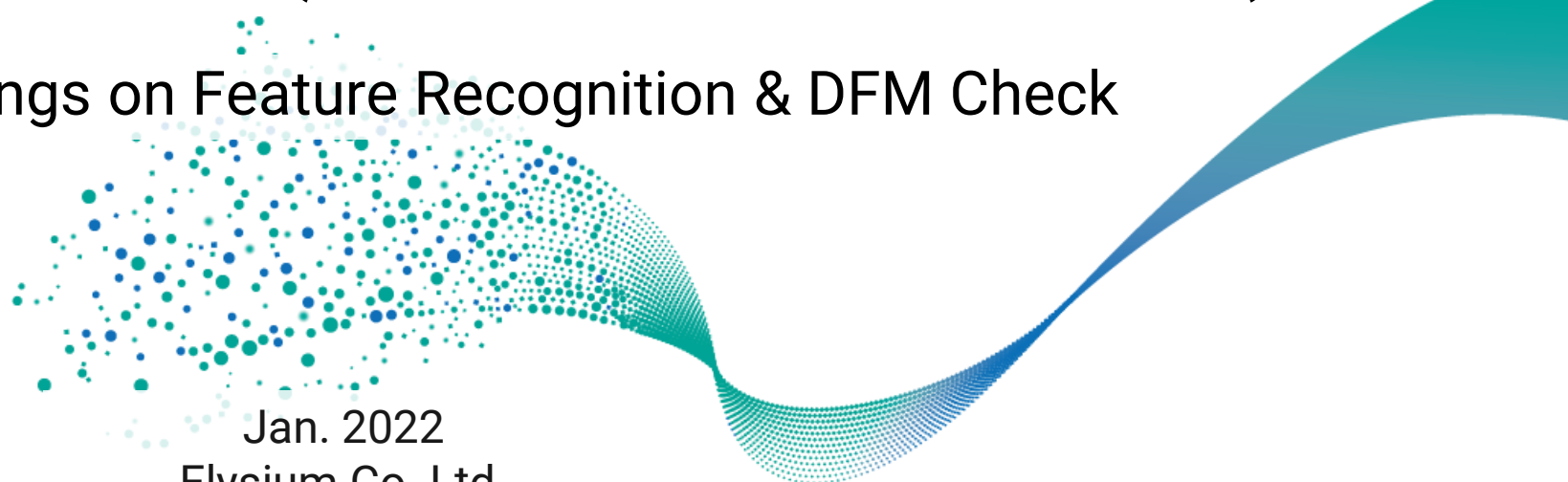




Check Criteria Guide (For Sheet Metal Model)

Parameter Settings on Feature Recognition & DFM Check

A decorative graphic on the right side of the slide, featuring a cluster of blue and green dots of varying sizes that form a wave-like shape, transitioning into a solid blue wave that curves upwards.

Jan. 2022
Elysium Co. Ltd.

