Autodesk® Moldflow® Insight 2012

AMI The Mesh

Revision 1, 22 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

Mesh entities

1

Mesh entities are known as elements. Different element types are used to model different mesh types.

The three element types are as follows:

- **Beam elements** are 2-noded elements, used for tasks like modeling cold runners and cooling channels.
- **Triangle elements** are 3-noded elements used to model Midplane or Dual Domain mesh types.
- **Tetrahedral elements** are 4-noded elements used to model the 3D mesh type.

Mesh entities

Mesh entities or elements can be assigned different properties according to their function. For example elements can be hot runners or cold runners, etc.

Changing element types

You can change the type of property assigned to each element by assigning a different property to the element. For example, you can change cold runners into hot runners.

1 Select the elements you want to change.

TIP: You can also click **Geometry tab > Selection panel** to select groups of model entities by properties.

---

2 Click **Geometry tab > Properties panel > Assign**.

The **Assign Property** dialog appears.

3 Assign the required property type and specify the property values in one of the following two ways:

- To make a selection from the supplied standard property database, click **Select**, then click on the required property type (for example, Hot runner), highlight a suitable database entry in the displayed list, and click **Select**.
- To specify the properties, click **New**, click on the required property type (for example, Hot runner), enter the required property values, and click **OK**.

Mesh entities

These dialogs are used to assign properties to mesh entities.

To access this dialog, click **Geometry tab > Properties panel > Assign**.

Assign Property dialog

The **Assign Property** dialog allows you to assign all model properties that can be assigned for an analysis. You must specify every aspect of the model, including all part surfaces, for the analysis to run. Only properties relevant to the selected model area will be available.

NOTE: The model section that you want to assign properties to should already be selected.

Commonly Used Mesh Tools

Select which tools are available on the commonly used mesh tools list.

To access this dialog select the **Tools** tab from the **Project** pane. Select any tool from the **Toolbox**, and click ....

Element types used in structural analysis

For Midplane and Dual Domain analysis technologies, structural analyses offer both shell and beam elements.

Shell Elements

Shell elements take the form of planar, straight-edged triangles. When modeling curved shells, they provide a “faceted” approximation to the true geometry. Currently, the thickness of each element is assumed to be constant, although the thickness of adjoining elements may be different.

In all cases, six displacement degrees of freedom are used at each node. These include, three translations parallel to the global axes and three rotations about these axes. For smooth shells it is possible to use only five degrees of freedom, that is three translations and the two local rotations of the mid-surface normal. However, the use of six degrees of freedom in all cases has distinct advantages. Firstly, it allows non-smooth structures such as boxes, trays and stiffened shells to be analyzed without applying any special constraint conditions to the junctions between intersecting surfaces. Secondly, beam elements (which in three-dimensional space must have six degrees of freedom) can be attached directly to the nodes of the underlying plate or shell mesh without applying constraints or transformations. Thirdly, in a large deflection (geometrically non-linear) analysis, the use of three rotational degrees of freedom allows an elegant and geometrically exact model for finite rigid body motion to be introduced [1-3]. It is worth noting that many of the deficiencies of existing geometrical non-linear models of thin-walled structures can be traced to approximations made with respect to the rotational parameters. The parameters adopted here are “natural” in the sense that they are directly related to spectral analysis of the associated rotation tensor. To obtain linearized forms, use is made of the classical analysis of rigid body kinematics by Euler and Rodriques [1,3,4].

The element formulations account for membrane, flexural and transverse shear deformations. The latter is based on the Reissner-Mindlin theory [5,6]. In this theory the classical Kirchhoff assumptions for thin plates and shells are relaxed by lowering the continuity requirements from C1 to C0 , including transverse shear. This allows both “thin” and “moderately thick” plates and shells to be modeled. In practice, the performance of such elements is known to deteriorate rapidly as the plate or shell becomes thin. This phenomenon is called shear locking and is caused by the inability of the element to approach the limiting condition of zero transverse shear strain at the appropriate quadratic rate. Shear locking is alleviated by the use of reduced integration, but contrary to early expectations [7], it is by no means eliminated. More recently, variants of the energy balancing techniques introduced by Fried [8] and McNeal [9] have produced excellent results over a wide range of structural aspect ratios (ratio of a characteristic dimension measured around the mid-surface of the shell structure to the average wall thickness). The form used for the present triangular elements is due to Tessler and Hughes [10], and allows accurate results to be obtained over the approximate aspect ratio range:

8<r<106

Apart from eliminating locking, the method reduces sensitivity of thin shells to element distortion, and improves the conditioning of the stiffness matrix and the quality of stress prediction.

NOTE: The analogous problem of membrane locking only occurs in thin curved elements and therefore need not be considered here.

Currently, only linear elastic materials can be modeled. These maybe either isotropic or orthotropic. In the latter case, the principal planes of the material are orthogonal, with one plane lying in the mid-surface of the shell and the other two planes intersecting this surface along two perpendicular lines referred to as the directions of orthotropy. The directions of orthotropy are determined by processing and are prescribed for each element from the Fill+Pack analysis. The material properties in each direction are specified by the user when preparing the analysis inputs file for the Warp or Stress analysis.

Geometric non-linearity is based on a convected Lagrangian approach in which the displacement field is referred to a set of local convected coordinates that co-rotate with the cross-sections of the shell at each point.

For consistency, the usual inextensibility condition is here applied to fibers that are collinear with the cross-sectional director at any point.

The co-rotational method derives from the polar decomposition theorem of continuum mechanics which asserts that any general spatial motion can always be decomposed into a pure stretch (deformation) followed by a rigid body motion. Adopting suitable finite rotation measures (see above) and explicitly discarding the rigid body component of the overall motion, a consistent method of evaluating the internal deformations and associated stresses within an element is achieved. In practice, this means that no limitations need to be placed of the magnitude of the rigid-body motion, and that the precision with which the internal deformations and stresses are determined will remain constant throughout the analysis.

The formulation is closely related to the finite deflection theories of Reissner [11,12], Simmonds and Danielson [13], Danielson and Hodges [14], and the finite element implementation of these theories by Bates [1] and Simo and Vu-Quoc [15].

Element LMT3

Element LMT3 is used in Midplane analysis technology. LMT3 is a three-noded triangular element with 18 degrees of freedom (six at each node). The element is constructed by superimposing the local membrane formulation due to Bergan and Nygard [16] and Bergan and Felippa [17] with the bending formulation due to Tessler and Hughes [10] and transforming the combined equations to the global coordinate system. The resulting element can model bi-linear variations of membrane and transverse shear strains, but the flexural strains (curvatures) are constant.

The “free formulation” of Bergan and Nygard is based on assumed displacement fields, but goes beyond the strict potential-energy formulation in allowing “nonconforming” shape functions to be used. To ensure convergence, patch test requirements are enforced a-priori. The displacement shape functions are separated into basic and higher-order modes, the former being associated with rigid body and constant strain states and the latter with coordinate invariant in-plane bending modes.

This leads in turn to a basic and higher-order stiffness denoted by Kb and Kh respectively. The combined membrane stiffness then takes the form:

$$K = K_b + \alpha K_h$$

where α is a free parameter that acts as a scaling factor on the higher-order stiffness. In line with detailed test results, $\alpha = 1.5$ has been adopted as the best choice.

The rotation about the local mid-surface normal (often referred to as the drilling freedom) is linked to the average in-plane rotation of the mid-surface by the penalty constraint technique. Thus,

$$\theta = \frac{1}{2}\left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y}\right)$$

The drilling freedom is a fully integrated active component in the formulation. The strength of the link between θ and the in-plane gradients remains even in the case of exactly co-planar elements, so that singularity problems no longer occur. Great attention must be paid to the use of ⊠ as a boundary condition component.

The basis of the bending formulation of the element is an explicit degree of freedom technique achieved via continuous transverse shear edge constraints [10]. This leads to a constrained (coupled) total displacement field with bi-quadratic variation in the lateral displacement uz and bi-linear variations in the normal rotations ⊠ and ⊠y . When combined with the transverse shear correction factor technique [10], the element exhibits a much improved flexural response (compared to the standard constant curvature formulation) across a wide range of aspect ratios.

Element Definition

The geometry, node numbering and local/global degrees of freedom for the element are shown in *Figure 1: The LMT3 Shell Element* on page 5. Note that nodes i, j, k refer to entries in the nodal connectivity table given in the analysis output file. For example, if the connectivity for an element is 11, 101, 85 then i = 11, j = 101, k =85, and the local X-axis runs from node 11 to node 101.

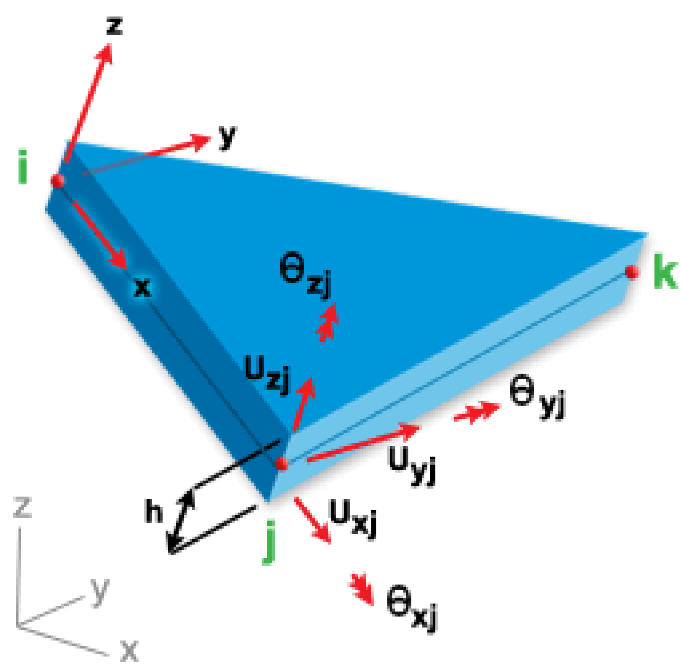


Figure 1: The LMT3 Shell Element

Element Loading

Two types of element loading are available: pressure loading and initial strains due to orthotropic shrinkage.

Pressure loads are assumed to act on and be normal to the element mid-surface. The pressure is assumed to have a constant value over any given element, although pressure values used in adjacent elements may be different.

Integration Rules

Integration over the mid-surface is carried out using three-point numerical quadrature. Since the material is linear elastic and the element is flat, both the strains and the stresses vary linearly through the shell walls. Consequently, explicit pre-integration in the thickness direction is used rather than the more expensive numerical integration.

The integration station locations and weights are shown in *Figure 2: Integration station locations for LMT3 Element* on page 6 and *Table 1: Integration location weights for LMT3 Element* on page 6.

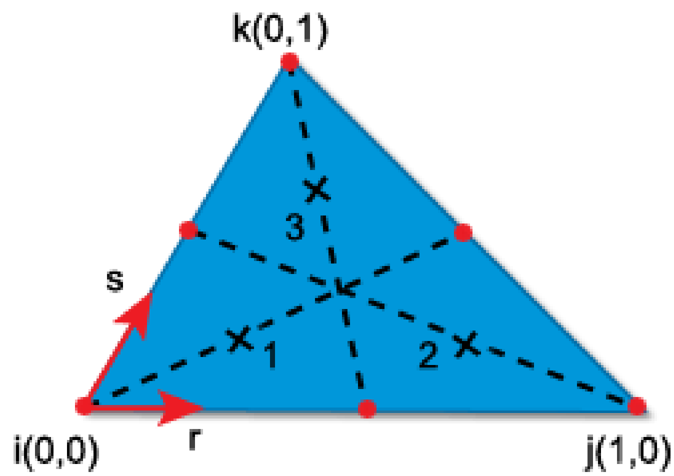


Figure 2: Integration station locations for LMT3 Element

Table 1: Integration location weights for LMT3 Element

Element	Integration Station	r-coordinate	s-coordinate
LMT3	1	1/6	1/6
	2	2/3	1/6
	3	1/6	2/3

Results Output

Membrane forces Nx,Ny,Nxy , bending moments Mx,My,Mxy and transverse shear forces Qx,Qy are calculated at the integration stations and are illustrated in *Figure 3: Results from Analysis for the LMT3 Element* on page 7.

These results are reproduced in the analysis output file.

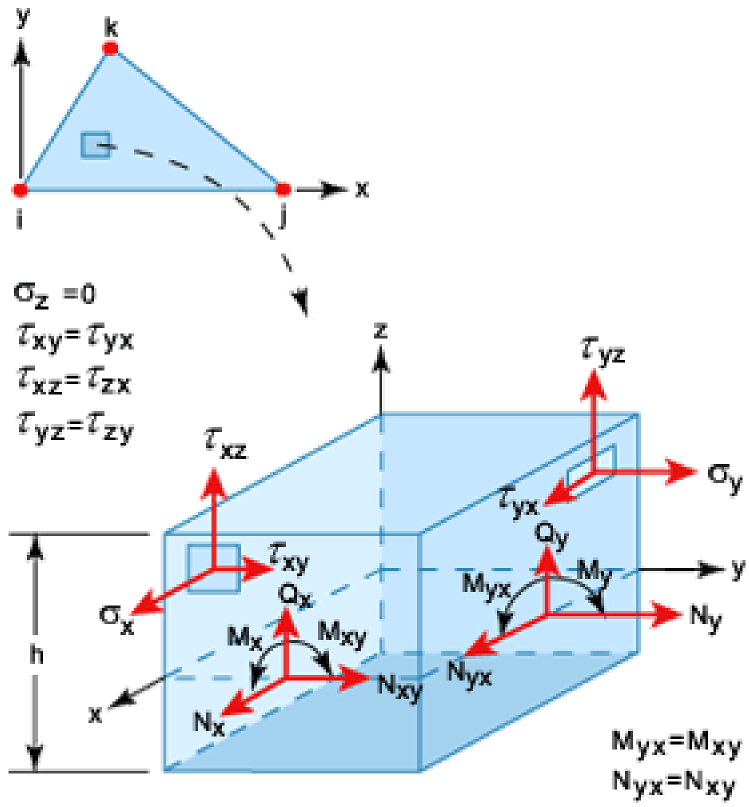


Figure 3: Results from Analysis for the LMT3 Element

Assuming plane stress conditions, the stresses at any level *z* are given by:

$$\sigma_x = \frac{N_x}{h} + \frac{12 z M_x}{h^3} \quad \sigma_y = \frac{N_y}{h} + \frac{12 z M_y}{h^3}$$

$$\tau_{xy} = \frac{N_{xy}}{h} + \frac{12 z M_{xy}}{h^3} \quad \tau_{xz} = \frac{1.5 Q_x}{h}\left(1 - \frac{4z^2}{h^2}\right)$$

$$\tau_{yz} = \frac{1.5 Q_y}{h}\left(1 - \frac{4z^2}{h^2}\right)$$

Note that the forces and moments are those acting on a unit width of the plate or shell and so the membrane and shear forces have dimensions force/length, whereas the moments have dimensions (force).

Element LBT3

Element LBT3 is used in Dual Domain analysis technology. LBT3 is a three-noded triangular element with 18 degrees of freedom (six at each node). The element is constructed by superimposing the local membrane formulation due to Bergan and Nygard and Bergan and Felippa with the bending formulation due to Batoz and transforming the combined equations to the global coordinate system. The resulting element can model bi-linear variations of membrane, flexural strains and transverse shear strains.

The “free formulation” of Bergan and Nygard is based on assumed displacement fields, but goes beyond the strict potential-energy formulation in allowing “nonconforming” shape functions to be used. To ensure convergence, patch test requirements are enforced a-priori. The displacement shape functions are separated into basic and higher-order modes, the former being associated with rigid body and constant strain states and the latter with coordinate invariant in-plane bending modes.

This leads in turn to a basic and higher-order stiffness denoted by Kb and Kh respectively. The combined membrane stiffness then takes the form:

$$K = K_b + \alpha K_h$$

where α is a free parameter that acts as a scaling factor on the higher-order stiffness. In line with detailed test results, $\alpha = 1.5$ has been adopted as the best choice.

The rotation about the mid-surface normal (drilling freedom) takes the same continuum mechanics definition as that used for LMT3, that is,

$$\theta = \frac{1}{2}\left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y}\right)$$

The bending formulation of the element is based on the generalization of discrete Kirchhoff techniques to include the transverse shear effects. The element is free of locking, and valid for the analysis of thick or thin parts. It coincides with the well-known DKT (discrete Kirchhoff triangle) element if the transverse shear effects are not significant.

Numerical tests indicate the LBT3 performs very well. Generally LBT3 is slightly better than LMT3. In addition, both single-layer and multi-layer formulations are available in LBT3, but LMT3 is currently only valid for single-layer analysis.

The multi-layer LBT3 is recommended to be used to conduct Stress and Warp analyses of fiber-filled plastics. With the fiber orientation considered layer by layer, the physical model is more realistic and the result is expected to be more accurate.

Element Definition

The geometry, node numbering and local/global degrees of freedom for the element are shown in *Figure 1: The LMT3 Shell Element* on page 5. Note that nodes i, j, k refer to entries in the nodal connectivity table given in the analysis output file. For example, if the connectivity for an element is 11, 101, 85 then i = 11, j = 101, k =85, and the local X-axis runs from node 11 to node 101.

