments is an optimization tool to help engineers understand which design variations produce higher quality outcomes and which processing parameters produce the most stable,

highest quality parts. Design of Experiments is easy to set up, requires no prior knowledge or understanding of statistical methods, and provides information to help you make key design decisions to achieve the required outcome.

Design of Experiments (DOE) enables you to determine the processing window that will make a quality part, within user-defined restrictions. DOE is available for all mesh types, molding processes, and analysis sequences.

Using DOE, you can change any boundary condition, such as coolant flow rate, and see the effect on the quality (for example, shrinkage or warpage) of the part. Depending on the results, you could make changes to the model geometry, such as thickness, to see the effect on quality. DOE allows you to set up multiple objectives or quality measures based on results from any analysis sequence. You can specify quality criteria, such as the amount of deflection at a node or distance between nodes. Once the required results have been obtained, you can easily see what process settings were used to produce the required quality. DOE response surface data can be exported to Microsoft Excel for later processing with a statistical analysis application.

Material database changes

Changes to the material database that have been implemented since the release of Autodesk Moldflow 2011, Service Pack 1, are summarized here.

The changes accommodate new data required to support solver changes implemented in this version, as well as new information received from material suppliers.

Link to material supplier web site added to Thermoplastics material database and Thermoset material database

In the Thermoplastics material database and Thermoset material database, the **Description** data provided for each material now includes a field to store the material supplier's web site address, if provided. This enhancement provides an easy way to connect with a material supplier directly, for example to ask for new or additional material data.

To find the material supplier's web link information for the selected material in a study, right-click on the material name in the **Study Tasks** pane, and select **Details**. Copy the **Link** text and paste it into a web browser to go to the given web site.

Changes to the Thermoplastics material database

Details of these changes are found in the *Material Database Changes* document provided separately.

- Total number of suppliers: 435
  - New suppliers added: 31
    - American Compounding Specialities
    - DuPont Fluoropolymers

- Lavergne Performance Compounds
- Plas-Tech Corporation
- Radici
- Ravago Manufacturing Americas
- SABIC Innovative Plastics, PP Automotive
- SJS Plastiblends Pvt Ltd
- Sun Arrow Kasei Corporation
- The Matrixx Group
- Unigel Plasticos
- Sumika Polymer Compounds UK Ltd
- BARLOG plastics GmbH
- INEOS Olefins & Polymers Europe
- Kao Corporation
- Mountain Valley Recycling
- Nilit Plastics
- POLYPIA
- Styron APAC
- Styron EUR
- Styron NA-LA
- Styron, LLC
- Thai Polypropylene Co Ltd
- Washington Penn Plastic Company
- Witcom Engineering Plastics B V
- DSM Japan Engineering Plastics
- JUNER NEW MATERIALS Co Ltd
- LCY Chemical Corp
- LG-Dow
- Lucite International
- Perrite

- Suppliers deleted: 12
  - Entec Engineered Resins
  - Phillips Sumika
  - SABIC Europe B.V., PP Automotive
  - The Matrix Group
  - Washington Penn Plastics Company
  - BP Solvay Polyethylene
  - Japan Synthetic Rubber Company Ltd
  - Dow Chemical Pacific
  - 3M
  - The Dow Chemical Company
  - Sumitomo-Dow
  - FACT GmbH

- Total number of grades: 8622
  - Grades added: 335
  - Grades deleted: 234
  - Grades amended: 453
- Specific data
  - Grades with specific pvT Properties data: 4958
  - Grades with Long Fiber data: 21
  - Grades with default Shrinkage Properties data: 2097
  - Grades with RSC Fiber shrinkage data: 694

Changes to the Thermoset material database

- Total number of suppliers: 44
  - New suppliers added: 2
    - Lorenz Kunststofftechnik GmbH
    - Raschig
- Total number of grades: 185
  - Grades added: 7
  - Grades deleted: 3

Autodesk Moldflow Communicator compatibility

Please use Autodesk Moldflow Communicator 2012 to view Autodesk Moldflow Results files (*.mfr) exported from Autodesk Moldflow Insight 2012.

Hardware requirements and supported platforms 2

Supported operating systems, hardware platforms and related information are described.

Autodesk Moldflow Insight supported platforms

Autodesk Moldflow Insight 2012 user interface and analysis solvers are supported on Windows platforms. Only analysis solvers are supported on Linux platforms.