Index

Check Criteria

1. Check Model (Pre-check)
2. Hole
3. Burring
4. Bend
5. Groove / Thin area
6. Dowel
7. Half-punching
8. Drawing

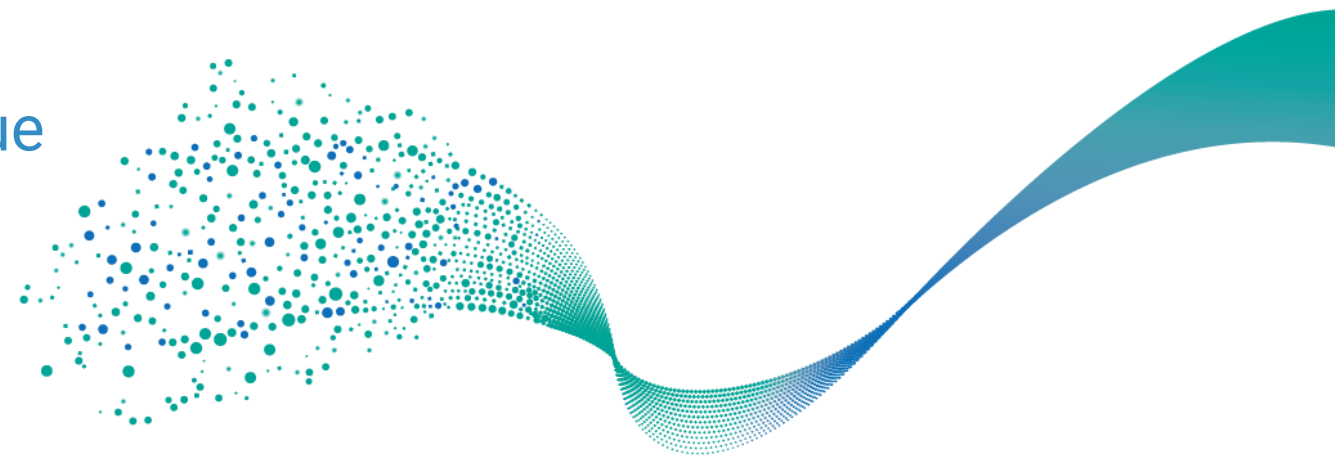
Feature Recognition

- Sheet metal thickness & Standard thickness
- Front face / Back face / Side face
- Burring
- Punching hole
- Half-punching
- Dowel
- Drawing
- Simple bend
- Hem bend

Appendix

Check Criteria

Parameters and the Judgment Value



Index: Check Criteria List – 1

1. Check Model (Pre-check)

- [1.1 Check model: Sheet metal thickness](#)
- [1.2 Check model: Inconsistent sheet metal thickness](#)
- [1.3 Check model: Outward bend without fillet \(Edge\)](#)
- [1.4 Check model: Outward bend with invalid fillet \(Sheet metal thickness\)](#)

2. Hole

- [2.1 Round hole diameter](#)
- [2.2 Hole: Distance to hole or part end face](#)
- [2.3 Hole: Distance to hole or part end face \(Parallel\)](#)

3. Burring

- [3.1 Burring diameter and height](#)
- [3.2 Burring: Distance to hole or part end face](#)
- [3.3 Burring: Distance to burring](#)

4. Bend

- [4.1 Bend height](#)
- [4.2 Z bend flange length](#)
- [4.3 U bend flange length](#)
- [4.4 Bend: Distance to hole](#)
- [4.5 Bend: Distance to hole \(Parallel\)](#)
- [4.6 Bend: Distance to burring](#)

5. Groove / Thin area

- [5.1 Groove width \(Thin area\)](#)
- [5.2 Thin area width](#)
- [5.3 Bend relief cut depth](#)
- [5.4 Bend relief cut width](#)

6. Dowel

- [6.1 Dowel outer diameter and height](#)
- [6.2 Dowel: Distance to hole or part end face](#)
- [6.3 Dowel: Distance to dowel](#)
- [6.4 Dowel: Distance to bend](#)
- [6.5 Dowel: Distance to burring](#)

Index: Check Criteria List – 2

7. Half-punching

[7.1 Half-punching height](#)

[7.2 Half-punching: Distance to hole or part end face](#)

[7.3 Half-punching: Distance to burring](#)

[7.4 Half-punching: Distance to half-punching](#)

[7.5 Half-punching: Distance to bend](#)

[7.6 Half-punching: Distance to dowel](#)

8. Drawing

[8.1 Drawing height](#)

[8.2 Drawing: Distance to hole or part end face](#)

[8.3 Drawing: Distance to burring](#)

[8.4 Drawing: Distance to bend](#)

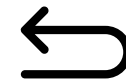
[8.5 Drawing: Distance to dowel](#)

[8.6 Drawing: Distance to half-punching](#)

[8.7 Drawing: Distance to drawing](#)



Check Model



1 Check model (Pre-check)

Description of Model Check

<Description>

This is to assess whether input CAD models satisfy the design requirements for sheet metal models so that the following DFM check will be conducted accurately and completely.

Please note that “1.1 Check model: Sheet metal thickness” is a prerequisite for other three check criteria, and Fail areas will be skipped in the following Check Model processes.

Please modify such areas back in the CAD system so that the sheet metal thickness will be measured throughout the model, and Check Model will be conducted appropriately.

Overview of Each Check Criteria and Process Flow

<Check Model Process Flows>

1.1 Sheet metal thickness

Calculates “Standard thickness”.
Following Check Model and DFM Check will be skipped on Fail areas.