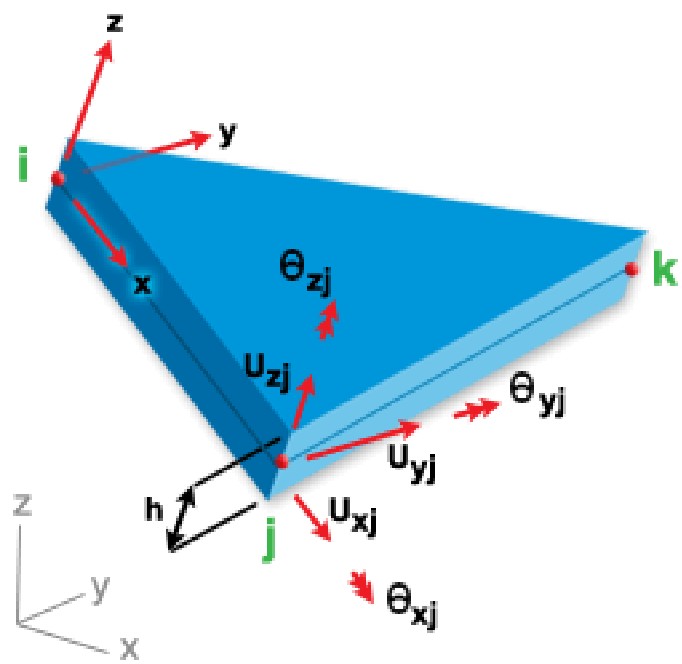


Figure 4: The LBT3 Shell Element

Element Loading

Two types of element loading are available: pressure loading and initial strains due to orthotropic shrinkage.

Pressure loads are assumed to act on and be normal to the element mid-surface. The pressure is assumed to have a constant value over any given element, although pressure values used in adjacent elements may be different.

Integration Rules

Integration over the mid-surface is carried out using three-point numerical quadrature. Since the material is linear elastic and the element is flat, both the strains and the stresses vary linearly through the shell walls. Consequently, explicit pre-integration in the thickness direction is used rather than the more expensive numerical integration.

The integration station locations and weights are shown in *Figure 5: Integration station locations for LBT3 Element* on page 10 and *Table 2: Integration location weights for LBT3 Element* on page 10.

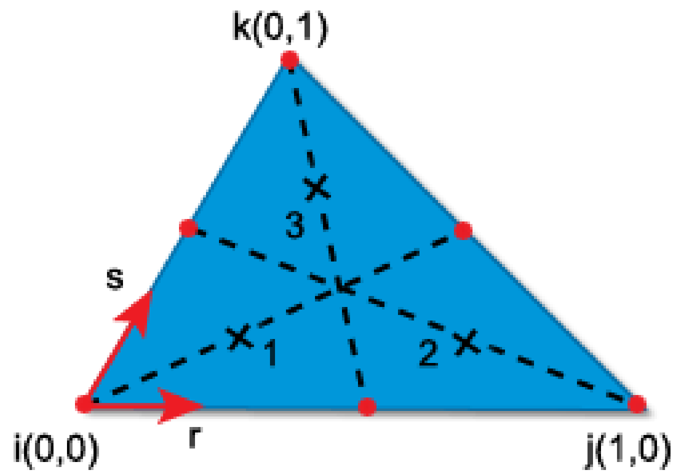


Figure 5: Integration station locations for LBT3 Element

Table 2: Integration location weights for LBT3 Element

Element	Integration Station	r-coordinate	s-coordinate
LBT3	1	1/6	1/6
	2	2/3	1/6
	3	1/6	2/3

Results Output

Membrane forces Nx,Ny,Nxy , bending moments Mx,My,Mxy and transverse shear forces Qx,Qy are calculated at the integration stations and are illustrated in *Figure 6: Results from Analysis for the LBT3 Element* on page 11.

These results are reproduced in the analysis output file.

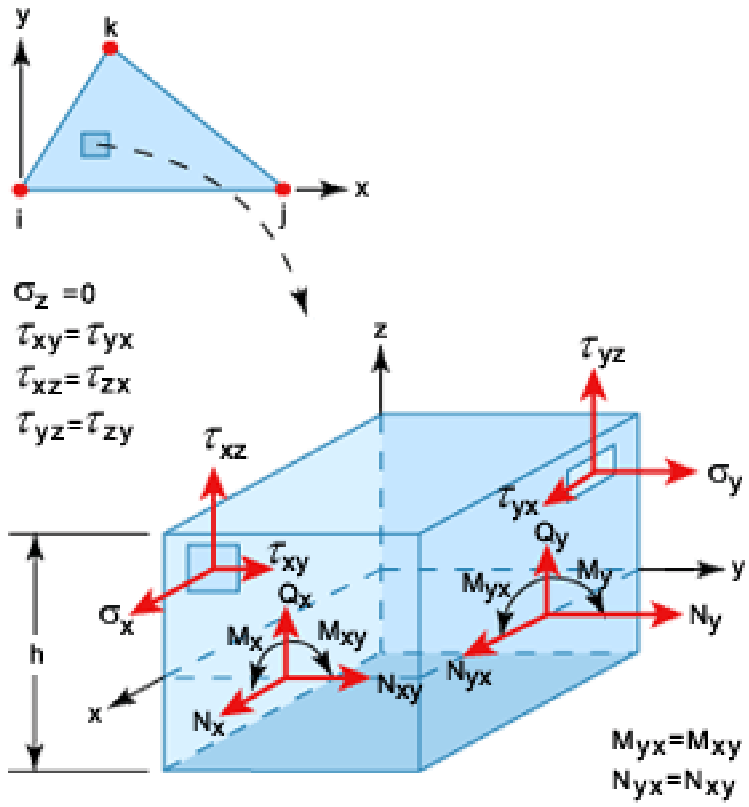


Figure 6: Results from Analysis for the LBT3 Element

Assuming plane stress conditions, the stresses at any level *z* are given by:

$$\sigma_x = \frac{N_x}{h} + \frac{12 z M_x}{h^3} \quad \sigma_y = \frac{N_y}{h} + \frac{12 z M_y}{h^3}$$
$$\tau_{xy} = \frac{N_{xy}}{h} + \frac{12 z M_{xy}}{h^3} \quad \tau_{xz} = \frac{1.5 Q_x}{h}\left(1 - \frac{4z^2}{h^2}\right)$$
$$\tau_{yz} = \frac{1.5 Q_y}{h}\left(1 - \frac{4z^2}{h^2}\right)$$

Note that the forces and moments are those acting on a unit width of the plate or shell and so the membrane and shear forces have dimensions force/length, whereas the moments have dimensions (force).

Beam Elements

The beam element BEAM2, which is a two-noded beam, is currently available in both Midplane and Dual Domain analysis technologies.

The longitudinal axis of the elements is straight, so that when modeling curved beams, they provide a “faceted” approximation to the true geometry. Currently, the beam is assumed to have a circular cross-section of constant radius. However, the cross-sectional radius of adjacent elements may be different.

Six degrees of freedom are used at each node, namely, three translations parallel to the global axes and three rotations about these axes. In the context of finite rotations, the rotational degrees of freedom have exactly the same definition as that discussed for shells. The physical connection of beam and shell elements at one or more nodes is straightforward. Note that if every beam node is attached to a shell node, then the total number of equations that are needed to model the combined structure is the same as the number required to model the shell structure on its own. Thus, the computational overhead involved in adding beam elements is generally quite small.

The element formulations account for axial, bending, torsional and transverse shear deformations. The basic assumptions are that the beam cross-section remains plane and undistorted but, in the presence of transverse shear, it will not remain normal to the longitudinal axis. The resulting model can be thought of as deriving from a classical Euler-Bernoulli beam in two stages. Firstly, the average effects of transverse shear are accounted for by adding a Reissner-Mindlin type shear model. This leads to what is generally referred as a Timoshenko beam. In the second stage, the torsional behavior is modeled using the theory due to St.Venant.

Where a runner, rib or stand-alone beam structure is modeled with beam elements, then the aspect ratio (ratio of total length measured along the axis of the beam to the average cross-sectional diameter) should lie in the approximate range:

$5<r<106$

The very high upper limit is made possible by using the penalty relaxation technique. Although the beam elements do not suffer from shear or membrane locking, this technique prevents the element stiffness matrix from becoming ill-conditioned at high aspect ratios.

Currently, only isotropic linear elastic materials can be modeled for beam elements. Note that it is not possible to account for general orthotropic material behavior without conflicting with the basic assumptions mentioned above.

Geometric non-linearity is based on a convected Lagrangian approach in which the displacement field is referred to a set of local convected coordinates that co-rotate with the cross-sections of the shell at each point. For consistency, the usual inextensibility condition is here applied to fibers that are collinear with the cross-sectional director at any point.

The co-rotational method derives from the polar decomposition theorem of continuum mechanics which asserts that any general spatial motion can always be decomposed into a pure stretch (deformation) followed by a rigid body motion. Adopting suitable finite rotation measures and explicitly discarding the rigid body component of the overall motion, a consistent method of evaluating the internal deformations and associated stresses within an element is achieved. This means that no limitations need to be placed of the magnitude of the rigid-body motion and that the precision with which the internal deformations and stresses are determined will remain constant throughout the analysis.

The formulation is closely related to the finite deflection theories of Reissner [11], Danielson and Hodges [14], Hodges and Simo. The finite-element implementation of these theories by Bates [1] and Simo and Vu-Quoc [15] is also relevant.

Element BEAM2

This is a 2-noded beam with 6 degrees of freedom at each node. The beam axis is assumed to be straight and the cross-section to be circular and solid. The element can be used in stand-alone form or, alternatively, it may be connected to one edge of the triangular shell elements LMT3.

The formulation is based on linear isoparametric shape functions. To avoid shear locking, reduced integration is used to find the stiffness and internal forces, which in this case means using 1-point Gauss quadrature. The resulting model exhibits constant axial, bending, torsional and shear strain fields.

In its unmodified form, the predicted element response depends on the bending mode to which the element is subjected.

For pure bending, exact nodal displacements and stress resultants are obtained at the control integration point.

However, the over-stiffness in the latter case are completely eliminated by applying the penalty relaxation method [19]. Exact displacements and central stress resultants are also predicted for constant stretching and twisting.

Element Definition:

The geometry, node numbering and local/global degrees of freedom for the element are shown in *Figure 7: The BEAM2 Beam Element* on page 14. Note that nodes i, j refer to entries in the nodal connectivity table that is given in the analysis output file. For example, if the connectivity for an element is 55, 77 then i = 55, j = 77, and the local X-axis runs from node 55 to node 77.

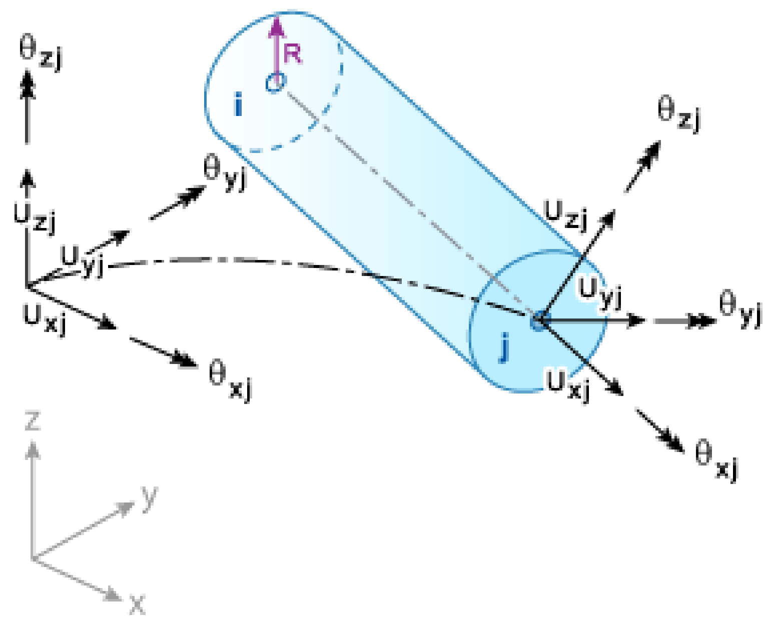


Figure 7: The BEAM2 Beam Element

Element Loading: The only type of element loading currently available is the specification of axial and curvature strains due to shrinkage. (Note, however, that concentrated external loads, that is forces and moments, can be applied directly to the nodes of the finite element mesh.)

Integration Rules

Internal forces (stress resultants) are calculated at the central integration point shown in *Figure 8: Integration Station for BEAM2 Element* on page 14 and reproduced in the analysis output file.

For a linear elastic material, both the strains and stresses vary linearly over the cross-section of the beam. Consequently, explicit pre-integration over the cross-section is used instead of the more expensive numerical integration.

(a) integration point.

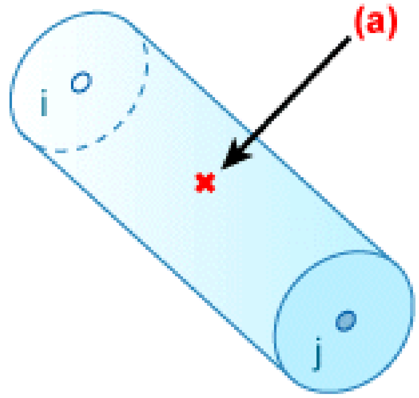


Figure 8: Integration Station for BEAM2 Element

Results Output

The following stress resultants are calculated at the integration station and reproduced in the analysis output file.

Moments:

$M_y=\int_A \sigma_x z\,dA$ $M_z=-\int_A \sigma_x y\,dA$

Torsion:

$T_x=\int_A (\tau_{xz} y-\tau_{xy} z)\,dA$

Forces:

$Q_y=\int \tau_{xy}\,dA$ $Q_z=\int \tau_{xz}\,dA$ $P_x=\int \sigma_x\,dA$

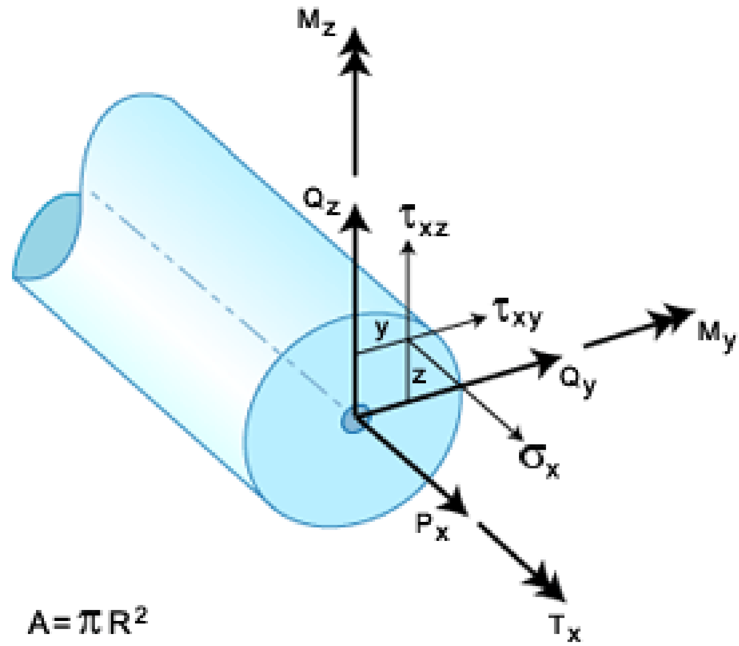


Figure 9: Results From Analysis for the BEAM2 Element

The stresses at any point (y,z) are then given by:

⊠=1AP⊠⊠+4⊠⊠My−y⊠MzR2
⊠⊠y=1A⊠43⊠Qy⊠−y2R2−2⊠⊠TXR2
⊠⊠⊠=1A⊠43⊠Qx⊠−x2R2−2y⊠⊠TXR2

References

1 Argyris, J.H., “An excursion into large rotations”, Comp. Meth. Appl. Mech. Engrg., Vol. 32, 1982, pp. 85-155.

2 Rankin, C.C. and Brogan, F.A., “An element independent corotational procedure for the treatment of large rotations”, in: Collapse Analysis of Structures (L.H. Sobel and K. Thomas eds.), ASME, New York, 1984, pp. 85-100.

3 Bates, D.N., The mechanics of thin walled structures with special reference to finite rotations, Ph.D. Thesis, University of London, 1987.

4 Hodges, D.H., “Finite rotation and non-linear beam kinematics”, Vertica, Vol. 11, No. 1/2, 1987, pp. 297-307.

5 Reissner, E., “The effect of transverse shear deformation on the bending of elastic plates”, J. Appl. Mech., ASME, Vol. 12, 1945, A69-A72.

6 Mindlin, R.D., “Influence of rotatory inertia and shear on flexural motions of isotropic, elastic plates”, J. Appl. Mech., ASME, Vol. 18, 1951, pp. 31-38.

7 Zienkiewicz, O.C., Taylor, R.L., and Too, J.M., “Reduced integration technique in general analysis of plates and shells”, Int. J. Num. Meth. Engrg., Vol. 3, 1971, pp. 275-290.

8 Fried, I., “Residual energy balancing technique in the generation of plate bending finite elements”, Comp. and Struct., Vol. 4, 1974, pp. 771-778.
9 McNeal, R.H., “A simple quadrilateral shell element”, Comp. and Struct., Vol. 8, 1978, pp. 175-183.
10 Tessler, A. and Hughes, T.J.R., “A three-node Mindlin plate element with improved transverse shear”, Comp. Meth. Appl. Mech. Engrg., Vol. 50, 1985, pp. 71-101.
11 Reissner, E., “On one-dimensional large-displacement finite-strain beam theory”, Stud. Appl. Math., Vol. 52, 1973, pp. 87-95.
12 Reissner, E., “Linear and non-linear theories of shells, in: Thin Shell Structures” (Y.C. Fung and E.E. Sechler, eds.), Prentice-Hall, Englewood Cliffs, New Jersey, 1974, pp. 29-44.
13 Simmonds, J.G. and Danielson, D.A., “Non-linear shell theory with finite rotation and stress-function vectors”, J. Appl. Mech., ASME, Vol. 39, 1972, pp. 1085-1090.
14 Danielson, D.A. and Hodges, D.H., “Non-linear beam kinematics by decomposition of the rotation tensor”, J. Appl. Mech.,ASME, Vol. 54, 1987, pp. 258-262.
15 Simo, J.C. and Vu-Quoc, L., “A three-dimensional finite strain rod model. Part II: Computational aspects”, Comp. Meth.Appl. Mech. Engrg., Vol. 58, 1986, pp. 79-116.
16 Bergan, P.G. and Nygård, M.K., “Finite elements with increased freedom in choosing shape functions”, Int. J. Num. Meth. Engrg., Vol. 20, 1984, pp. 643-664.
17 Bergan, P.G. and Felippa, C.A., “A triangular membrane element with rotational degrees of freedom”, Comp. Meth. Appl. Mech. Engrg., Vol. 50, pp. 25-60.
18 Nygård, M.K., The free formulation for non-linear finite elements with applications to shells, Report No. 86-2, Division of Structural Mechanics, The Norwegian Institute of Technology, Trondheim, 1986.