Table 1: Supported Windows platforms

Operating System	Service Pack
Windows 7 (32-bit edition) Professional, Enterprise and Ultimate	
Windows 7 (64-bit edition) Professional, Enterprise and Ultimate	
Windows Vista (32-bit edition) Business, Enterprise and Ultimate	SP 2
Windows Vista (64-bit edition) Business, Enterprise and Ultimate	SP 2
Windows XP Professional (32-bit edition)	SP 3
Windows XP Professional (64-bit edition)	SP 2
Windows Server 2003 (64-bit edition)	SP 1
Windows Server 2008 (64-bit edition)	SP 1

Table 2: Supported Linux platforms

Operating System	Bits	Hardware Platform
Red Hat Enterprise Linux 5.6 or later	64	AMD or Intel x86-based 64-bit processor

ATTENTION: Autodesk Moldflow Synergy - User Interface pre- and post-processor is not supported on Linux platforms.

Hardware requirements

- Ethernet card[1]
- DVD-ROM drive[2]
- Microsoft Mouse-compliant pointing device[3]

System specifications for Windows platforms

Table 3: Recommended minimum hardware and software specifications for Autodesk Moldflow Insight on Windows platforms

System component	Recommended
CPU speed	2 GHz or higher
Main memory (RAM)	2 GB or higher
Virtual memory/swap space	2 GB or higher
Disk space	4 GB or higher free disk space (at least 1.5 GB free disk space for installation)
Web browser	Internet Explorer 7
Video viewer	Adobe Flash Player 9 or higher

Third-party software

The help system requires Microsoft Internet Explorer 7.0.

[1] Autodesk Moldflow Insight software uses a network to request and obtain licenses. The network hardware is used to allow license servers to be uniquely identifiable, and these constraints require that all machines which will run Autodesk Moldflow Insight have an enabled Ethernet card.

[2] A DVD-ROM drive is not required if you are installing from an electronic download. You must have an internet connection to download software.

[3] A pointing device is not required for Linux systems.

- Flash Player 9.0 or later is required for 32-bit browsers.
- If you are using a 64-bit browser, you may have to install *http://labs.adobe.com/downloads/flashplayer10.html*.

The reports generated by Autodesk Moldflow products are optimized for use with Internet Explorer 7.0 and Office 97 and later versions (PowerPoint and Word applications only).

System specifications for Linux platforms

Table 4: Recommended hardware specifications for Autodesk Moldflow Insight - Solvers on Linux platforms

System component	Recommended
CPU speed	1 GHz or higher
Main memory (RAM)	2 GB or higher
Virtual memory/swap space	1 GB or higher
Disk space	4 GB or higher free disk space (at least 600 MB free disk space for installation)

NOTE: The more physical memory available, the better Autodesk Moldflow Insight will perform, especially when analyzing larger models. If you plan to use the Cool or Warp solvers on large models, or any of the 3D modules extensively, you should consider installing 2 GB of RAM per processor core.

Disk space requirements

Installation disk space requirements

The table below outlines the recommended disk space allocation for installing the Autodesk Moldflow Insight applications and optional companion products.

Table 5: Installation disk space requirements

Software installed	Installed size on disk (approximate)
Autodesk Moldflow Synergy - User Interface	430 MB
Autodesk Moldflow Insight - Solvers	300 MB
Autodesk Moldflow Design Link	1.85 GB
Autodesk Inventor Fusion	520 MB

NOTE: During the installation process, you will need approximately twice the amount of disk space as specified in the installed size requirement.

Project directory disk space requirements

The disk space requirement for an Autodesk Moldflow Insight study depends on the specified analysis sequence and the size of the model. A 3D Cool or 3D Warp analysis of a large, complex model could require up to 10 GB of disk space in the project directory.

Table 6: Disk space requirements for typical Autodesk Moldflow Insight studies

Analysis sequence	Analysis technology	Disk space required in project directory (approximate)
Fill+Pack (20,000 element model)	Midplane	500 MB
Fill+Pack+Cool	Midplane	700 MB
Fill+Pack+Cool+Warp	Midplane	500 MB
Fill+Pack (20,000 element model)	Dual Domain	500 MB
Fill+Pack (300,000 tetrahedra model)	3D	1 GB
Fill+Pack+Cool	3D	10 GB[4]

Graphics requirements

The graphic requirements listed here reflect the minimum required to obtain acceptable simulations.