1.2 Inconsistent sheet metal thickness

Detects areas where the sheet metal thickness is thicker than Standard Thickness x “Max. result to mode value ratio tol for sheet metal thickness”, or thinner than Standard Thickness x “Min. result to mode value ratio tol for sheet metal thickness”.
This does not affect the following DFM Check, but may be a modeling error, so please modify as required.

1.3 Outward bend without fillet (Edge)

Detects outward bend without a fillet.
Geometry recognition on “Simple bend” and “Hem bend” will fail on Fail areas, and following DFM Check will be skipped on Fail areas.
Please add a fillet to Fail areas to treat them as bend in the following DFM Check.

1.4 Outward bend with invalid fillet (Sheet metal thickness)

Detects outward bend with an invalid fillet (the sheet metal thickness does not match “Standard thickness”).
This does not affect the following DFM Check, but may be a modeling error, so please modify as required.



1.1 Check model: Sheet metal thickness

Description of Check Criterion & Possible Molding Defects

<Description>

Calculates the standard sheet metal thickness which will be used for the tolerances in DFM check. Also, detects areas where it failed to measure the sheet metal thickness. It means that Standard thickness was calculated excluding such areas, and DFM check will be skipped on such areas. Please modify such areas back in the CAD system, so that the sheet metal thickness will be measured throughout the model, and DFM check will be conducted appropriately.
* Please refer to [Sheet metal thickness & Standard thickness](#) for how to calculate the sheet metal thickness and the standard thickness.

1. How to Check & Prerequisites

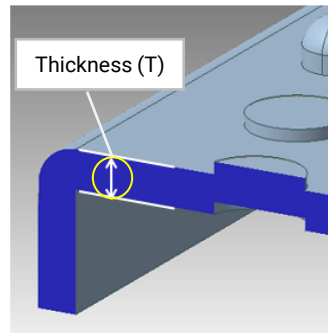
<How to Check>

Calculates Standard thickness of the input model.

Marks the area as Fail when it failed to measure the thickness because the face pair of Front face and Back face was not found.

<Prerequisites>

Geometry recognition is completed on "[Sheet metal thickness & Standard thickness](#)".



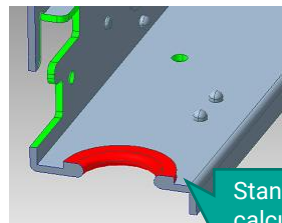
2. How to Judge & Default Value

<How to Judge>

Marks as Fail when it failed to recognize the geometry ([Front face / Back face / Side face](#)) to set face pairs of Front face and Back face. (=The sheet metal thickness was not measured.)