Beam elements

Beam elements are two-noded elements that are used to model the mesh associated with various model features.

A variety of model features are represented by beam elements, including runner system components, cooling channels, gas channels, and tapered or non-tapered structural beams on the part.

Beam elements

Beam elements can be created and modified as required.

Creating beam elements using the Create Beams tool

You can specify the number of beam elements to be created between two selected nodes using the **Create Beams** tool.

1 Click **Mesh tab > Mesh panel > Create Beams**.

2 In the **Create Beams** dialog, enter the coordinate of the first node to represent one end of the beam, or click on the model to select the node.

3 Enter the coordinate of the second node to represent the other end of the beam, or click on the model to select the node.

4 Enter the number of beams to be created along the length between the selected nodes.

   If you enter a number greater than 1, the beam will be divided into segments of equal length to match the number you enter.

5 To change the properties of the newly created beam element(s), click **Change**, and select the appropriate property from the **Assign Property** table.

Creating beam elements using the Generate Mesh tool

Beam elements can be used to represent a variety of model features, including runner system components, cooling channels, gas channels, and structural beams (tapered or non-tapered) on the part.

You can automatically specify the length of beam elements to be created on a curve using the **Generate Mesh** tool.

1 Create a curve **Geometry tab > Create panel > Curves**.

2 Select the curve.

3 Click **Geometry tab > Properties panel > Assign**.

   The **Assign Property** dialog appears.

4 Click the **New** button, and select a property type from the drop-down list.

   The relevant dialog appears for you to set addition properties.

5 Click **OK** to close all dialogs.

6 Click **Mesh tab > Mesh panel > Generate Mesh**.

   The **Generate Mesh** dialog appears.

   NOTE: Notice the **Global edge length** is already defined. You can change this, and **Preview** its effect.

7 Click **Mesh Now**.

   The **Global edge length** value automatically determines the number of beam elements that can be created on the length of the curve.

Deleting duplicate beam elements automatically

Duplicate beam elements can cause analysis problems and convergence errors in Cool analyses (for example, "** WARNING ** Solution iteration limit reached before convergence"). Duplicate beams can be present in cooling channels, runners or connector elements.

1 Click **Mesh tab > Mesh Diagnostics panel > Mesh Statistics** to check if the mesh contains duplicate beams.

   The Intersection Details section of the **Mesh Statistics** dialog reports the number of duplicate beams.

2 If the number of duplicate beams is greater than 0, click **Mesh tab > Mesh Repair panel > Auto Repair** to repair the mesh automatically.

3 Click **Mesh tab > Mesh Diagnostics panel > Mesh Statistics** again to check if the mesh contains duplicate beams.

   If duplicate beams are still present in the mesh, you need to correct the problem manually.

Deleting duplicate beam elements manually

Duplicate beam elements can cause analysis problems and convergence errors in Cool analyses (for example, "** WARNING ** Solution iteration limit reached before convergence"). Duplicate beams can be present in cooling channels, runners or connector elements.

In most cases, you can use the **Auto Repair** mesh tool to remove duplicate beams automatically. If manual repair is necessary, follow these steps to identify and delete duplicate beams.

NOTE: The following procedure requires you to make significant changes to the study. Autodesk recommends that you work on a copy of the study.

---

1 Click **Mesh tab > Mesh Diagnostics panel > Mesh Statistics**.

   The Intersection Details section of the **Mesh Statistics** dialog reports the number of duplicate beams.

2 Show a layer containing the beam elements and hide all other layers.

   If you have many cooling channels and a complicated runner system, you can separate them into different layers.

   TIP: Use **Mesh tab > Selection panel > Properties** to make a selection by properties.

   ---

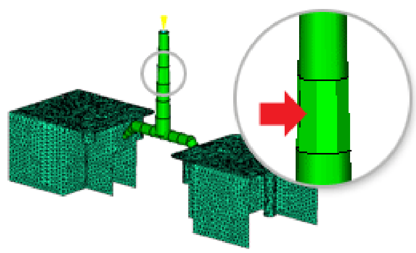
3 Delete the visible layer containing beam elements.

4 Check the **Mesh Statistics** again:

- If the number of duplicate beams does not decrease, the beams in the deleted layer are not causing the problem. Click **Undo** to recover the deleted layer, then hide it. Repeat steps **2** to **4** with another layer.
- If the number of duplicate beams decreases or equals 0, you have found the problem area. Click **Undo** to recover the deleted layer and continue with step **5**.

5 Zoom in on the problem area to visually identify duplicate beams.

Duplicate beams are displayed with a choppy surface compared to non-duplicate beams as shown in the image below.



6 Click **Geometry tab > Utilities panel > Query**.

The **Query Entities** dialog appears.

7 Band select a single element.

8 Click **Show** on the dialog and check the number of entities displayed.

9 If you see one element selected but the dialog displays two, you have found a duplicate beam.

It is likely that all duplicate beams are along the same section of cooling circuit or runner.

10 Click once on the duplicate beam to select one element.

Do not use band selection as it will select both the original and duplicate elements.

NOTE: The highlighted selection is hidden by the duplicate beam and you will not be able to see it.

---

11 Click **Delete**.

If the delete icon is not enabled, try to select the beam again.

12 After the duplicate beam is deleted, the beam element should appear smooth. If it does not, repeat steps **10** and **11** as there may be further duplicate beams in the same location.

13 Check the **Mesh Statistics**.

The number of duplicate beams should decrease.

14 Repeat steps **6** to **12** until the **Mesh Statistics** shows 0 duplicate beams.

Remeshing beams

There are several reasons to remesh beam geometry. For example, you should remesh if you are not satisfied with the density of the mesh in the runner system, cooling channel(s), gas channel(s), structural beam(s), or if you have modified portions of the beam geometry. You can only specify one edge length when meshing, and not different mesh densities for specific areas.

NOTE: Before proceeding, it is important to remember that the mesher will only mesh what is visible on screen, and it is best to have curves defining runners and cooling channels rather than just elements.

---

1. Using the **Layers** pane, display only the beam elements and curves of the entities to be remeshed. Make sure the part geometry and mesh are not visible.
2. Click **Mesh tab > Mesh panel > Generate Mesh**, and select the option to **Remesh already meshed parts of the model**.
3. On the General tab, enter the required **Global edge length**.
4. Click **Preview** to preview the edge length.
5. Click **Mesh Now**.

   The beam geometry will be remeshed using the new edge length that you specified.

TIP: For accurate results, a beam should be meshed into at least three parts, each part no more than 2.5 times the beam's diameter.

Beam elements

These tools are used to model features, such as runner system components, or to repair mesh defects.

Create Beams tool

The **Create Beams** tool allows you to create beam (1-dimensional) mesh elements. Typically, beams represent runners and cooling circuits on a model.

To use this tool, click (**Mesh tab > Mesh panel > Create Beams**).

NOTE: Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Part Beam dialog—Mold Properties

The **Mold Properties** tab of this dialog is used to specify the properties of the mold block in contact with the selected beam elements or curves of type **Part beam**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Mold material	Specifies the mold material in contact with the selected model feature.

Part Beam dialog—Overmolding Component

The **Overmolding Component** tab of this dialog is used to specify which stage/component in the overmolding process the selected elements or curves of type **Part beam** relate to.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Component	Specifies which stage of a 2-Shot Sequential Overmolding process the model feature relates to.

Part Beam dialog—Beam Properties

The **Beam Properties** tab of this dialog is used to specify the physical properties of the selected beam elements or curves of type **Part beam**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Cross section is	Select the type of cross-sectional shape you want to apply to the selected element(s).
Shape is	Specifies whether the curve or beam element has constant dimensions [non-tapered] along its length, or is tapered.
Occurrence number	Occurrence numbers are used to specify the number of times that a given flow path is repeated in a model.

Exclude from clamp force calculation	Select this option if you want to exclude the selected part surface elements from the clamp force calculation.

NOTE: We recommend that thick ribs with a width-thickness ratio of less than 4:1 be modeled as beam elements so that the solver can better account for the effect of heat transfer through the edges of the rib.

Cross-sectional shape equivalents for beam elements

To simplify calculations, beam elements with non-round cross-sectional shapes can be converted to equivalent circular beams.

Heat transfer of the original shape also needs to be considered. A shape factor, which is the ratio of the outer circumference of the actual cross-sectional shape to the circumference of the circular beam with equivalent cross-sectional area, makes adjustments for the additional surface area the converted beam should have.

Supported cross-sectional shapes for beam elements

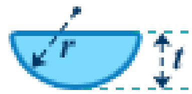
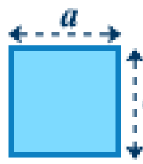
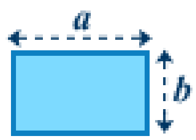
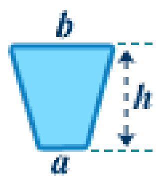
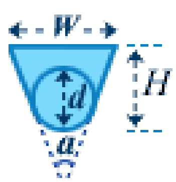
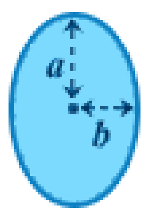
The following table lists the available cross-sectional shapes and the required dimensions to define them.

Shape	Required dimensions
Circular	Diameter (2r)
Annular	Outer diameter (d_o), inner diameter (d_i)
Semicircular	Diameter (2r), height (t)
Rectangular	Width (a), height (b)
Trapezoidal	Top width (b), bottom width (a), height (h)
Rounded trapezoidal	Width (W), height (H), diameter (d)
Ellipse	Long radius (a), short radius (b)
Other	Equivalent diameter, shape factor

Equivalent diameter and shape factor for non-round beam cross-sectional shapes

The following table illustrates the required dimensions to define non-round beam cross-sectional shapes and presents the calculations that define equivalent diameter and shape factor for each beam shape.

NOTE: Circular and annular beam shapes are solved explicitly by the analysis. It is not necessary to calculate an equivalent diameter or shape factor for circular or annular beam shapes.

Shape	Cross-sectional geometry	Equivalent diameter, based on equivalent cross-sectional area	Shape factor
Semicircular		[illegible] x=r−tr	[illegible] x=r−tr
Rectangular		2[illegible]π	2π
Rectangular		2[illegible]π	a+bπ[illegible]
Trapezoidal		2a+b[illegible]π	a+b+2[illegible]a22+h2[illegible]
Rounded trapezoidal		2Hπ[illegible]W+H[illegible]n−1[illegible]W	[illegible]
Rounded trapezoidal		4[illegible]Aπ where A[illegible]	cπ[illegible] where [illegible]
Elliptical		4[illegible]	0.5[illegible]2+b2a[illegible]

ATTENTION: In the formulas presented above, angle measurements must be given in *radians* (not degrees).

Worked example

A rectangular shaped runner has a cross-sectional width (a) of 5 mm and height (b) of 3 mm.

Using the formula above, this runner could be modeled by a circular beam element with a diameter of 2abπ or 4.37 mm.

The runner would need a shape factor of a+bπab or 1.16.

When modeling this beam element, select **Geometry tab > Properties panel > Edit**. In the **Cross-section is** box, select **Other shape**. Click **Edit dimensions** and enter the calculated values for Equivalent diameter (4.37) and Shape factor (1.16) in the **Cross-Sectional Dimensions** dialog that appears.

Triangular elements

Triangular elements are three-noded elements used to model Dual Domain and Midplane meshes.

Triangular elements

The **Create Triangles** command enables you to create a two-dimensional triangle mesh element from specified nodes.

Creating Triangular elements

1. Click **Mesh tab > Mesh panel > Create Triangles**.
2. Enter the **Node 1** or select it on the model.
3. Enter the **Node 2** or select it on the model.
4. Enter the **Node 3** or select it on the model.
5. **Optional:** Select **Inherit properties from neighbors** if the triangle is to be created with the same properties as the neighboring elements.
6. Click **Apply**.

   The Triangle will be created using the three specified nodes.

Triangular elements

The **Create Triangles** tool is used to repair mesh defects.

To use this tool, click (**Mesh tab > Mesh panel > Create Triangles**).

Create Triangles tool

The **Create Triangles** tool allows you to create triangle (2-dimensional) mesh elements.

To use this tool, click  (**Mesh tab > Mesh panel > Create Triangles**).

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Cross-sectional shape equivalents for surface features

Not all plastic parts have regular cross-sections, that is, are flat on both sides. Analyzing a complex shape such as a grill, requires you to accurately model the exact shape of a part with an extremely fine mesh. This is a time consuming task that requires a high computational overhead.

For Midplane analysis technology, such complex cross-sections can be approximated by representing the area with a thinner, flat cross-section that has an equivalent material volume.

A Midplane mesh does not take into account the edges of the part. A shape factor accounts for the additional surface area in contact with the mold. The shape factor for a surface pattern is defined as the ratio of the actual contact surface to that of the equivalent flat surface.

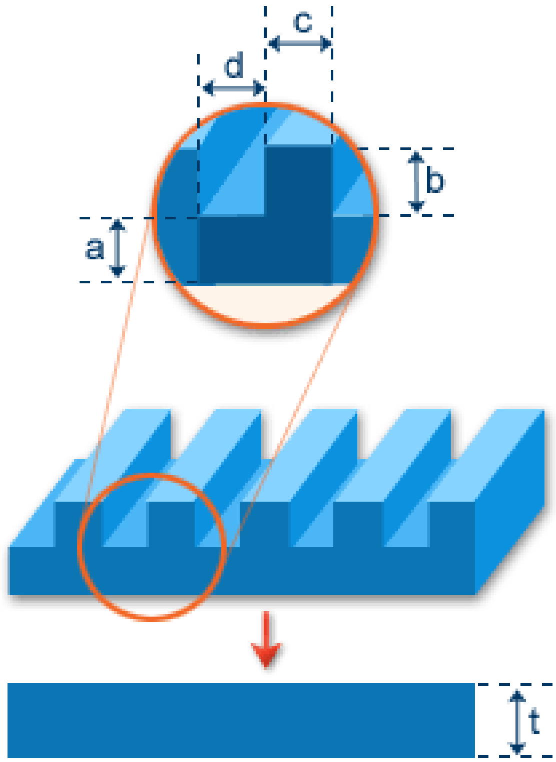
NOTE: Features in a 3D mesh cannot be modeled with an equivalent thickness and must be modeled in full with appropriate mesh densities.

Supported cross-sectional shapes for triangular elements

Thickness and shape factor are attributes for Midplane triangular elements, similar to diameter and shape factor used for beam elements. When a surface has a regular cross-section, the thickness is specified and the shape factor is equal to 1. However, surface patterns are frequently used in plastic part designs for appearance and functional reasons. There are two ways to model parts with surface patterns: use elements that are smaller than the feature in the surface patterns, or use elements with an equivalent thickness and a shape factor to correct the additional cooling and flow resistance from the surface patterns. The first method could result in a very large number of elements for models with small features in the pattern. The second method is a more efficient way to solve practical problems.

The following diagram shows a slab-like surface. The equivalent thickness and shape factor are calculated from one basic unit of the pattern, which is repeated.

The basic unit of the pattern has a projected area, A; a contact surface, S, which includes the top, bottom, and lateral areas; and the volume, V, between the top and bottom surfaces. For the same equivalent thickness, a larger shape factor means a given volume has more contact surface and has more cooling effects from the wall as well as more flow resistance from the geometric irregularity. The program automatically adjusts these two effects for Midplane elements based on their respective shape factors.



t=VA=a⊠+d+b⊠c+d S2⊠A=b+c+dc+d

There are two ways to model geometries with grills, depending on the relative dimensions of the grill and the mesh size. If the length of a grill segment is larger than the mesh size, as shown in part (a) of the following diagram, the grills should be modeled by beam elements with the proper equivalent diameter and shape factor. If the grill size is small in relation to the mesh size, the total grid area should be modeled as triangular elements with an equivalent thickness and shape factor, as shown in part (b) of the following diagram.