- 512 MB DRAM or higher, OpenGL-capable graphics card. Ensure you have installed the latest driver for your card. Please refer to the card manufacturer's website for drivers and instructions.
- 24-bit color setting at 1280×1024 or higher screen resolution.

NOTE: Autodesk does not guarantee the use of any particular graphics card with Autodesk Moldflow products.

Autodesk will make every effort to work with card manufacturers to resolve any problems that may arise in the use of our software with a particular card. However, we cannot guarantee that all graphics-related issues can be fixed, as they may lie within the card manufacturer's drivers rather than Autodesk Moldflow software.

[4] All 3D Cool analyses have high disk space requirements for the temporary directory.

Recommended

Autodesk Moldflow software makes extensive use of OpenGL and texture mapping, therefore any device offering OpenGL acceleration and texture memory will give improved performance.

Using offscreen rendering

1 Click  , then click **Options**..

The **Options** dialog appears.

2 Select the **Viewer** tab.

3 Select the **Use offscreen rendering** option in the **OpenGL** dialog box.

4 Select **OK**.

OpenGL

Specifies whether OpenGL graphics render on-screen or off-screen.

Off-screen rendering uses the OpenGL specification for displaying on a memory buffer before copying to the screen. This allows a significant performance increase compared to on-screen rendering that requires graphics card optimization.

NOTE: Unix computers and some graphics cards on a PC computer do not support off-screen rendering.

Use offscreen rendering

Clear this check box if you are experiencing problems when performing screen captures, creating reports or animations and when printing screen images.

NOTE: Unix computers and some graphics cards on a PC computer do not support off-screen rendering.

GPU card

Autodesk Moldflow Insight software takes advantage of GPUs (Graphics Processing Units) to improve analysis time.

Through GPU technology, you can benefit from the following:

Reduced load on the CPU	GPU cards have their own built-in memory and have the ability to resolve complex simulations. The additional built-in processing power in GPUs relieves task-intensive strain on the CPU, enabling your application to run faster.

Shortened analysis time	The software leverages the processing power offered by CUDA technology in GPU cards. This allows analyses to run at a much greater speed, thus resulting in a faster analysis time. On a very large model, this can reduce the analysis time by a few hours.
Efficient use of data-parallel computing	GPU technology and our parallel solver complement each other. If you use GPU technology in conjunction with a parallel solver, you can achieve even greater speed improvements.

By default, the software automatically detects and uses a compatible GPU card for 3D Fill+Pack analyses, if one is found.

You also can enable the GPU option for 3D Warp analyses. The option is off by default.

You can change GPU option settings on the **Fill+Pack Analysis** tab or **Warp Analysis** tab of the solver parameters dialog, depending on the specified molding process.

Compatible graphics card

You will benefit from speed enhancements offered by GPU technology only if you have a compatible graphics card installed.

To determine if you have a compatible GPU card installed, please check the specifications provided by the card's manufacturer.

The minimum required hardware is a CUDA-enabled card capable of double-precision (64-bit floating point precision) computations. Cards which meet these requirements will have a CUDA Compute Capability of 1.3 or higher.

NVIDIA Fermi GPU card	For optimal performance when using an NVIDIA Fermi GPU card, it is important that the latest drivers are installed.

NOTE: The automatic Windows Update function may not install the most recent version of the drivers.

Supported operating systems

GPU technology is available on all Windows operating systems supported by Autodesk Moldflow products, for both 32-bit and 64-bit systems.

Turning off GPU technology (3D Flow analysis)

You can turn off automatic detection of a GPU card for 3D Flow analyses.

Ensure you have specified a 3D Flow analysis.

1. Click  **Home tab > Molding Process Setup panel > Process Settings**.
   The Fill+Pack Settings dialog appears.
2. Click **Advanced options**.
   The Fill+Pack Analysis Advanced Options dialog appears.
3. From the **Solver Parameters** section, click **Edit**.
4. On the **Fill+Pack Analysis** tab, within the Solver setup section, select **Use coupled solver** from the drop-down list and then click **Solver Parameters**.
   The **Solver Parameters** dialog appears.
5. In the **GPU option** list, click **Do not use GPU**.
   Automatic detection of GPU cards is now turned off.

GPU card limitations

Certain constraints apply when using GPU (Graphics Processing Unit) technology within Autodesk Moldflow software.

Analysis types

Use of GPU technology is limited to certain types of analyses. At present, 3D Fill+Pack, 3D Flow and 3D Warp analyses support GPU technology.

