Autodesk® Moldflow® Insight 2012

AMI Modeling

Autodesk®

Revision 1, 21 March 2012.

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Toolbox 1

The Toolbox provides you with quick access to the most frequently used tools from the **Meshing**, **Modeling**, and **Analysis** menus.

You can obtain help for any of the following tools by selecting it and clicking **Help** in the dialog that appears. If a tool is greyed out, it is unavailable, possibly because it applies to a different mesh type.

 **Create Nodes**	Contains tools for creating nodes in the model.
 **Create Curves**	Contains tools for creating curves in the model.
 **Create Regions**	Contains tools for creating regions and holes in the model.
 **Surface Tools**	Contains tools for diagnosing and fixing surface connectivity problems in the mesh.
 **Move/Copy**	Contains tools for moving and copying mesh elements.
 **Create Beam/Tri/Tetra**	Contains tools for creating beams, triangles, and tetrahedra.
 **Nodal Mesh Tools**	Contains tools for altering nodes in a mesh.
 **Edge Mesh Tools**	Contains tools for altering edges in a mesh.
 **Global Mesh Tools**	Contains tools for altering the mesh.
 **Mesh Diagnostics**	Contains tools for performing mesh diagnostics.
 **Set Constraints**	Contains tools for setting constraints on a model for Stress and Warp analyses.
 **Set Loads**	Contains tools for setting loads on a model for Stress and Warp analyses.

Using filters when modeling or meshing

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Filters help you select a required model or mesh entity by automatically snapping to the nearest instance of the selected model or mesh entity when you click in the model pane.

Filters are available from a drop-down list in the dialogs related to creation (**Geometry tab > Create panel**) and meshing (**Mesh tab > Mesh Repair panel**) tools.

An appropriate subset of the filters is available in each dialog. Select one of the following filters from the drop-down list.

Any item	After selecting this mode, you can click anywhere in the model pane, including locations which are not on the model.
Modeling plane	After selecting this mode, you can click anywhere in the model pane, or on the active modeling plane to create nodes or curves.
Node	Click on a node to select it.
Nearest Node	Click on the model. The nearest existing node is selected for you.
Curve	Click on the model. The nearest existing curve is selected for you.
Center of arc	Click anywhere on an arc. The center coordinate of the circle described by the arc is selected for you.
End of curve	Click anywhere on a curve. The point at the nearest end of the curve is selected for you. This option is useful if, for example, you are using  **Geometry tab > Create panel > Curves > Connect Curves**, and it is important that you select one end of the curve rather than the other end.
Middle of curve	Click anywhere on a curve. The middle point of the curve is selected for you.
Point on curve	Click anywhere on a curve. The closest coordinate on the curve is selected for you.

Modify CAD geometry with Autodesk Inventor Fusion

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You can modify the geometry of an imported CAD solid model with Autodesk Inventor Fusion.

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

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

To use this tool, the current study must contain at least one CAD model (part or assembly) in a supported native geometry format, which has been imported by **Direct Import using Autodesk Moldflow Design Link**. 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.

The **Modify with Autodesk Inventor Fusion** tool automatically exports an imported CAD model and opens the model in Autodesk Inventor Fusion so you can make changes to the model geometry.

When you choose to **Return to Moldflow** or close the Autodesk Inventor Fusion application, the modified CAD model is imported automatically into a new study in Autodesk Moldflow Insight. For the round-trip workflow to complete successfully, do not close the current Autodesk Moldflow Insight project 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

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 correct; and reapply any required boundary conditions on the new mesh.

Modify CAD geometry with Autodesk Inventor Fusion

Use the following procedure to modify an imported CAD model with **Autodesk Inventor Fusion**.

Modify CAD geometry with Autodesk Inventor Fusion

The **Modify with Autodesk Inventor Fusion tool** automatically exports an imported CAD model and opens the model in Autodesk Inventor Fusion.

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

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

The current study must contain at least one CAD model (part or assembly) in a supported native geometry format, which has been imported by **Direct Import using Autodesk Moldflow Design Link**. At least one layer containing a CAD Body must be visible in the graphics pane.

1. Click  (**Geometry tab > Modify panel > Autodesk Inventor Fusion**). The **Modify with Autodesk Inventor Fusion** tool opens.
2. Select the CAD model you want to modify.

   Click on the CAD body in the graphics pane, or type the identifier in the **Select** box. Only one CAD model at a time can be modified with Autodesk Inventor Fusion.

   a. If more than one CAD model exists in the study, you are prompted to select the model you want to edit.

   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.

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3. Click **Apply**.

   The selected CAD model is exported and opened automatically in Autodesk Inventor Fusion.

TIP: Do not close the current Autodesk Moldflow Insight project until you have finished modifying CAD geometry with Autodesk Inventor Fusion.

a 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 in the current Autodesk Moldflow Insight project.

IMPORTANT: 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

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

4 Click **Close** to close the **Modify with Autodesk Inventor Fusion** tool.

Before you start an analysis in the new study, you must:

1 Generate a mesh on any unmeshed geometry that is returned from Autodesk Inventor Fusion.
2 Ensure that all retained settings, injection locations, and coolant inlets are correct.
3 Reapply any required boundary conditions on the new mesh.

Modify CAD geometry with Autodesk Inventor Fusion

Use the **Modify with Inventor Fusion** tool to select an imported CAD model and open it in Autodesk Inventor Fusion.

Modify with Inventor Fusion dialog

To access this dialog, click  (**Geometry tab > Modify panel > Autodesk Inventor Fusion**).

Click on a CAD body in the graphics pane, or type its identifier in the **Select** box to select the CAD model you want to modify. Then click **Apply** to open the model in Autodesk Inventor Fusion.

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.

REMEMBER: 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 in the current Autodesk Moldflow Insight project.

Surface defects 4

If surface defects in the geometry model are not identified and corrected, then they will produce large numbers of defects in the mesh, such as mesh connectivity defects, holes, and overlapping elements. Surface ties are used to identify minor mismatches between surface boundaries.

After importing geometry files such as IGES (*.igs), STEP (*.stp,*.step), Parasolid (*.x_t, *.x_b and so forth), you should perform the following diagnostics on the imported surfaces:

- Surface Boundaries Diagnostic to ensure that all inner and outer surface boundaries form a closed loop.
- Surface Connectivity Diagnostic to ensure that for those surfaces that share a boundary, that these boundaries are fully connected.

Surface repair tools

Surface repair tools investigate surface ties to identify minor mismatches between surface boundaries. If these mismatches are not identified and corrected, they will produce defects in the mesh.

A surface tie defines an association between a curve on the boundary of one surface and a curve on the boundary of another surface. When the part is later meshed, the mesher will use the surface tie definitions to correct for the boundary mismatches and ensure that no free edges are created in these areas.