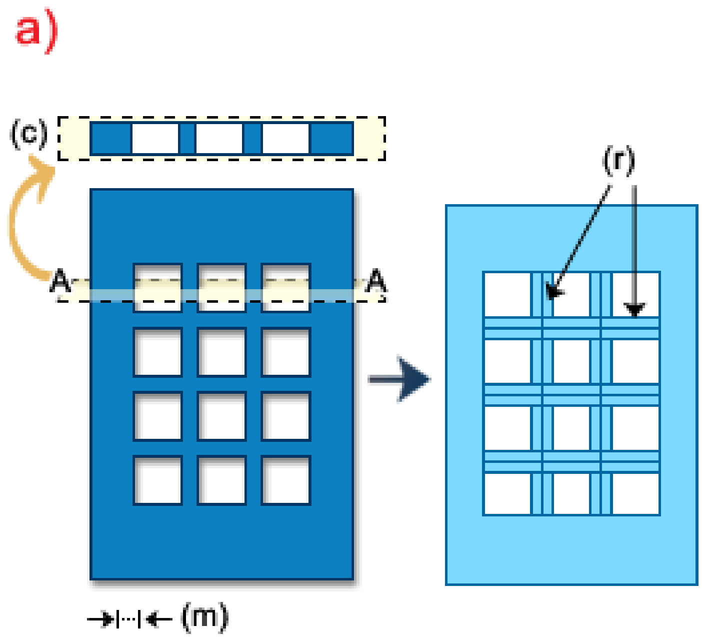


Figure 10: (c) Cross-section A-A, (m) Mesh size, (r) Beam (runner) elements

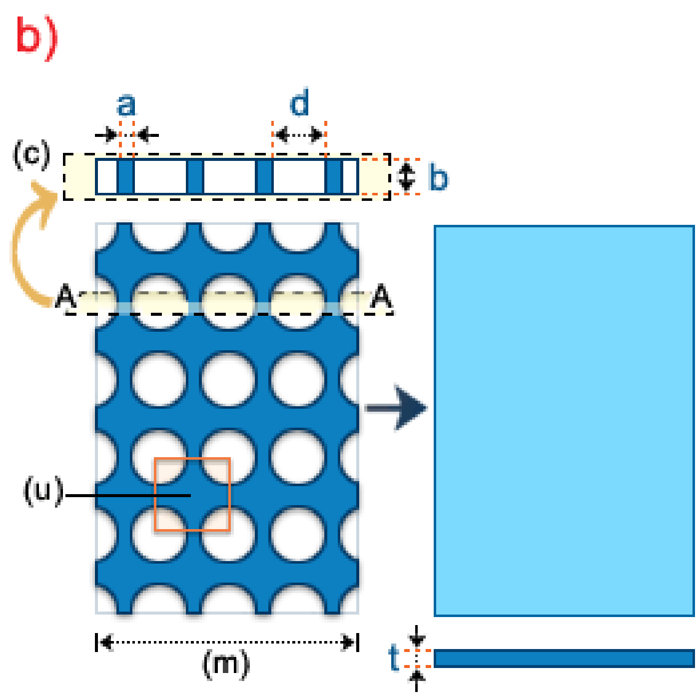


Figure 11: (c) Cross-section A-A, (m) Mesh size, (u) Basic Unit

A=a+d2 S=2+d2−π22+π V=a+d2−π42 t=VA S2

Worked example

To model a grill with a hole size (d) of 2 mm and a distance between holes (a) of 1.5 mm in a part that is 3 mm thick, first determine an area that is representative of the grill as the basis of the calculations. This is the area *u* in the above diagram.

The **Projected Area (A)** is the area of *u*. A=a+d2 or A = 12.25.

The **Contact Surface (S)** is [2 x Projected area (top and bottom surfaces)] – [2 x area of the hole] + [Circumference of the hole x hole depth(b)]. This reduces to S=2⊠+d2−π2⊠2+π⊠⊠. Substituting our values, we get S = 37.07.

The **Volume (V)** is [Volume of the Projected area] – [Volume of the hole]. This equals V=a+d2⊠−π4⊠2⊠ or V = 27.33.

The **Equivalent Thickness (t)** is t=VA or 27.33/12.25 = 2.23mm.

The **Shape Factor** is [Contact Surface area (S)] / [2 x Projected Area (A)], or 1.51.

Select the grill area on your Midplane meshed model. Right-click and select **Properties** (alternatively, click **Geometry tab > Properties panel > Edit**). In the **Cross-section is** box, select **Other shape**. Click **Edit dimensions** and enter the calculated values for Equivalent thickness (2.23) and Shape factor (1.51) in the **Cross-Sectional Dimensions** dialog that appears.

NOTE: Grills are a commonly used design feature.There is an option in the **Cross-section is** drop down box for the automatic calculation of Equivalent Thickness and Shape Factor for grills with either square or round holes. Select the appropriate grill type from the dropdown box, click **Edit dimensions** and enter the grill geometry. The appropriate factors will then be used by the program.

Tetrahedral elements

Tetrahedral elements are four-noded elements that are used in 3D meshes.

The tetrahedral element, which is used to provide an accurate 3D representation of a thick or solid part, has four nodes, four faces and six edges.

Tetrahedral elements

The **Create Tetras** command allows you to create a three-dimensional, tetrahedral mesh element from specified nodes.

Creating tetrahedral elements

1 Click **Mesh tab > Mesh panel > Create Tetras**.

2 Enter the **Node 1** or select it on the model.
3 Enter the **Node 2** or select it on the model.
4 Enter the **Node 3** or select it on the model.
5 Enter the **Node 4** or select it on the model.
6 **Optional:** Select **Inherit properties from neighbors** if the tetra is to be created with the same properties as the neighboring elements.
7 Click **Apply**.

The Tetra will be created using the four specified nodes.

Tetrahedral elements

These tools are used to repair mesh defects.

To access this tool, click (**Mesh tab > Mesh panel > Create Tetras**).

Create Tetras tool

Use the **Create Tetras** tool to create tetrahedral (3-dimensional) mesh elements. The new tetra sits within the 4 nodes you select.

To use this tool, click (**Mesh tab > Mesh panel > Create Tetras**).

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Meshing the model

2

After you create or import an unmeshed geometry model, you must mesh the model before it can be analyzed.

The mesh consists of triangular elements connected with nodes, and provides the basis for the analysis. The three supported finite element mesh types are: Midplane, Dual Domain, and 3D.

The mesh

This example shows you how to model a basic plate and mesh it. You can select which mesh type you want to use, but midplane is selected by default, due to the simplicity of the model.

Modeling example: creating and meshing a flat plate

The following procedure illustrates basic modeling functionality:

1. *Create a new project* and *Create a new study*
2. Click **Geometry tab > Create panel > Curves > Create Line** to open the **Create Line** dialog.
3. Enter **0 0 0** in the **First Coordinates (x,y,z)** box and **0 100 0** in the **Second Coordinates (x,y,z)** box, then click **Apply**.
4. Repeat the previous step for the following coordinates:

   100 100 0, **100 0 0**, and **0 0 0**.
5. Click **Close** to close the **Create Curves—Create Line** dialog.
6. Click **Zoom All**, then click **Orbit** to rotate the model to a suitable viewing angle.
7. Click **Geometry tab > Create panel > Regions > Region By Boundary** to open the **Region by Boundary** dialog.
8. Click on one of the sides of the plate.

   It will turn red to indicate that it is selected.
9. Whilst holding down the **Ctrl** key, click on each of the remaining 3 sides of the plate going around the periphery in a consistent sequence.
10. Click **Apply** to create the region and then click **Close** to close the **Region by Boundary** dialog.

You will now assign a thickness to the plate.

11 Click  (**Select**), then click on the plate.

The plate will turn red to indicate that it has been selected.

12 Right-click in the **Model** pane and select **Properties**. If a message is displayed indicating that no properties have yet been assigned to the part, click **Yes** to assign a new property.

13 If a **Select Entity** dialog appears, click on **Region** and then **OK**.

14 In the **Part Surface** tab, specify a thickness of **2** mm, enter **2 mm part surface** in the **Name** box and click **OK**.

The **Part Surface** property will now be listed in the **Change Property Type To** dialog. Click **OK** to accept the changes and close the dialog.

15 In the **Study Tasks** pane, double-click on the **Create Mesh** task and then click **Mesh Now**.

This completes the modeling task. You should now continue with other tasks in the **Study Tasks** pane to set up and run an analysis on the plate.

The mesh

The Customize Commonly Used Mesh Types dialog is used to edit the model or workspace properties.

Customize Commonly Used Mesh Types dialog

To access this dialog, select **Tools > Workspace > Customize**, and then select **Mesh Types**.

TIP: You can also select **Mesh > Set Mesh Types > Customize Mesh Types** to access this dialog.

About this dialog

This dialog is used to edit the default list of mesh types available from the current workspace. When the workspace is set up to suit your requirements, it can be saved for use on this or any other computer on the local network.

Meshing the model

After you create or import an unmeshed geometry model, you must mesh the model before it can be analyzed. The mesh consists of triangular elements connected with nodes, and provides the basis for the analysis.

Using the Generate Mesh... tool

The three supported finite element mesh types are: Midplane, Dual Domain, and 3D. To mesh a 3D model, you must first create a Dual Domain mesh, fix any mesh problems, and then remesh the model with a 3D mesh.

NOTE: In the **Layers** pane, select the model regions that you want meshed. Only the visible model layers are meshed.

---

1 Click  **Mesh tab > Mesh panel > Generate Mesh**.

The **Generate Mesh** dialog allows you to preview the mesh, change the meshing parameters and save the parameters for future use.

2 As necessary, select the **General**, **CAD**, **NURBS**, **Tetra** or **Tetra Advanced** tab, depending on the type of mesh you are generating, and make changes to the meshing parameters.

NOTE: Use the **3D Mesher** option from the **Tetra** tab to choose between the Legacy mesher or the Advancing Front mesher. The Advancing Front meshing technique allows you set the **Maximum allowed edge length through thickness** as well as the **Maximum allowed aspect ratio**.

TIP:

- Click **Save Default Values** to save all the parameters marked with a * as the default value.
- Once the parameters are saved, they can be included as part of any user-defined or company workspace.
- Click **Use Default Values** to reset all the parameters marked with a * to the current default option.

---

3 Click **Mesh Now.**

The model will be updated immediately with the new mesh.

NOTE: The mesh generator is run in the Job Manager. This enables you to choose when and where you would like the meshing task run. For example, you may run the meshing task on a different machine, and continue working on another task while the mesh is generated.

Changing the mesh type and remeshing the part

Before running a 3D Fill+Pack analysis on your model, you must first create a Dual Domain mesh, diagnose and correct any meshing issues, then create a 3D mesh. In order to change from one mesh type to another, follow the instructions below.

1 Right-click on the meshing icon (second from top) in the **Study Tasks** pane and select **Set Mesh Type**.
2 Select the appropriate mesh type; Midplane, Dual Domain, or 3D.
3 Click **Mesh tab > Mesh panel > Generate Mesh**.
4 In the **Generate Mesh** dialog, select the required meshing options.
5 Make any required changes to the mesh control options. Depending on the mesh type you set, tabs appear that allow you to choose CAD, NURBS, or Tetra options.
6 Click **Mesh Now**.

Meshing the model

This dialog is used to generate a mesh for the imported or created geometry.

To access this dialog click (**Mesh tab > Mesh panel > Generate Mesh**).

Generate Mesh tool

This panel is used to define and create a finite-element mesh for a geometry imported into or created in Autodesk Moldflow Insight.

NOTE: Any mesh option marked with a * can be saved as part of a workspace default (**Classic UI** style option only).

To access this panel, click (**Mesh tab > Mesh panel > Generate Mesh**.

NOTE: The type of mesh created (Midplane, Dual Domain or 3D) is determined by the mesh type selected at the **Mesh** node in the **Study Tasks** pane, or from the **Mesh** panel.

The options that are visible and available for selection depend on the selected mesh type.

Remesh already meshed parts of the model	If this option is selected and you click **Mesh Now**, all visible sections of the model will be remeshed.
Place mesh in active layer	Ensures that the mesh layer is selected and made active in the **Layers** pane after the mesh has been created.
Source geometry type (Midplane mesh only)	Specifies the type of geometry that is to be meshed.
Mesh Now	Runs the mesh generation task immediately on the current machine.
Job Manager	Launches the Job Manager, where you can view and control analyses that are running.

General tab	Determines what size elements will be created when the part is meshed, that is, the global mesh density.
NURBS tab	Allows you to match a Dual Domain mesh or smooth a Midplane mesh to assist post-processing after analyzing the model.
Tetra tab (3D mesh only)	Sets the minimum number of elements and maximum aspect ratio to use when generating a 3D mesh.
Tetra Advanced tab (3D mesh only)	Moves and distributes nodes in order to reduce the aspect ratio of generated tetrahedral elements when generating a 3D mesh from a Dual Domain mesh.
Save Default Values (**Classic UI** style option only)	Saves all options marked with a ***** as the default for the current workspace.
Use Default values (**Classic UI** style option only)	Returns all options marked with a ***** back to the default value for the current workspace.

Remeshing the model

When generating a mesh in Autodesk Moldflow Insight, you are provided with the option to **Remesh already meshed parts of the model**

If this option is selected, and you click **Mesh Now**, the entire model will be remeshed.

If this option is not selected, and you click **Mesh Now**, Autodesk Moldflow Insight will mesh only those sections of the model that have not been meshed already.

Typically, it is best to leave the remesh option deselected when meshing a model. This is because it is good practice to construct the mesh in a modular fashion.

Remeshing procedures

For example, when you are importing and meshing a model, runner system or cooling system, the procedures described below can be followed in order to create a suitable mesh for analysis:

Part model only	Create or import a part model, mesh the part model, and then diagnose and correct the part mesh.
Part and runner system	Create or import a runner system model, mesh only the runner system, keeping the remesh option deselected, so you do not disturb the corrected part

	mesh, and then diagnose and correct the runner mesh.
Part and cooling system	Create or import a cooling system model, mesh only the cooling system, keeping the remesh option deselected, so that you do not disturb the corrected part or runner mesh, and then diagnose and correct the cooling system mesh.

Meshing methods

There are two methods which may be used to generate the mesh on imported Nonuniform Rational B-Splines (NURBS) surfaces.

Advancing Front

This is the default meshing method. It places nodes on the surface, then places nodes in the interior of the surface being meshed. It creates elements using these nodes. This method allows more control in the placement of the nodes, allows better conformance to the part surface, and can implement local density variations more smoothly. This method, which was introduced in MPI 5.1, is highly recommended for parts with fillets and other highly curved surfaces. Two options available with this meshing method were introduced in MPI 6.0 to improve aspect ratio; these options are off by default.

Optimize aspect ratio by surface curvature control	With this option on, mesh sizing is automatically adjusted to the local curvature on NURBS surfaces. This leads to better quality triangles with aspect ratios that approach that of an equilateral triangle.
Optimize aspect ratio by proximity control	With this option on, proximities between boundary curves are detected automatically and adequate mesh refinement is ensured at locations where boundaries are close to one another.

Legacy

This is the original meshing method that is available in all Autodesk Moldflow Insight releases. This meshing method places nodes on the surface, then creates elements using a simple “trim and notch” strategy. This method is the faster of the two, and is recommended for parts which do not have fillets and highly curved surfaces.

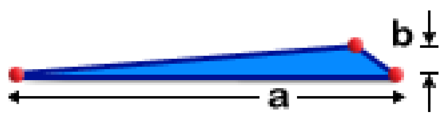
Midplane mesh

A Midplane mesh consists of a web of 3-noded triangular elements and forms a 2D representation of a solid model.

The Midplane mesh provides the basis for the Fill+Pack analysis. This topic provides you with some Midplane mesh considerations.

Aspect Ratio

The aspect ratio of mesh elements can affect analysis performance. High aspect ratios can cause a slower analysis, and affect results. With the longest side in the direction of flow, the end node of high aspect ratio elements will add an excessive resistance factor to flow front calculations. Avoid very high aspect ratio triangles, which have their longest side in the direction of flow. If very high aspect ratios cannot be avoided, the longest side should, if possible, be at right angles to the flow direction. Click for a more detailed aspect ratio definition.



Mesh Density

Increase the mesh density until there is no significant change in result detail. The best solution for controlling mesh density, is to apply a uniform mesh density across the part, and then refine the mesh in areas of interest. In general, we recommend that the mesh be refined in areas where rapid changes in conditions occur (for example, at the gate).

In general there should be at least 2 elements across each surface. Around holes and other obstructions there should be at least 3 elements between the obstruction and the adjacent surface, so that weld lines and other transient effects are highlighted. The ideal mesh is sufficiently fine to enable reasonable accuracy and detail while not using excessive computer time. A fine mesh will give you more detailed results but will increase the analysis time.

TIP: Ensure that mesh size is proportionate to wall sections, with a minimum size of 1.5 times the wall thickness or greater.

Mesh Orientation

Mesh orientation is used to provide a consistent means of differentiating between the two sides of a two-dimensional, 3-noded element. The simplest convention is to call one side of the element the top, and the other side the bottom.

When viewing the mesh orientation, the top side of the element is shown blue, and the bottom side is shown red. In general, for a Midplane model, the elements on one side should all be blue, or all be red.

Midplane mesh

You can edit the thickness property of Midplane models, allowing you to change mesh thickness without having to modify the part in your CAD system and then re-import the part.

Changing the mesh thickness

NOTE: Editing the mesh thickness property should be performed when only a fill or Fill+Pack analysis is required. The analysis accuracy will be compromised if a Cool analysis is run anywhere in the sequence.

1. Select the entities whose thickness you want to change.
2. Right-click on a selected entity and select **Properties...** from the pop-up menu.

   The **Part surface (Midplane)** dialog appears.
3. Select the **Part Surface Properties** tab.
4. Select **Flat** in the **Cross-sectional shape** drop-down list.
5. Enter the required thickness value, and click **OK**.

   NOTE: If you select a cross-sectional shape other than flat, click **Edit dimensions...**, enter the required values in the **Cross-Sectional Dimensions** dialog, and click **OK** twice.

Midplane mesh

This dialog allows you to change the thickness of the mesh, without having to modify the part in your CAD system and re-import.

To access this dialog, select the elements whose thickness you want to change, then click **Mesh tab > Properties panel > Edit**.

Part Surface (Midplane) dialog—Part Surface Properties

The **Part Surface Properties** tab of this dialog is used to specify the part surface properties of the selected elements or regions of type **Part surface (Midplane)** on a Midplane model.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part Surface (Midplane) dialog—Mold Properties

The **Mold Properties** tab of this dialog is used to specify the properties of the mold blocks in contact with the select elements or regions of type **Part surface (Midplane)**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part Surface (Midplane) dialog—Overmolding Component

The **Overmolding Component** tab of this dialog is used to specify which stage/component in the overmolding process the selected elements or regions of type **Part surface (Midplane)** relate to.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part Surface (Midplane) dialog—Eccentricity

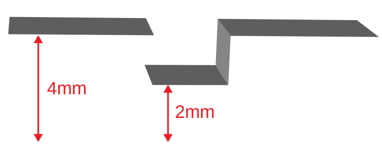
The **Eccentricity** tab of this dialog is used to specify an optional centerline offset for the selected elements or regions of type **Part surface (Midplane)** on a Midplane model.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Eccentricity example

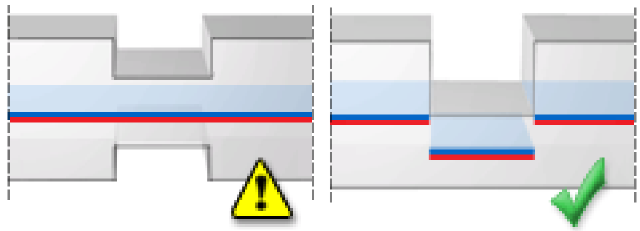
The illustration below shows a part with a negative offset, that is, an offset towards the bottom side of the element.