Multiple GPUs

Some graphics cards have multiple GPUs. At present, Autodesk Moldflow software does not support multiple GPUs. If your card has multiple GPUs, the analysis will select the fastest GPU among the available GPUs with adequate memory to perform the analysis.

Maximum model size

The analysis you run is restricted by the size of the available memory on the GPU. For example, if your GPU card memory size is not large enough to run the full model, then the analysis will use the GPU for smaller time steps when the model is partially filled.

The analysis size is determined by the number of elements a model has. The more elements there are, the bigger the model size. Approximately one tetrahedral element equates to 1 KB of memory, so a model with 1 million elements will require 1 GB of device memory on the GPU card to run the model completely on the GPU.

Computer stops responding

When running computation-intensive tasks on a GPU card, the card will be diverted from other tasks such as redrawing the Windows desktop or refreshing other applications. This can be especially noticeable when running multiple jobs simultaneously or when running a very large analysis.

CAUTION: In extreme circumstances, Microsoft Windows may determine that the card has reached a locked up state and trigger a reset of the graphics drivers. If this happens, it is possible that the system will stop

	responding. Details on this behavior are available from Microsoft *here*.
Windows Remote Desktop	Windows Remote Desktop does not support direct access to GPU resources. If an analysis which is set to use GPU resources is launched through a Remote Desktop command line (for example, using the **runstudy** command), it will not be able to utilize the GPU card on the remote machine.
Launching a job in Windows Vista or Windows 7	There is a limitation for Windows Vista and Windows 7 users when using GPU technology. If you launch a job locally on a Windows Vista or Windows 7 machine, or if you remotely launch a job through Job Manager where the remote server is a Windows Vista or Windows 7 machine, we recommend that you do not log out. Logging out will cause the analysis to fail.
GPU card drivers	The latest drivers for any installed card will need to be updated.
	NOTE: The automatic Windows Update function may not install the most recent version of the drivers.

2 GB memory limit on 32-bit Windows systems

Windows 32-bit operating systems have a 2 GB memory limit per process; however there are some guidelines to help you avoid errors due to this memory limit.

- A large number of open studies in a project may exceed the memory limit.
  - Close any open studies that are not being used at a given time.
  - Consider splitting one project with a large number of studies into two or more projects with a smaller number of studies in each.
- Even a small number of large studies open in a project may exceed the memory limit.
  - It is recommended to open only one study per project.
- Post-processing large result files may exceed the memory limit. In particular, 3D results require greater amounts of memory.
  - Consider using the **Optimize memory for results display** options in the results preferences (**Application menu > Options > Results** tab) to minimize the amount of memory required to display results.

- Running a Fill+Pack analysis on a model with a large number of elements may exceed the memory limit.
  - Try remeshing the model using a coarser mesh to reduce the number of elements.

    NOTE: Note that using a coarse mesh may affect analysis accuracy, in particular for 3D models.

  - For a 3D model only, turn off the AMG matrix solver. Change the **AMG matrix solver selection** option setting to **Off** in the **Solver Parameters** dialog. This option is set to **Auto** by default.
- Running a Cool analysis on a model with a large number of elements may exceed the memory limit.
  - Review the mesh density in the part and on cooling circuits, and remesh using a coarser mesh if possible.
- Running a 3D Warp analysis may exceed the memory limit.
  - If the model contains more than 800,000 elements, change the **Use AMG matrix solver** option to **No** on the **Warp Settings** page of the **Process Settings Wizard**. This option is on by default. Turning this option off enables the legacy (SSORCG) matrix solver. The AMG matrix solver is faster but requires more memory compared to the legacy solver.
  - Enable mesh aggregation to reduce memory requirement for analyses of thin-wall parts. Make sure the **Use mesh aggregation and 2nd-order tetrahedral elements** option is selected on the Warp Settings page of the Process Settings Wizard. This option is on by default.

    NOTE: If the part geometry is thick or solid (true 3D geometry), using mesh aggregation is not recommended. Turning off the mesh aggregation option causes the original mesh to be used for the analysis. This increases analysis time and memory requirement but improves accuracy for true 3D parts.

  - The maximum model size on which a 3D Warp analysis can be run on a 32-bit operating system is up to 1.5 million elements (using legacy solver and mesh aggregation options).
- Free resources (memory, swap space) on the machine running the analysis.
- Add more resources (memory, swap space) on the machine running the analysis.

If you have tried the above options but still exceed the 2GB memory limit, consider changing to a computer with a 64-bit Windows operating system.