Dimensions of part:



Midplane model: -1mm eccentricity:

Solver interpretation of Midplane:



Part Surface (Midplane) dialog—Mold Temperature Profile

The **Mold Temperature Profile** tab of this dialog is used to select the temperature profile for the selected elements or regions.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Dual Domain mesh

Dual Domain analysis technology allows you to perform detailed analyses directly on thin-wall, surface meshed models imported from Autodesk Moldflow Adviser or an external CAD package.

The Dual Domain analysis technology removes the need to midplane the geometry for an analyses, significantly reducing model preparation time. The surface mesh analysis works by simulating the flow of the melt on both the top and bottom parts of the mold cavity.

NOTE:

1. The recommended average aspect ratio for a surface mesh to be used in a Dual Domain analysis is less than 6, while the maximum individual value is less than 20.
2. The absolute maximum aspect ratio for Dual Domain meshes that will be later remeshed as a 3D tetrahedral mesh is 30.

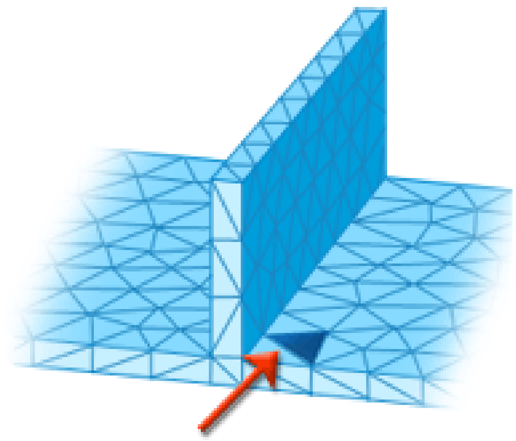
Mesh Orientation

Mesh orientation is used to provide a consistent means of differentiating between the two sides of a two-dimensional, 3-noded element. The simplest convention is to call one side of the element the top, and the other side the bottom.

When viewing the mesh orientation, the top side of the element is shown blue and should face outwards, and the bottom side is shown red and should face toward the inner of the part.

Boundary edges

A boundary edge is an element edge that is not connected to any other elements. Boundary edges are valid in a Midplane model but must not be present in a surface meshed model for Dual Domain analyses. Boundary edges indicate holes or tears in the mesh and must be corrected either in the original CAD system used to create the model, or, using the mesh editing tools in Autodesk Moldflow Insight.



Mesh matching

If you intend to run a Fiber Warp analysis on a surface mesh model using Dual Domain analysis technology, you must ensure that it is meshed with the Autodesk Moldflow Insight mesh translator, or the Autodesk Moldflow Design Link translator. “Mesh matching” technology has been included in the Autodesk Moldflow Insight and Autodesk® Moldflow® Design Link translators, to ensure that elements on opposite skins of a surface mesh match one another spatially, as best as possible. This is especially important for Fiber Warp analysis because inconsistent fiber orientation on both skins arising from mesh anomalies, rather than true flow behavior in the cavity, can adversely affect the warpage prediction.

Dual Domain mesh

These dialogs are used to create a dual domain mesh and change it's thickness without having to modify the part in your CAD system and re-import it.

Creating a Dual Domain mesh from a 3D mesh

Converting a 3D mesh to a Dual Domain mesh is useful when modeling inserts where you want to have an exact match between the elements of the part (3D mesh) and the elements of the insert (Dual Domain mesh).

NOTE: Ensure the 3D mesh of the model does not have any errors.

1. Change the mesh type of the model to Dual Domain.
   a. Right-click on the meshing icon (second from top) in the **Study Tasks** pane and select **Set Mesh Type**.
   b. Select **Dual Domain**.
2. Select **Mesh tab > Mesh panel > Generate Mesh**.
3. Select the **General** tab to view the Target edge length information.
4. Ensure the **Remesh boundary** check box is not selected.
5. Select **Mesh Now**.

Changing the thickness on a Dual Domain mesh

You can edit the thickness property of Dual Domain models, allowing you to change mesh thickness without having to modify the part in your CAD system and then re-import the part.

NOTE: Editing the mesh thickness property should be performed when only a fill or Fill+Pack analysis is required. The analysis accuracy will be compromised if a Cool analysis is run anywhere in the sequence.

1. Select the entities whose thickness you want to change.
2. Right-click on a selected entity and select **Properties...** from the pop-up menu.

   The **Part surface (Dual Domain)** dialog appears.
3. Select the **Part Surface Properties** tab.
4. Select **Specified** in the **Thickness** drop-down list, enter the required thickness value, and click **OK**.

TIP: You only need to select a single entity, then select the check box that will apply changes to all entities that share the properties of the selected entity.

Exporting a surface mesh for use with Autodesk Moldflow Adviser

You can export a **Surface mesh for AMA (*.amm)** file to prepare a Dual Domain model for import into Autodesk Moldflow Adviser software.

The current study must include a Dual Domain meshed model for the **Surface mesh for AMA (*.amm)** export format option to be available.

This format allows you to export a known surface mesh from Autodesk Moldflow Insight for import into Autodesk Moldflow Adviser.

1. Click then **Export > Model** .
2. Navigate to the location where you want to store the exported file.

3 In the **Save as type** drop-down list, select **Surface mesh for AMA (*.amm)**.
4 In the **File name** box, enter a name for the exported file, and then click **Save**.
5 Click **OK**.

The new *.amm file is available for import into Autodesk Moldflow Adviser and subsequent analysis without any changes to the mesh.

NOTE: A **Surface mesh for AMA (*.amm)** file only includes the portion of the Dual Domain model that is meshed with triangles; typically this is the part only. Other model entities, such as beams, and boundary conditions, such as injection locations, are not included in this export file format.

Dual Domain mesh

This dialog allows you to create a dual domain mesh and edit the thickness without having to modify the part in your CAD system and re-import the part.

Part Surface (Dual Domain) dialog—Part Surface Properties

The **Part Surface Properties** tab of this dialog is used to specify the part surface properties of the selected elements or regions of type **Part surface (Dual Domain)** on a Dual Domain model.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part Surface (Dual Domain) dialog—Mold Properties

The **Mold Properties** tab of this dialog is used to specify the properties of the mold block in contact with the selected elements or regions of type **Part surface (Dual Domain)**.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part Surface (Dual Domain) dialog—Mold Temperature Profile

The **Mold Surface Temperature Profile** tab of the **Part surface** dialog is where the local surface temperature profile for the selected area is selected and edited if required.

Part Surface (Dual Domain) dialog—Overmolding Component

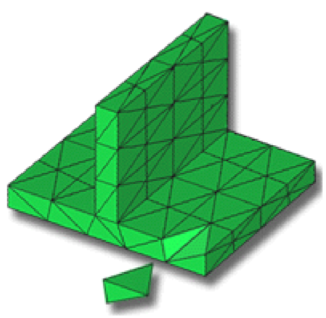
The **Overmolding Component** tab of this dialog is used to specify which stage/component in the overmolding process the selected elements or regions of type **Part surface (Dual Domain)** relate to.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

3D mesh

A 3D mesh represents the CAD model by filling the volume of the model with four-node, tetrahedral elements (tetra).

3D meshes work well for parts that are thick or solid because tetra give a true 3D representation of the model. A 3D analysis does not make the assumptions that are made for Midplane or Dual Domain analyses. Therefore, 3D analyses often require additional computational time to complete. This makes a 3D mesh more appropriate for thick models with complicated shapes, while Midplane and Dual Domain meshes are more applicable for thin-walled, shell-like parts.



3D mesh

When you import a model, you select the type of mesh you would like to work with for that model. Once imported, you can change the density of selected tetrahedral (3D) elements in a portion of the model, or various 3D element properties, if you need to.

Changing the properties of 3D tetrahedral elements

You can edit various part surface properties of 3D tetrahedral elements.

1 Select the entities whose properties you want to change.

2 Right-click on a selected entity and select **Properties...** from the pop-up menu.

The **Part (3D)** dialog appears.

3 Select the **Part Surface Properties** tab.

4 Edit the property options as required, and click **OK**.

Changing tetrahedral (3D) mesh density in an area

The Remesh Tetras tool allows you to selectively change the density of the tetrahedral (3D) elements in a portion of the model.

1 Click **Mesh tab > Mesh Repair panel > Remesh Tetras**.

2 Specify the desired **Selections** option to select tetrahedral elements to be remeshed. Options include:

Remesh all tetras in active layer Enables all tetrahedral elements in the currently active layer to be remeshed without the need to explicitly select regions of the model. Use this option when optimizing sections of the mesh that would be time consuming to reselect, for example to iteratively test various mesh settings. If necessary, use the model entity selection tools to select individual mesh elements or regions of the mesh and assign the selected elements to a new layer.

NOTE: When the computer you are using has Far Eastern character sets installed, use this option as the default selection method. This is necessary due to an issue with speed of operation. This issue only occurs when Far Eastern character sets are installed.

Select a region Enables the selection of a specific region in the current layer, which may include a single tetrahedral element to multiple elements. Use the model entity selection tools to select individual mesh elements or regions of the mesh.

3 Set the **Target number of elements through thickness** of the area to be remeshed. Specify a higher value than the default of 6 to increase the level of result precision through the thickness of the area.

4 Specify the desired **Surface** options to control how the surface will be remeshed. Options include:

Do not remesh surface Select this option to prevent the surface mesh from being refined. Use this option when you want to refine the **Target number of elements through thickness** and leave the granularity of the surface mesh unchanged.

Remesh surface by edge length Select this option to specify to what extent the surface mesh will be refined. Use this option to refine the granularity of result in across the surface of the model. Leave the **Target number of elements through thickness** set to the default value of 6 if you do not want to refine the mesh through the thickness.

TIP: Use the Measure tool to establish the edge length of existing elements.

---

5 Specify **Selection Options** to enable model properties to be assigned to the remeshed area of the model.
6 Click **Apply**.

3D mesh

These dialogs are used to create a 3D mesh and change the properties of that mesh.

Part (3D)

This dialog allows you to save a collection of property values to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Part (3D) dialog—Part Surface Properties

The **Part Surface Properties** tab of this dialog is used to specify the properties of the selected elements.

The set of property values defined by the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

NOTE: 3D-Cool analysis does not support the use of occurrence numbers.

Part (3D) dialog—Mold Properties

The **Mold Properties** tab of this dialog is used to specify the properties of the mold block in contact with the selected tetrahedral elements of type **Part (3D)**.

The set of property values defined by the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

NOTE: 3D-Cool analysis does not support the use of occurrence numbers.

Part (3D) dialog—Surface tension properties

The **Surface tension properties** tab of this dialog is used to specify the surface tension of the part material being injected.

You can choose to use data from the materials database, if your material is in the database, or you can specify your own values.

Part (3D) dialog—Overmolding Component

The **Overmolding Component** tab of this dialog is used to specify which stage/component in the overmolding process the selected elements or regions of type **Part (3D)** relate to.

The collection of property values defined on the dialog are saved to a property set with the description shown in the **Name** box. In addition, you may be given the option to also apply the property values to related entities in the model.

Mesh density

Mesh density is the number of elements per unit area in a mesh.

Higher density meshes usually produce more accurate analysis results, but take longer to analyze. You can mesh different areas with different densities. Using a higher density mesh on important features generally provides more accurate results in those areas.

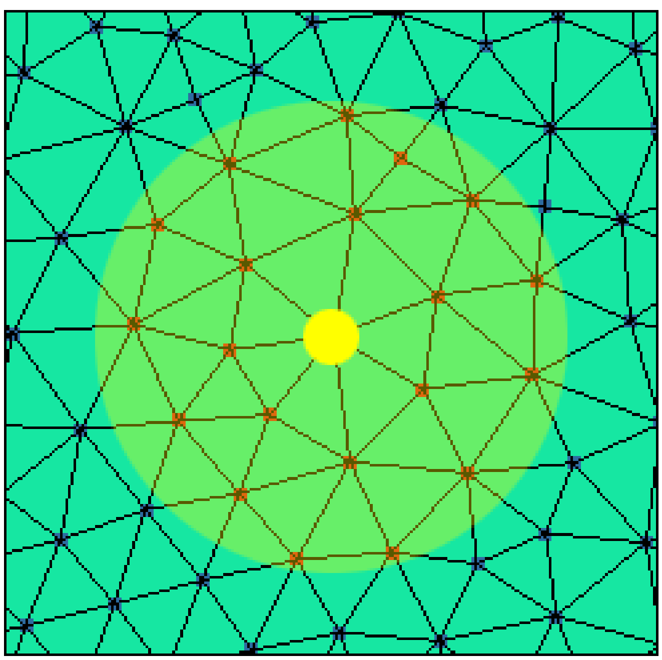


Figure 12: Coarse Mesh

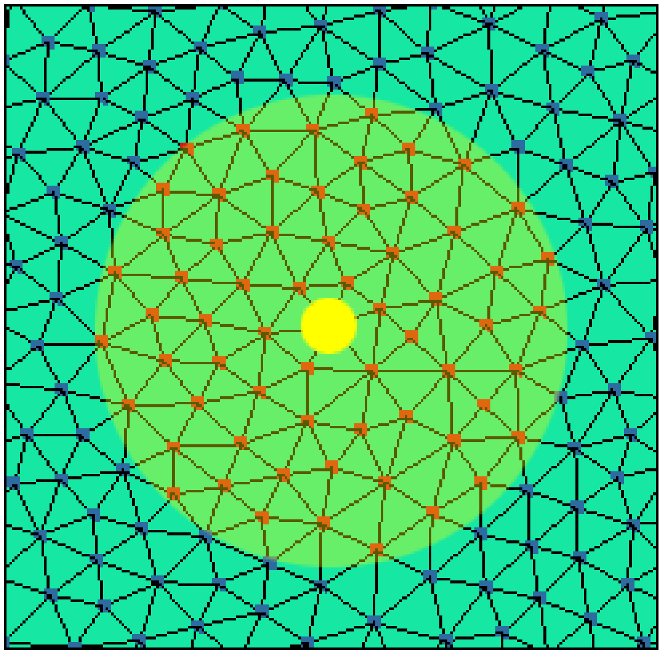


Figure 13: Fine Mesh

Mesh density

The mesh density can be changed for edges, surfaces and specified areas.

Increasing/decreasing mesh density along edges

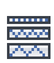
Before you define your mesh size, it is important to understand the starting point, as different model types allow the use of different functionality in the **Define Mesh Density** dialog. For example, (*.stl) models are imported as a solid model, whereas (*.iges) models contain separate surfaces and entities, and make definition in localized areas more feasible.

The following instructions use an (*.iges) model to illustrate the process of increasing or decreasing mesh size. Although the same, if not similar process can be applied to any other type of model.

TIP: To make surfaces more visible once selected, click  **> Options**. Select the **Default Display** tab, and then change the **Surface** drop-down option to **Solid**.

---

1 Open or import an (*.iges) model.

2 Click  **Mesh tab > Mesh panel > Density**.
The **Define Mesh Density** dialog appears.

3 Click a section of the model.

The Explorer view within the **Define Mesh Density** dialog updates depending on what type of feature you have selected. For example, **Loop<x> Curve<x>** indicates a curve.

4 Click + **Part surface** to make visible curve properties comprising the chosen surface, then click an entry in the Explorer view.

The surface or curve will be highlighted on your model.

5 You can now either enter a **Global edge length** so the chosen surface mesh is constrained to a global size, or deselect **Use global edge length** to enter a new figure in the **Target edge length** box or enter a new figure in the **Target number of divisions** box.

6 Click **Apply**.

TIP: The **Global edge length** is the same size as seen in the **General** tab of the **Generate Mesh** dialog.

Increasing/decreasing mesh density on surfaces

Before you define your mesh size, it is important to understand the starting point, as different model types allow the use of different functionality in the **Define Mesh Density** dialog. For example, (*.stl) models are imported as a solid model, whereas (*.iges) models contain separate surfaces and entities, and make definition in localized areas more feasible.

The following instructions use an (*.iges) model to illustrate the process of increasing or decreasing mesh size. Although the same, if not similar process can be applied to any other type of model.

TIP: To make surfaces more visible once selected, click **> Options**. Select the **Default Display** tab, and then change the **Surface** drop-down option to **Solid**.

---

1 Open or import an (*.iges) model.

2 Click **Mesh tab > Mesh panel > Density**.

The **Define Mesh Density** dialog appears.

3 Click a section of the model.

The Explorer view within the **Define Mesh Density** dialog updates depending on what type of feature you have selected. For example, **Loop<x> Curve<x>** indicates a curve.

4 You can now either enter a **Global edge length** so the chosen surface mesh is constrained to a global size, or deselect **Use global edge length** and enter a new figure in the **Target edge length** box.

5 Click **Apply**.

TIP: The **Global edge length** is the same size as seen in the **General** tab of the **Generate Mesh** dialog.

Increasing/decreasing mesh density in an area

The Remesh Area dialog modifies the selected elements by remeshing them to the specified edge length.

1. Click **Mesh tab > Mesh Repair panel > Remesh Area**.
2. Enter the element numbers into the **Select entities to remesh** drop-down list, or select the appropriate area on the model.
3. Enter the **Target edge length**.
4. **Midplane only:** Select the required **Transition**, which specifies how coarse or fine the transitional mesh will be between two areas.
5. Click **Apply**.

   Autodesk Moldflow Insight will modify the specified elements to the specified edge length value.

Mesh density

This dialog is used to set local mesh densities, either finer or more coarse than the global mesh density.

To access this dialog, click (**Mesh tab > Mesh panel > Density**).

Define Mesh Density tool

This panel is used to set local mesh densities, either finer or coarser than the global mesh density, in selected areas of the part. To access this panel, click (**Mesh tab > Mesh panel > Density**).

As you select entities in the model, they are listed in the window on the left side of the panel. You can configure the mesh density for each entry individually by first selecting the entry and then adjusting the value on the right. Higher level entities are also broken down into components, for example, the individual curves of a region are displayed as subentries of the region. This means that you can set the mesh density for specific sides of a region.

TIP:

- An imported STL model will appear as a single entity so it is not possible to define local mesh densities directly to the STL model. You can however use the Create Regions mesh tool to create regions from the STL model and then define local mesh densities to the region entities.
- You can click directly on the model to automatically select multiple elements that belong to a face or body, depending on the selection priority you set initially. For example, you can select multiple faces and edges of a body, or multiple edges of a face.

Using different types of meshes in the same model

As a general rule, mixing meshes is not supported. However, some exceptions can be considered with elements used in structural analyses, and beams used in Flow or Warp analyses.

Beams Generally, beams are used to model feed systems regardless of the type of mesh initially used. However, it is possible to use Part beam properties to represent model features in Dual Domain or Midplane mesh models.

NOTE: Beam elements can be used in Flow and Warp analyses.

Midplane elements in a Dual Domain mesh A Dual Domain mesh model can be mixed with Midplane elements, typically to create a Midplane gate. This is only supported by the Flow and Warp solvers.

NOTE: Midplane elements used in a Dual Domain mesh model will be excluded from a Warp analysis.

Structural analyses Elements used in structural analyses can use a different mesh than the one used initially for the analysis, like in the examples below.

Analysis	Study mesh type	Paddle/Core/Wire mesh
Paddle-shift analysis	3D / Dual Domain	(Paddle) 3D / Midplane
Core-shift analysis	3D / Dual Domain / Midplane	(Core) 3D
Wire sweep analysis	3D / Dual Domain / Midplane	(Wire) Beam

Warped mesh/geometry

If you have run a warp analysis and have warp results, you can export the warped mesh/geometry for use in another applications. Likewise, you can import warped mesh/geometry from a previous warp analysis into a new study.

Importing warped mesh/geometry is similar to importing any file. You can export the mesh so that it shows the Actual Warp deflection result or the opposite deflection.

Warped mesh/geometry

You can import or export the results of a warp analysis.

Importing the warped mesh/geometry

You can import a warped mesh/geometry exported from a previous analysis into a new study. The procedure is similar to importing any file.

1 Click **Home tab > Import panel > Import**.

The **Import** dialog appears.

2 Navigate to the file you want to import.

The format of this will be either: ASCII STL (*.stl), Binary STL (*.stl), or Moldflow Study file (*. sdy).

3 Click **Open**.

Exporting the warped mesh/geometry

You can export the warped mesh/geometry in STL format, or Moldflow study file format, so that it can be opened in another application.

To use this function, you must have Warp results.

There are two ways to export a warped mesh/geometry: export the mesh so that it shows the **Actual** Warp Deflection result, or export the mesh so that it shows an **Opposite** deflection and so the negated geometry.

1 Navigate to a Deflection result in the **Study Tasks** pane and click it.

The selected result is displayed on the model.

2 Click **Results tab > Export and Publish panel > Warp Shape**. The **Export Warpage Mesh/Geometry** dialog appears.

3 Click a file format type option button in the **Format** area.

Your selection will depend on the type of application to which you will export.

4 Click either **Actual** or **Opposite** in the **Direction** area.

5 Enter a **Scale factor** if you want to visually magnify the warpage, and click **OK**.

The **Save As** dialog appears.

6 Enter a **File name** and navigate to the required location, and then click **Save**.

Warped mesh/geometry

Saving or exporting warped mesh/geometry data allows you to view them externally.

The save or export warp data function is accessed from **Results** on the toolbar.

Export Warpage Mesh/Geometry dialog

This dialog is used to export the deformed geometry predicted from a Warp analysis as an STL or UDM file format.

The geometry can be exported either with the actual deflections, or with the opposite deflection, that is, a representation of how the mold would need to be tooled to compensate for the predicted warpage.

To access this dialog, ensure you have selected a Deflection result in the **Study Tasks** pane. Click **Results > Export Warpage/Mesh Geometry**

Aggregated mesh solver

Mesh aggregation is a feature that is incorporated into the Cool analysis solvers and is recommended to be used in all circumstances. The method works by aggregating similar elements with their immediate neighbors to form larger master elements, reducing the overall number of the elements of the model internal to the solver, and therefore, analysis time.

During the boundary element integration, the elements are all treated individually, as if aggregation was not used, with these integrals aggregated to the master element in the building of the system matrices.

The mesh aggregation scheme principles include:

- Mesh aggregation is only applied to triangular or shell elements in both part and mold models.
- For 3D analysis, the outer facets of the boundary of the tetrahedral mesh are separated from the part mesh. This mesh is used for performing the boundary element calculation on, and acts as the interface between the part model and mold model. Mesh aggregation is performed on these facets or the triangular mesh.
- The mesh aggregation routine will only aggregate neighboring elements that have the same element orientation. (Midplane)
- The aggregation routine will only aggregate neighboring elements that are approximately planar. Little mesh aggregation will occur if the model has high curvature.
- An element can only be aggregated into a master element once.
- It is assumed that the elements aggregated into the master element all have the same temperature.

NOTE: Mesh aggregation is not available for 3D Gas-Assisted injection molding.

Mesh diagnostics

3

The diagnostics tools show more detailed information about the problems listed in the Mesh Statistics report.

The **Mesh Diagnostics** panel (**Mesh tab > Mesh Diagnostics panel**) contains the following diagnostics to check the mesh:

Panel item	Use this diagnostic to...	Supported mesh type
Aspect Ratio Diagnostic	Locate elements that have an aspect ratio above a specified limit	
Beam L/D Ratio Diagnostic	Locate cooling channel elements that are too short or too elongated	
Overlapping Elements Diagnostic	Locate elements that overlap or intersect, and therefore occupy the same space	
Orientation Diagnostic	Locate elements whose top/bottom sides are defined inconsistently	
Connectivity Diagnostic	Locate elements that are not connected to the rest of the mesh	
Free Edges Diagnostic	Locate element edges that are not shared, which may indicate gaps or holes in the mesh	
Collapsed Faces Diagnostic	Locate places where opposite faces of a plane share a node, and so have zero thickness	
Dimensional Diagnostic	View model dimension values	
Thickness Diagnostic	Check that thickness values in the mesh are set correctly	
Occurrence Number Diagnostic	Check that occurrence numbers in the mesh are set correctly	

Panel item	Use this diagnostic to...	Supported mesh type
Zero Area Element Diagnostic	Locate very small element	
Dual Domain Mesh Match Diagnostic	Check that there is a good correspondence between the elements on each side of a Dual Domain mesh	
Beam Element Count Diagnostic	Locate cooling channels meshed with insufficient elements	
Trapped Beam Diagnostic	Locate beam elements that are too close together	
Centroid Closeness Diagnostic	Locate triangle elements that are too close together	
Cooling Circuit Diagnostic	Check that each cooling circuit has only one inlet and outlet	
Bubbler/Baffle Diagnostic	Check that bubblers and baffles are modeled correctly	

Checking the mesh before analysis

After meshing the model, it is strongly recommended that you perform a series of diagnostic checks to ensure that the mesh is suitable for analysis.

Autodesk Moldflow Insight provides the following tools for assessing the quality of the mesh:

- Mesh Statistics
- Mesh Repair Wizard
- Mesh Diagnostics

NOTE: If you are changing mesh types from Dual Domain to 3D, you should diagnose and correct any meshing issues on the Dual Domain mesh before generating the 3D mesh.

To obtain the best analysis results, the mesh must be free of errors and meet the following conditions:

- Element aspect ratios must be below the following limits:
  - 6:1 for triangular elements in Midplane and Dual Domain meshes.
  - 30:1 for triangular elements in a Dual Domain mesh that will be converted to a 3D mesh.
  - 50:1 for tetrahedral elements in a 3D mesh.

- In a Dual Domain model, the mesh match ratio must be above 85% for flow analysis, and above 90% for warpage analysis.
- In Midplane and Dual Domain models, significant thickness changes must be represented with at least three rows of elements to ensure accurate prediction of flow issues such as hesitation or racetracking.

Optionally, you can remove small features like corner blends and radii from Midplane and Dual Domain models to reduce the computing time. These features can be analyzed in 3D models, with a very high mesh density and little overall change in the results.

A good mesh is critical for accurate results. The recommendations below will help you to evaluate the mesh and clean up the model.

Visual Inspection

Some anomalies can be identified just by looking at the mesh, such as the mesh density:

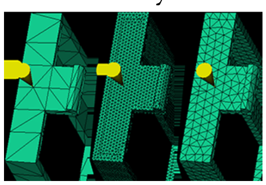


Figure 14: Visual inspection of a Dual Domain mesh

The figure above shows three significantly different mesh densities on the same model.

- The first mesh is a bit coarse (has few elements), and therefore, the mesh match ratio may be low and this mesh may not represent part thickness changes well.
- The second mesh has many more elements than necessary; this will result in a long computing time.
- The third mesh has about the right number of elements. There are several rows of elements on the side, and the elements are approximately equilateral triangles, which indicates acceptable aspect ratios have been achieved.

Mesh Statistics

Once the visual inspection has determined the mesh density looks right, use the Mesh Statistics report to determine if the quality of the mesh is acceptable.

For Midplane and Dual Domain models, the Mesh Statistics report includes the following sections:

- Entity counts.
- Edge details.
- Orientation details.
- Intersection details.

- Surface triangle aspect ratio.
- Match percentage (Dual Domain models only).

For 3D models, the Mesh Statistics report includes the following sections:

- Nodes (Number).
- Elements (Tetras: Number, Volume, Aspect ratio, Maximum dihedral angle; and Beams: Number, Volume).
- Total volume.

Once a problem has been identified in the mesh statistics, corrective action can be taken to fix the problem.

Mesh Diagnostics

The diagnostics tools show more detailed information about the problems listed in the Mesh Statistics report.

These diagnostics include:

- Aspect Ratio Diagnostic (Midplane and Dual Domain models only).
- Beam L/D Ratio Diagnostic.
- Overlapping Elements Diagnostic. This includes intersecting and overlapping elements (Midplane and Dual Domain models only).
- Orientation Diagnostic (Midplane and Dual Domain models only).
- Connectivity Diagnostic.
- Free Edges Diagnostic. This includes free and non-manifold edges (Midplane and Dual Domain models only).
- Collapsed Faces Diagnostic (Dual Domain models only).
- Dimensional Diagnostic (3D models only).
- Thickness Diagnostic (Midplane and Dual Domain models only).
- Occurrence Number Diagnostic.
- Zero Area Elements Diagnostic (Midplane and Dual Domain models only).
- Dual Domain Mesh Match Diagnostic (Dual Domain models only).
- Beam Element Count Diagnostic.
- Trapped Beam Diagnostic.
- Centroid Closeness Diagnostic.
- Cooling Circuit Diagnostic.
- Bubbler/Baffle Diagnostic.

When a diagnostic is displayed, the Diagnostic Navigator toolbar is automatically displayed. This allows you to quickly zoom on elements in the model that were detected by the diagnostic.

Mesh Repair Wizard

After you have evaluated the mesh using visual inspection, mesh statistics and mesh diagnostics, you need to fix the identified problems in order to have a clean model before you run an analysis. This can be done with the Mesh Repair Wizard.

The Mesh Repair Wizard runs a series of diagnostics on your model and, for each diagnostic, allows you to fix any problems detected.

For Midplane and Dual Domain models, the available tools are:

- Stitch Free Edges.
- Fill Hole.
- Overhang.
- Degenerate Elements.
- Flip Normal.
- Fix Overlap.
- Collapsed Faces.
- Aspect Ratio.

For 3D models, the available diagnostics and fixes are:

- Inverted tetras
- Collapsed faces
- Insufficient refinement through thickness
- Internal long edges
- Tetras with extremely large volume
- Tetras with high aspect ratio
- Tetras with extreme angle between faces

CAUTION: Running the Mesh Repair Wizard on 3D entities may alter the location of surface nodes. On multi-component assemblies, this may change the element matching between components and cause unwanted gaps.

Validate the repairs

Whenever an automatic fix is applied to the mesh, the repairs must be validated to ensure that no new problems were created.

After you have completed the mesh clean up, it is recommended that you run the Mesh Statistics report again, and if necessary, run the Mesh Repair Wizard again to confirm that all potential problems have been resolved.

Mesh statistics

Once the visual inspection has determined the mesh density looks right, use the Mesh Statistics report to determine if the quality of the mesh is acceptable.

Specify the element type and report options you want to review.

The Mesh Statistics report includes the following sections, depending on the model mesh type and the element type specified in the input parameters:

- Entity counts
- Area
- Volume
- Aspect ratio
- Edge details
- Orientation details
- Intersection details
- Match percentage (Dual Domain models only)
- Tetra aspect ratios (3D models only)

Once a problem has been identified in the mesh statistics, corrective action can be taken to fix the problem.

Mesh statistics

Use the **Mesh Statistics** tool to identify any mesh quality issues or defects that should be corrected before starting an analysis.

Using the Mesh Statistics report

1. Click  **Mesh tab > Mesh Diagnostics panel > Mesh Statistics**.
2. In the **Input parameters** area, select the **Element type** for which you want to show the mesh statistics summary report.

   Choose **Beam**, **Triangle**, or **Tetrahedra**.
3. If desired, in the **Options** area, select options that affect the content and display of the mesh statistics report.

   The options available for selection depend on the model mesh type.
4. Click **Show** to display the mesh statistics summary report for the element type you specified.
   The summary report appears in a separate window. The summary report includes the following sections, depending on the model mesh type and the specified element type:

   - Entity counts
   - Area
   - Volume
   - Aspect ratio
   - Edge details
   - Orientation details
   - Intersection details
   - Match percentage (Dual Domain models only)
   - Tetra aspect ratios (3D models only)
5. Click **Close** to close the summary report window and return to the **Mesh Statistics** tool.
6. When you have finished reviewing the mesh statistics summary report for each element type, click **Close** to exit the **Mesh Statistics** tool.

For any problems indicated in this mesh statistics summary report, use the tools found on the **Mesh Diagnostics panel** and **Mesh Repair panel** (**Mesh tab**) to display diagnostic details and fix defects.

Mesh statistics

Use these dialogs to set up and display a mesh statistics summary report for each element type in the model.

Mesh Statistics dialog

This dialog is used to set inputs for and display a text report, which summarizes the number of entities and other information that can be used to assess the quality of the mesh.

To access this dialog, click (**Mesh tab > Mesh diagnostics panel > Mesh Statistics**).

In the **Input parameters** section, select the **Element type** for which you want to display the summary report. If desired, in the **Options** area, select options that affect the content and display of the mesh statistics report.

Click **Show** to display the mesh statistics summary report for the element type you specified. The summary report appears in a separate window. The summary report includes the following sections, depending on the model mesh type and the specified element type:

- Entity counts
- Area
- Volume
- Aspect ratio
- Edge details
- Orientation details
- Intersection details
- Match percentage (Dual Domain models only)
- Tetra aspect ratios (3D models only)

Mesh statistics summary report

The mesh statistics summary report comprises the following information, depending on the model mesh type and the specified element type:

Entity Counts

Reports the total number of elements of the specified type, the number of connected nodes associated with those elements, and the number of connectivity regions.

NOTE: The reported number of connectivity regions should be 1.

Area	Reports the total surface area of all Triangle elements in the model.
Volume	Reports the total volume represented by all elements of the specified type.
Aspect Ratio	Reports the minimum, maximum, and average aspect ratios of **Beam** or **Triangle** elements found in the mesh.
Edge Details	Reports the number of free edges, manifold edges, and non-manifold edges. **NOTE:** The interpretation of these statistics depends on the type of mesh. For example, a Dual Domain mesh must have no free edges, whereas free edges are normal for a Midplane mesh.
Orientation details	Reports the number of elements which are not oriented consistently.
Intersection Details	Reports the number of element intersections, fully overlapping elements, and duplicate beams.
Match Percentage (Dual Domain mesh only)	Reports the mesh match percentage and reciprocal percentage for the opposing surfaces of a Dual Domain mesh. **NOTE:** A mesh match percentage of 85% or higher is acceptable for a Dual Domain Fill+Pack analysis. A percentage of 50% or lower will cause the Fill+Pack analysis to abort. For a Dual Domain Warp analysis, the mesh match percentage should exceed 85%.
Tetra aspect ratios (3D mesh only)	Reports the minimum, maximum, and average aspect ratios, and the maximum dihedral angle of **Tetrahedra** elements found in the mesh.

For any problems indicated in this mesh statistics summary report, use the tools found on the **Mesh Diagnostics panel** and **Mesh Repair panel** (**Mesh tab**) to display diagnostic details and fix defects.

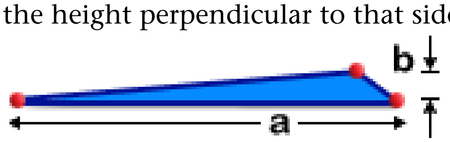
Mesh statistics report

The Mesh Statistics report is divided into a number of different sections that provide a quick overview of various characteristics of the mesh.

- In the **Entity Counts** section, it is important to check the Connectivity regions value. There should only be one region. Any more than this will indicate disconnected sections of the part or runner system.
- In the **Edge Details** section, the integrity of the surface edges are determined.

  Free edges are not connected to another surface. This value should be zero in Dual Domain and 3D meshes.

  A manifold edge is a mesh edge that has two elements attached to it. This is the only edge type that is allowed for a Dual Domain mesh.

  A non-manifold edge has more than two elements attached to it. For a Dual Domain mesh, this value should be zero.
- In the **Orientation Details** section, the **Elements not oriented** value should be zero. Orientation is not a meshing error but a data handling requirement of the program. This will be explained in the Mesh editing tutorial.
- In the **Intersection Details** section, shared surfaces are reported. All values should be zero.
- The **Surface Triangle Aspect Ratio** section refers to the geometry of the mesh elements. The aspect ratio of an element is the ratio of the longest side to the height perpendicular to that side (a / b in the following figure). 

  The aspect ratio value should ideally be less than:

For:	Max. Aspect Ratio
Midplane/Dual Domain	6:1
Midplane/Dual Domain—noncritical areas	20:1
Tetra elements	50:1
Dual Domain mesh before conversion to 3D	20:1
Cool and Warp analysis	6:1

  Very high aspect ratio triangles should be avoided, especially when the longest side is in the direction of flow as this can affect localized results. The report indicates that there is at least one element with a high aspect ratio. You will investigate this later.

- In the **Match Percentage** section, the mesh match values should ideally be 85% or higher. Mesh matching is only applicable for Dual Domain meshes. This is a measure of how elements on one surface correspond with elements on the opposite surface.This measure is very important for correct part thickness determination and fiber orientation prediction.

  The report shows that there are no major defects in the mesh (overlaps, intersections, disconnected elements) but that the aspect ratio of some elements needs improving. You will now run the Mesh Repair Wizard to check for further mesh defects, and to try to correct the high aspect ratio elements automatically.

Using Mesh Diagnostics tools effectively with large models

In Autodesk Moldflow Insight there are five mesh diagnostic tools that can be used for diagnosing problems with the mesh.

These are:

Aspect Ratio	Identifies elements that have an aspect ratio above a specified limit.
Overlapping Elements	Identifies elements that overlap or intersect, and therefore occupy the same space.
Orientation	Helps identify elements whose top/bottom sides are defined inconsistently.
Connectivity	Identifies elements that are not connected to the rest of the mesh.
Free Edges	Identifies element edges that are not shared, which may indicate gaps or holes in the mesh.

Prior to using any of these mesh diagnostic tools, it is recommended that you select: **Mesh tab > Mesh panel > Mesh Statistics** to get an overview of your model quality. This assists you when working with large models. If any issues exist with the mesh, you can then select the relevant diagnostic check.

For models with a large number of elements, Mesh Diagnostics can take a bit longer. The following information focuses on ways to work effectively in these situations.

- Select **Place results in diagnostics layer**. You can then turn off all the remaining layers in the model and work with the elements in the diagnostics layer.
- If the diagnostic check finds a large number of problem elements, it may be worthwhile to create new layers and assign a subset of the problem elements to these layers.

- At the time of mesh editing, turn off the display of mesh diagnostics. You can do this by selecting the keyboard shortcut **Ctrl+D**. This is important, because if the diagnostics display is on, it keeps track of changes made to the mesh, and on large models, this can be memory intensive.
- Change the color of the elements in the diagnostics layer. This can be done by clicking the **Display** option in the **Layers** pane and will help you in separating them from the other elements.
- With certain mesh problems, it may be necessary to view the adjacent elements (for example, when the mesh has large aspect ratio elements or free edge elements). In these cases, use the **Expand** option in the **Layers** pane, which automatically pulls the adjacent elements into that layer, allowing you to modify the mesh.
- Change the Display option to **Text**. When you click the **Show** button, the textual result shows the maximum aspect ratio and the number of elements that have an aspect ratio value between the minimum and maximum values you specified.

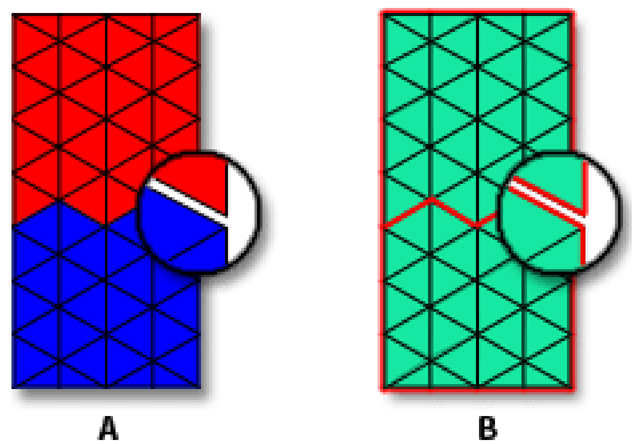
After you have completed the mesh editing, we recommend that you run the mesh diagnostic again to confirm that the problem has been resolved.

Connectivity problems in the mesh

Some imported models (particularly STL models) include sections which are not connected to the rest of the model.

This produces disconnected mesh elements that need to be either connected or removed before running an analysis. Another common connectivity problem occurs when creating a runner system. Both of these examples can be diagnosed with the Mesh Connectivity Diagnostic.

In the Mesh Connectivity Diagnostic highlight report, all elements connected to the selected element are displayed in red, and disconnected elements are displayed in blue. As shown in Figure A below, one half of the elements are not connected to the other half. At the intersection where the elements are not connected, there will also be a free edge (Figure B). The Free Edge diagnostic can be used to locate the free edge. Beam elements can also be disconnected, for instance, when the runners are disconnected from the part.



In the Mesh Connectivity Diagnostic text report, the following statistics are reported (see Figure below):

- **Number of connected entities:** The total number of connected entities in the mesh.
- **Edges:** The total number of element edges in the mesh.
- **Free Edges:** The number of free edges in the mesh. If there are any free edges in a Dual Domain or 3D mesh, you must correct them before running an analysis. A Midplane mesh has free edges at the outer edges of the model.
- **Manifold Edges:** The number of manifold edges in the mesh. Properly connected elements have manifold edges. In a Dual Domain or 3D mesh, only manifold edges are correct.
- **Non-manifold Edges:** The number of non-manifold edges in the mesh. If there are any non-manifold edges in any mesh, you must correct them before running an analysis.

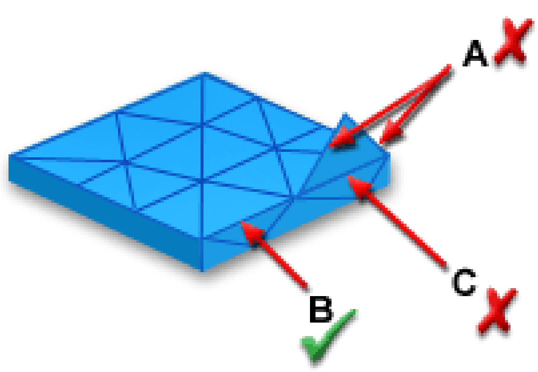


Figure 15: A: Free Edges, B: Manifold Edges, C: Non-Manifold Edges.

Connectivity problems in the mesh

Some imported models (particularly STL models) include sections which are not connected to the rest of the model. You should fix the disconnected mesh elements using the **Mesh Tools** before running an analysis.

Finding connectivity problems in the mesh

If you run an analysis on a model with disconnected elements, the simulated flow of plastic through the part may be interrupted at the discontinuities.

1 Click **Mesh tab > Mesh Diagnostics panel > Connectivity**.

2 Use the **Mesh Connectivity Diagnostic** dialog.

3 Click **Mesh tab > Mesh Diagnostics panel > Show** to toggle the diagnostic display on or off.

4 Fix the elements using one of the following methods.

   - Check the model in your CAD system and re-import if necessary, or
   - Delete the disconnected elements if they are unwanted, or
   - Use the Mesh Repair Wizard.

If the problem still exists, use the Free Edge diagnostic and study the problem areas. Global Merge or Merge Nodes should fix the problem. For example there can be mesh connectivity problems in the model where the tip of a gate is not connected correctly to the runner beam that is attached to the part after creation. In order to remove this type of mesh connectivity problem, click **Mesh tab > Mesh Repair panel > Merge Nodes** and check off **Merge nodes along an element edge only**, if the problem relates to very close or coincident nodes belonging to the runner system.

NOTE: After using Auto Repair tools, carefully inspect the mesh to ensure that it correctly represents the part geometry.

Connectivity problems in the mesh

The **Connectivity** diagnostic is used to check for and locate mesh connectivity problems in a Midplane, Dual Domain or 3D mesh.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Connectivity**).

Mesh Connectivity Diagnostic

In the **Input parameters** section of this dialog, choose which entity you want to start the connectivity check from, and whether or not you want to include beam elements in the check.

In the **Options** section of the dialog, decide how you would like to view the results.

Zero area elements

Zero area elements are very small elements in the mesh. If these exist, they are reported in an error message.

Zero area elements

Zero area elements are very small elements in the mesh. If these exist, they are reported in an error message.

Fixing zero area elements

1 Click **Mesh tab > Mesh Diagnostics panel > Zero Area**.

The **Zero Area Elements Diagnostic** pane is displayed.

2 Set a minimum edge length in the **input parameters** as the tolerance for the diagnostic to identify elements with an area less than the equivalent area.

3 Set the Options you require in the **Zero Area Elements Diagnostic** pane.

- Show zero area elements in the graphics pane or as text results by selecting from the drop down list.
- Use the checkboxes to display the mesh/model, to place the results in a diagnostic layer, and to restrict the display to visible entities only.

4 Click **Mesh tab > Mesh Diagnostics panel > Show** to toggle the diagnostic display on or off.

5 Use appropriate tools in the **Mesh > Mesh Repair panel** to fix the elements.

For example, depending on the specific problem, Global Merge, Merge Nodes, Move Nodes, or Remesh Area may help.

Zero area elements

This dialog is used to find zero area elements.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Zero Area**).

Zero Area Elements Diagnostic

The **Zero Area Elements** diagnostic is used to identify and locate elements in a model with a small or zero area.

Using this tool, you can set a minimum edge length from which the equivalent area is automatically calculated, as the tolerance for the diagnostic to identify elements with an area less than the calculated value.

In the **Options** section of the dialog, decide how you would like to view the results.

Beam element diagnostic

There are several diagnostic tools related to potential beam element problems.

Beam element count diagnostic	displays a plot of the number of beam elements per curve used to model a cooling channel or runner system. A low beam element count may be representative of insufficient mesh density.
Beam L/D ratio diagnostic	displays a plot of the number of the length/diameter ratio of all beam elements in the model. If the ratio is too high, or too low, this may result in solver convergence warnings.
Trapped beam diagnostic	locates overlapping beam elements.

Beam element diagnostic

These dialogs are used to set up the parameters for various beam element diagnostic tools.

Beam Element Count Diagnostic

The **Beam Element Count** diagnostic is used to display a diagnostic plot of the number of beam elements per curve used to model a cooling channel or runner system. A low beam element count may indicate insufficient mesh density and adversely affect solution accuracy.

To access this diagnostic, click (**Mesh > Mesh Diagnostics > Beam Count**).

Beam L/D Ratio Diagnostic

The **Beam L/D Ratio** diagnostic is used to display a diagnostic plot of the length/diameter ratio of all beam elements in the model.

The optimum L/D ratio for cooling channel related beam elements is 2.5. Significantly lower L/D ratios can result in solver convergence warnings in the analysis. Significantly higher L/D ratios may result in solver convergence warnings and a reduction in solution accuracy.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Beam L/D Ratio**).

Trapped Beam Diagnostic

The **Trapped Beam** diagnostic is used to locate beam elements that are overlapping, as indicated by the position of their centroids.

The diagnostic looks for beam element centroids that are contained (or "trapped") inside other beam elements, and highlights them in red.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Trapped Beam**).

Cooling circuit diagnostic

The cooling circuit diagnostic is used to check that the cooling circuit has been modelled correctly.

Each cooling circuit must have at least one inlet and outlet. Correctly modeled circuits will appear blue when the diagnostic is run. Sections of the cooling circuit which are invalid will appear red. The default colour of a cooling channel is dark blue.

Cooling circuit diagnostic

The cooling circuit diagnostic is used to check the cooling circuit for defects.

Using the Cooling Circuit Diagnostic

1. Click **Mesh tab > Mesh Diagnostics panel > Cooling Circuit**.
   The **Cooling Circuit Diagnostic** pane is displayed.
2. Click **Show** to run the diagnostic.
   Correctly modeled circuits will appear (or remain) blue when the diagnostic is run. Sections of the cooling circuit which are invalid will appear red.
3. Use appropriate tools to fix the cooling circuit.
   For example, depending on the specific problem, maybe you need to model a cooling inlet.

Cooling circuit diagnostic

The **Cooling Circuit** diagnostic is used to check whether a cooling circuit has been modelled correctly.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Cooling Circuit**).

Cooling Circuit Diagnostic dialog

This dialog box has no set-up tasks; simply press **Show** to run the diagnostic.

In the **Options** section of the dialog, decide how you would like to view the results.

Bubbler/Baffle Diagnostic

The **Bubbler/Baffle Diagnostic** tool is used to check whether any bubblers and baffles in the cooling circuit model, if present, are modeled incorrectly.

When the diagnostic plot is run, bubbler and baffle elements that are valid will remain orange and yellow respectively. Bubbler or baffle elements which are invalid, and would cause a Cool analysis to display an error message and stop, will apear red.

Bubbler/Baffle diagnostic

The bubbler/baffle diagnostic is used to determine if there are any errors related to any bubblers or baffles you may have modeled.

Using the Bubbler/Baffle Diagnostic

1 Click **Mesh tab > Mesh Diagnostics panel > Bubbler/Baffle**.
  The **Bubbler/Baffle Diagnostic** pane is displayed.
2 Click **Show** to run the diagnostic.
  Correctly modeled bubblers and baffles will remain orange and yellow when the diagnostic is run. If they are incorrectly modeled, they will appear red.
3 Use appropriate tools to fix the problem.
  See *Cooling components* for help on modeling bubblers and baffles.

Bubbler/Baffle Diagnostic

The **Bubbler/Baffle** diagnostic is used to check whether bubblers and baffles in the cooling circuit model, if present, are modeled correctly.

To access this diagnostic, click (**Mesh tab > Mesh Diagnostics panel > Bubbler/Baffle**).

Bubbler/Baffle Diagnostic dialog

This dialog box has no set-up tasks; simply press **Show** to run the diagnostic.

In the **Options** section of the dialog, decide how you would like to view the results.

Mesh repair

4

After you have evaluated the mesh using visual inspection, mesh statistics and mesh diagnostics, you need to fix the identified problems in order to have a clean model before you run an analysis.

To fix mesh problems, you can use the Mesh Repair Wizard, which runs a series of diagnostics on your model and, for each diagnostic, allows you to fix any problems detected.

Alternatively, you can use the mesh repair tools to fix any problems you have identified from the mesh statistics report.

Mesh repair

The **Project Mesh** command is used to project the chosen mesh back towards the surface of the parent surface. This command is useful when your mesh has become unexpectedly deformed and no longer follows the surface of your model.

To use this tool, click  (**Mesh tab > Mesh Repair panel > Project Mesh**).

Project Mesh tool

This panel provides a collection of tools for automatically or manually repairing mesh defects identified using the mesh diagnostic displays in the Mesh menu.

In the **Input parameters** section of the dialog, select the elements you want to project back towards the parent surface of your model.

NOTE: Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Mesh Repair Wizard

The **Mesh Repair Wizard** helps you to repair several types of mesh defect.

You can make changes on each page, or skip the pages that do not concern you.

As you open each page of the Wizard, the model is scanned for defects. You can see the results from each scan and then decide what to do next. The Wizard addresses the errors listed below.

- Stitch free edges
- Fill hole
- Overhang
- Degenerate elements
- Flip normal
- Fix overlap
- Collapsed faces
- Aspect ratio

After you have completed the mesh clean up, it is recommended that you run the mesh statistics report again to confirm that all potential problems have been resolved.

The Mesh Repair Wizard also includes diagnostics and fixes for tetrahedral meshes. The available diagnostics and fixes for 3D meshes are:

- Inverted tetras
- Collapsed faces
- Insufficient refinement through thickness
- Internal long edges
- Tetras with extremely large volume
- Tetras with high aspect ratio
- Tetras with extreme angle between faces

CAUTION: Running the Mesh Repair Wizard on 3D entities may alter the location of surface nodes. On multi-component assemblies, this may change the element matching between components and cause unwanted gaps.

Validate the repairs

Whenever an automatic fix is applied to the mesh, the repairs must be validated to ensure that no new problems were created.

After you have completed the mesh clean up, it is recommended that you run the Mesh Statistics report again, and if necessary, run the Mesh Repair Wizard again to confirm that all potential problems have been resolved.

Mesh Repair Wizard

The mesh repair wizard provides a simple method of repairing the majority of mesh errors.

Using the Mesh Repair Wizard

1. Click **Mesh tab > Mesh Repair panel > Mesh Repair Wizard**.
2. Click **Next** and step through each of the mesh diagnostics.

3 If errors are reported, click **Fix** and the Wizard will attempt to automatically repair the errors.

Mesh Repair Wizard

The **Mesh Repair Wizard** allows you to repair several types of mesh defect. You can make changes on each page, or skip the pages that do not concern you.

To access this panel, click (**Mesh tab > Mesh Repair panel > Mesh Repair Wizard**).

Mesh Repair Wizard

As you open each page of the Wizard, the model is scanned for defects. You can see the results from each scan and then decide what to do next.

Click **Fix** to repair the detected mesh defects, or **Next** to fix any defects found by the Wizard for the current page, and move forward to the next page.

The Wizard addresses the errors listed below.

- Stitch free edges
- Fill hole
- Overhang
- Degenerate elements
- Flip normal
- Fix overlap
- Collapsed faces
- Aspect ratio

Mesh Repair Wizard (3D)

As you open each page of the Wizard, the model is scanned for defects. You can see the results from each scan and then decide what to do next.

Click **Fix** to repair the detected mesh defects, or **Next** to fix any defects found by the Wizard for the current page, and move forward to the next page.

The Wizard addresses the errors listed below.

- Inverted tetra
- Collapsed faces
- Insufficient refinement through thickness
- Internal long edges
- Tetra with extremely large volume
- High aspect ratio
- Tetra with extreme angles between faces

Nodes repair

Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Use the **Align Nodes** command to reposition nodes to lie on a straight line. You must first select the two nodes that define the straight line (alignment edge), and then select the nodes to move.

Use the **Global Merge** command to search the entire mesh to find and merge all nodes that are within a specified distance from each other. This distance is the merge tolerance. The Global Merge command acts on the entire mesh, whereas the Merge command acts only on the nodes you specify.

Use the **Insert Nodes** command to split an existing triangle or tetrahedral element into smaller elements by inserting a new node either midway along an edge of a triangle, or in the center of a triangle or tetrahedron.

Use the **Match Nodes** command to project nodes from one surface of a Dual Domain mesh into the selected triangles on the other surface of the Dual Domain mesh to re-establish a good mesh match after manually fixing the mesh.

Use the **Merge Nodes** command to merge one or more nodes to a single node. The Merge command acts only on the nodes that you specify, whereas the Global Merge command searches the entire mesh and merges all nodes within the specified distance from each other.

Use the **Move Nodes** command to either move one or more nodes to an absolute location, or, move one or more nodes by a relative offset.

Use the **Purge Nodes** command to remove all nodes that are not connected to elements.

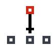
Use the **Smooth Nodes** command to create element edge lengths of a similar size for a more uniform mesh.

Nodes repair

Manipulating nodes is a method of modifying the mesh.

Aligning nodes

The Align Nodes dialog allows you to move all specified nodes to an established line.

1 Click  **Mesh tab > Mesh Repair panel > Align Nodes**.
2 Enter the location of the first Alignment node or click on the node on the model.
3 Enter the location of the second Alignment node or click on the node on the model.
4 Enter node(s) to be moved to the specified line, or hold down the CTRL key and click on the model to select multiple nodes.
5 Click **Apply**.

   All specified nodes will be moved to follow the defined line.

Merging coincident nodes automatically

The Global Merge dialog will search the model for nodes within a certain distance and merge where possible.

1 Click  **Mesh tab > Mesh Repair panel > Global Merge**.
2 Enter a **Merge tolerance** value (mm).

   Nodes closer than the specified distance (tolerance) will be merged.

3 Click **Apply**.

   Nodes closer than the specified tolerance are searched for and merged.

Splitting elements by inserting nodes

The **Insert Nodes** command in the Mesh Tools dialog is used to split an existing triangle or tetrahedral element into smaller elements by inserting a new node either midway along an edge of a triangle, or in the center of a triangle or tetrahedron.

1 Click  **Mesh tab > Mesh Repair panel > Insert Nodes**.
2 Select whether you want the new node to be inserted midway along a triangle edge, or in the center of a triangle or tetrahedron.

   Depending on your selection, either two or three of the Node 1/2/3 input boxes are active, or the Tetra to split input box is active.

3 If the mesh nodes are not visible, turn on the relevant layer(s) in the Layers pane.
4 Specify the required number of nodes either by selecting them in the model pane, or by entering their numbers manually, e.g. N123, in the text boxes.

NOTE: the Insert Nodes command will only work if the selected nodes all belong to the same element.

5 Click **Apply** to insert a new node at the specified location.

New element edges are automatically created to split the existing element into smaller elements.

Merging coincident nodes manually

If there are problems with your meshed model, such as high aspect ratio elements, mesh connectivity problems or free edges, manually merging nodes is a useful way of repairing the model.

1 Click **Mesh tab > Mesh Repair panel > Merge Nodes**.

2 In the **Nodes to merge from** box, enter the number of the node to be merged, or select a node on the model.

3 Click **Apply**.

Moving nodes

The Move Nodes command in the Mesh Repair panel relocates an existing node or nodes to a new position on the model.

1 Click **Mesh tab > Mesh Repair panel > Move Nodes**.

2 Enter the number(s) of the node(s) to be moved, or select the node(s) on the model.

3 Specify the new location by entering a set of coordinates in the **Location** or **Offset** text box.

Location	Applies to Absolute positioning that is relative to the origin.
Offset	Applies to Relative positioning that is relative to the current position of the node(s).

TIP: You can also move a node by holding down the mouse button and dragging the node to the required location.

4 Click **Apply**.

The selected node(s) are moved to the specified position.

NOTE: If a node is on an edge, it can only be moved along that edge. If a node is on a corner, it cannot be moved.

Deleting unused nodes

The Purge Nodes dialog removes any nodes that are not connected to an element.

1 Click **Mesh tab > Mesh Repair panel > Purge Nodes**.
2 Click **Apply**.

Optimizing node distribution in mesh

The Smooth Nodes command in the Mesh Tools dialog moves nodes as required to generate a smoother mesh.

1 Click **Mesh tab > Mesh Repair panel > Smooth Nodes**.
2 Select the node(s) which need to be smoothed, by either entering the node number(s) into the **Select** drop-down list or selecting the nodes on the model.
3 Select **Preserve feature edges** if you do not want to move nodes on feature edges.
4 Click **Apply**.

   The node will be moved to create a smoother element line.

Using mesh tools to improve mesh matching

1 If necessary, select the required layer(s) in the **Layers** pane to display all nodes in the model.
2 Click **Mesh tab > Mesh Repair panel > Match Nodes**.

   The Match Nodes tool allows you to pick a selection of nodes and adjust their position by associating them with corresponding elements on the opposite side of the part.
3 Click **Mesh tab > Selection panel > Select Facing Entities**.

   This ensures that when you select nodes or elements on one side of the part, no nodes or elements on the opposite side are selected.
4 Select a region of nodes where there are one or more unmatched elements that are not on an edge.

   Unmatched elements are displayed in red on the Dual Domain Mesh Match Diagnostic plot. The selected nodes are listed in the **Nodes to project into mesh:** text box.
5 Rotate the part and select a group of triangles on the opposite side of the part to the selected nodes.

   The selected elements are listed in the **Triangles for nodes to project to:** text box.

6 Click **Apply** to perform the matching.
   The Mesh Match diagnostic plot updates automatically.
7 Repeat the above steps for other areas of the part you would like to correct.

TIP: To improve mesh matching, you can also use the **Remesh Area** mesh editing tool. (**Remesh Area**) allows you to select a group of unmatched elements and remesh them, possibly improving the mesh match. It is important to select the elements on either side of the thickness when performing the remesh operation.

Redefining curve by moving an end point

You can redefine a curve by moving one of its endpoints.

1 Click on the model to select the node that corresponds to the end point of the curve you want to move.
2 Reposition the selected node using one of the **Move/Copy** tools.
3 Click on the model to select the original curve and delete it.

TIP: Before you delete the curve, identify the node that corresponds to the end point you did not move.

---

4 Create a new curve using one of the **Create Curves** tools to connect the end point nodes.

Nodes repair

The **Mesh Repair** panel provides a collection of tools for automatically or manually repairing mesh defects identified using the **Mesh Diagnostics** displays in the **Mesh** tab.

Align Nodes tool

Use the **Align Nodes** command to reposition nodes to lie on a straight line. You must first select the two nodes that define the straight line (alignment edge), and then select the nodes to move.

To access this panel, click (**Mesh tab > Mesh Repair panel > Align Nodes**).

In the **Input parameters** section of the dialog, select two nodes to identify a line onto which other selected nodes are to be aligned. Then select the nodes you would like to align.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Merge Nodes tool

Use the **Merge Nodes** command to merge one or more nodes to a single node. The Merge command acts only on the nodes that you specify, whereas the Global Merge command searches the entire mesh and merges all nodes within the specified distance from each other.

To access this panel, click (**Mesh tab > Mesh Repair panel > Merge Nodes**).

In the **Input parameters** section of the dialog, select the nodes you want to merge and the node you would like to merge them to.

Decide whether you want merge nodes along an element edge only.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Insert Nodes tool

The **Insert Nodes** command is used to split an existing triangle or tetrahedral element into smaller elements by inserting a new node either midway along an edge of a triangle, or in the center of a triangle or tetrahedron.

To use this tool, click (**Mesh tab > Mesh Repair panel > Insert Nodes**).

In the **Input parameters** section of the dialog, decide where you would like the new node created.

Select the nodes between which the new node is to be created.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Move Nodes tool

Use the **Move Nodes** command to either move one or more nodes to an absolute location, or, move one or more nodes by a relative offset.

To use this tool, click (**Mesh tab > Mesh Repair panel > Move Nodes**).

In the **Input parameters** section of the dialog, select the nodes you want to move.

In the **Location** section of the dialog, indicate where you would llike to move the selected nodes.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

TIP: You can drag a triangle node by clicking on the node and dragging it to the new position. You can also drag beam nodes in the same manner, however, selecting a node on a beam element will activate a modeling plane.

Purge Nodes tool

Use the **Purge Nodes** command to remove all nodes that are not connected to elements.

To use this tool, click (**Mesh tab > Mesh Repair panel > Purge Nodes**).

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering. No other set-up parameters are required for this tool.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Match Nodes tool

Use the **Match Nodes** command to project nodes from one surface of a Dual Domain mesh into the selected triangles on the other surface of the Dual Domain mesh to re-establish a good mesh match after manually fixing the mesh.

To use this tool, click (**Mesh tab > Mesh Repair panel > Match Nodes**).

In the **Input parameters** section of the dialog, select the nodes you want to project and the triangles you want to project to.

Decide whether you want the new nodes placed in a new layer.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Global Merge tool

Use the **Global Merge** command to search the entire mesh to find and merge all nodes that are within a specified distance from each other. This distance is the merge tolerance. The Global Merge command acts on the entire mesh, whereas the Merge command acts only on the nodes you specify.

To use this tool, click (**Mesh tab > Mesh Repair panel > Global Merge**).

In the **Input parameters** section of the dialog, specify the merge tolerance. Nodes closer together than this tolerance will be merged together.

Decide whether you want to merge nodes along an element edge only. This option should not be selected when using the **Global merge** tool to fix free edges.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Smooth Nodes tool

Use the **Smooth Nodes** command to create element edge lengths of a similar size for a more uniform mesh.

To use this tool, click (**Mesh > Mesh Tools > Smooth Nodes**).

In the **Input parameters** section of the dialog, select the nodes you want to smooth.

Choose whether you would like to preserve feature edges.

In the **Selection option** section of the dialog, indicate whether you would like to use filtering and if so, the type of filtering.

NOTE: Many of the mesh repair tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Edges

Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

The **Stitch Free Edges** tool is used to merge together a pair of nodes that are within a certain distance from each other, and that form a free (unconnected) edge with a similar pair of nodes. The distance between the nodes to be merged is the tolerance, and the default tolerance is 0.1 mm. You can specify a tolerance of your choice if you want.

Use the **Swap Edge** command to exchange the edges of two adjoining mesh elements. The **Swap Edge** command works on triangular elements, however, it cannot be used on tetrahedral meshes.

Edges

Edges can be adjusted to improve the quality of the meshing.

Stitching free edges in the mesh

The **Stitch Free Edges** tool merges nodes that are part of a free edge, so that all edges are connected properly.

1. Click **Mesh tab > Mesh Repair panel > Stitch Free Edges**.
2. Select the nodes that are part of the free edges you want to stitch together.

TIP: In most cases, selecting all the nodes in the model works well. Only the nodes that are part of a free edge will be merged.

---

3. Choose the tolerance you want to use. The tolerance is the distance between the nodes that will be merged.

   The default tolerance is defined in the software and is 0.1 mm. You can specify an alternative tolerance if you wish.
4. Click **Apply**.

   Autodesk Moldflow Insight will automatically connect the selected nodes that define free edges.

Repairing edges with the Swap Edges tool

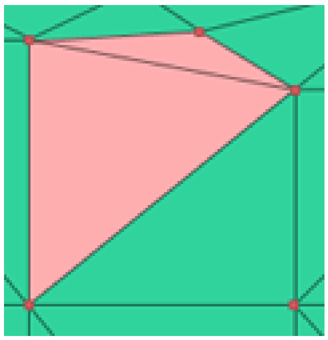
1. Click **Mesh tab > Mesh Repair panel > Swap Edges**.
2. Enter a value in the **Select first triangle** box, or select the appropriate element on the model.

3 Enter a value in the **Select second triangle** box, or select the appropriate element on the model.

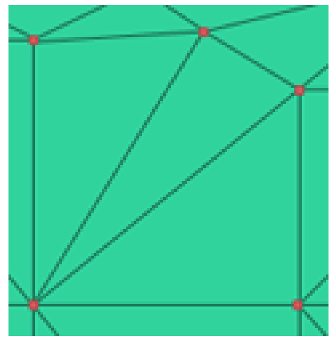
4 Select **Allow remesh of feature edges** if you want to remesh feature edges.

5 Click **Apply**.

Autodesk Moldflow Insight will change the direction of the shared edge between the two elements.



Before swapping edges



After swapping edges

Edges

The **Stitch Free Edges** tool is used to merge together a pair of nodes that are within a certain distance from each other, and that form a free (unconnected) edge with a similar pair of nodes.

To use this tool, click (**Mesh tab > Mesh Repair panel > Stitch Free Edges**).

Mesh Tools—Stitch Free Edges tool

In the **Input parameters** section of the dialog, select the nodes which define the free edge to stitch. Specify the stitch tolerance; nodes closer than this will be merged.

In the **Selection option** section of the dialog, choose whether you want to use a filter.

NOTE: Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

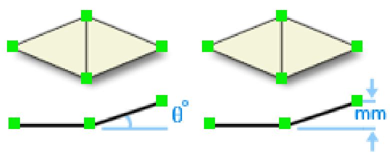
The distance between the nodes to be merged is the tolerance, and the default tolerance is 0.1 mm. You can specify a tolerance of your choice if you want.

Mesh regions

Regions are areas or selections of connected entities that may include both planar and non-planar surfaces.

Working with regions is often easier than working with a mesh because it is possible to group mesh elements together logically and to ensure that individual mesh elements are not omitted from receiving a property.

Regions are always planar. Since mesh elements may deviate slightly from being coplanar, you can specify a tolerance within which adjacent mesh elements will be considered part of the same plane, and hence become part of the same region. There are two types of tolerance to choose from when creating regions: angular or planar. Angular tolerance specifies the maximum allowable angle between mesh elements in degrees (see image on left, below). Planar tolerance specifies the maximum allowable distance from the same plane between mesh elements in mm (see image on right, below). Angular tolerances may be used when your part has large curved surfaces. Planar tolerances may be used when you want to preserve the shape of your part.



Mesh regions

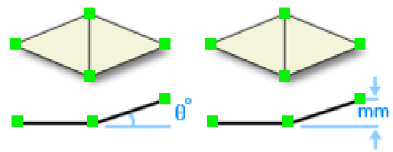
The Create Regions tool allows you to define areas on the mesh and then assign these regions default properties.

Creating mesh regions

The Create Regions dialog allows you to define areas on the mesh and then assign default properties to these regions. It is often easier to work with regions rather than a mesh, because when using a region it is possible to group mesh elements together logically and ensure that individual mesh elements are not omitted from receiving a property.

Regions are always planar. Since mesh elements may deviate slightly from being coplanar, you can specify a tolerance within which adjacent mesh elements will be considered part of the same plane. By doing this, you can ensure that they become part of the same region.

There are two types of tolerance to choose from: angular or planar. The angular tolerance enables you to specify the maximum allowable angle between mesh elements in degrees as shown in the image below left. The Planar tolerance enables you to specify the maximum allowable distance from the same plane between mesh elements in mm, as shown in the image below, right. Angular tolerances are often used for parts with large curved surfaces. Planar tolerances are used when you want to preserve the shape of your part.



Create Regions from Mesh/STL produces one region that includes all the mesh elements.

1. Click **Geometry tab > Create panel > Regions > Region From Mesh/STL**.
2. Click either **Planar** or **Angular** in the **Tolerance** area and enter a value.
3. Click **STL** or **Mesh** in the **Create from** area, depending on the type of model used.
4. **Optional:** Click **Browse** in the **Selection option** group and select a different property type for the region to be created.
5. Click **Apply**.
   Regions are created on the model as specified in step 4.

Mesh regions

The **Region from Mesh/STL** tool allows you to create regions for easier manipulation of a model.

To access this tool, click (**Geometry tab > Create panel > Region from Mesh/STL**).

Regions from Mesh/STL tool

In the **Input parameters** section of the dialog, choose whether to set a planar or angular tolerance and whether to create the regions from an STL format or a mesh.

In the **Selection option** section of the dialog, choose which part surface properties you would like to assign to the region.

Use the **Regions from Mesh/STL** command to consolidate all elements from a Midplane or Dual-Domain mesh, or from a model's STL representation, into a single editable region.

Delete entities

Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

Use the **Delete Entities** command to remove from the mesh all entities that you select with the mouse cursor.

TIP: If you manually enter the entity to delete, you must also enter the entity ID. For example, if you want to delete node 400, enter **N400** into the box and click **Apply**.

NOTE: Body faces or bodies of a CAD assembly cannot be deleted independently.

Delete entities

The Delete Entities dialog allows you to remove entities from the model.

Deleting entities

1 Click **Mesh tab > Mesh Repair panel > Delete Entities**.

2 Enter node numbers or select entities on the model.

3 Click **Apply**.

NOTE: Body faces or bodies of a CAD assembly cannot be deleted independently.

Delete entities

The **Delete Entities** command is used to remove from the mesh all entities that you select with the mouse cursor.

To use this tool, click (**Mesh tab > Mesh Repair panel > Delete Entities**).

Mesh Tools—Delete Entities tool

In the **Input parameters** section of the dialog, select the entities you would like to delete.

In the **Selection options** section of the dialog, decide whether you would like to use filtering.

NOTE: Many of the mesh tools involve selecting or manipulating nodes. Ensure that the nodes in the part are visible and selectable by activating the required layers in the **Layers** pane.

TIP: If you manually enter the entity to delete, you must also enter the entity ID. For example, if you want to delete node 400, enter **N400** into the box and click **Apply**.

NOTE: Body faces or bodies of a CAD assembly cannot be deleted independently.