<Default Value>

- N/A

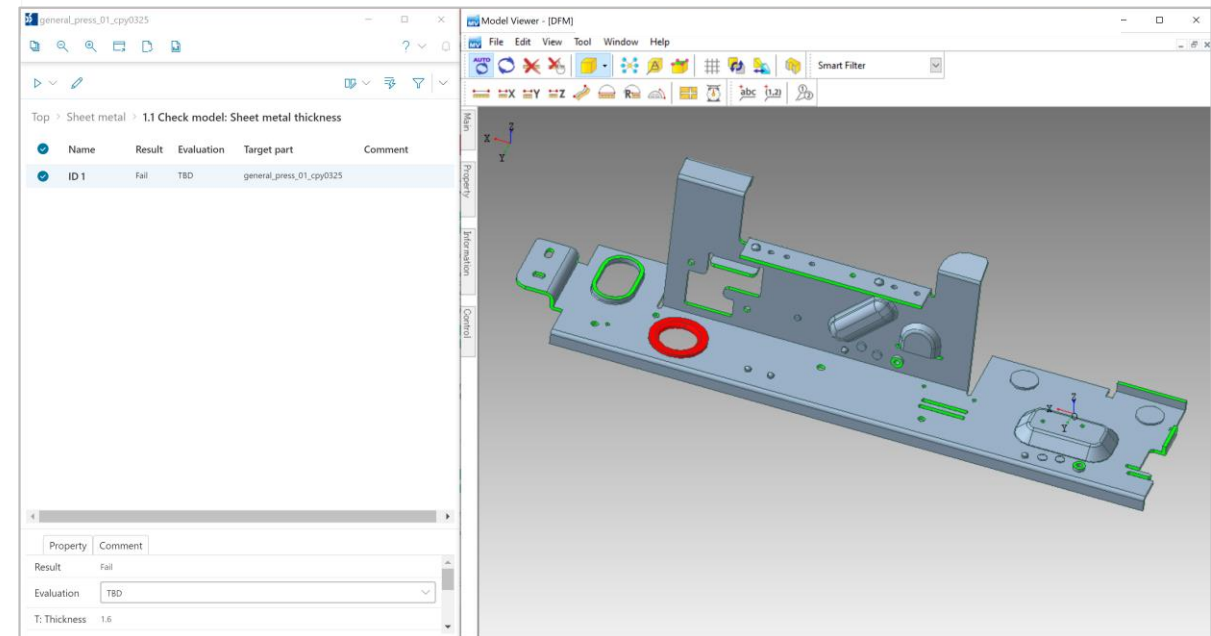


Standard thickness will not be calculated because the front side or back side of face cannot be recognized (Side face cannot be recognized)

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: Side faces of the input model Red: [Fail] Faces of areas where it failed to recognize Front face / Back face / Side face





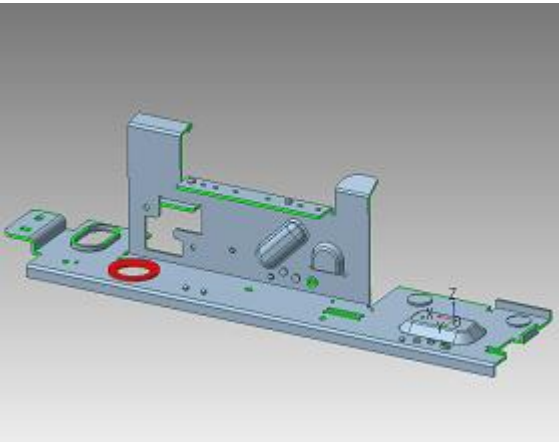
1.1 Check model: Sheet metal thickness

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

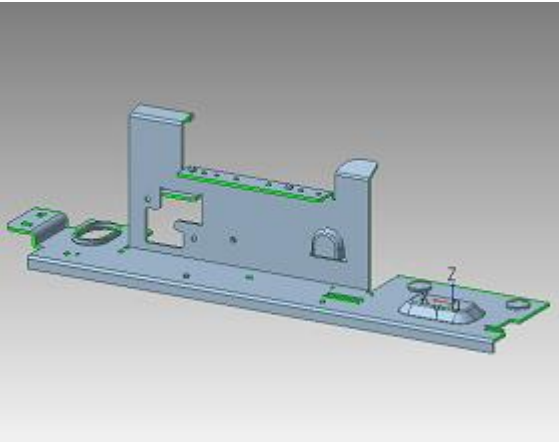
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: Side faces of the input modelRed: [Fail] Faces of areas where it failed to recognize Front face / Back face / Side face
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	N/A
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

Highlight Checked Area - Fail



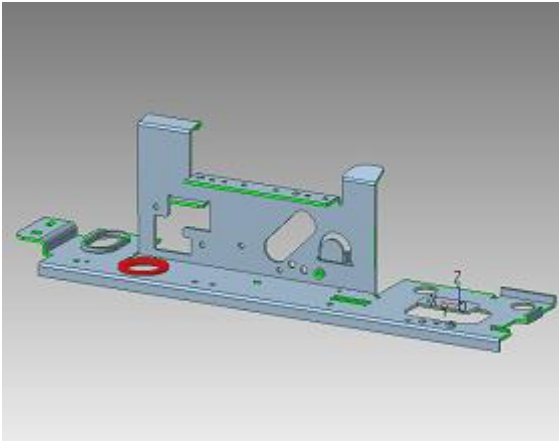
Highlight Checked Area - Pass



Section View

N/A

Hide Surroundings





1.2 Check model: Inconsistent sheet metal thickness

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the sheet metal thickness is thinner than Standard thickness multiplied by the ratio tolerance specified with "Min. result to mode value ratio tol for sheet metal thickness" parameter, or thicker than Standard thickness multiplied by the ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" parameter.

This does not affect the DFM check result, but may be a modeling error. So please confirm that it meets the designer's intent.

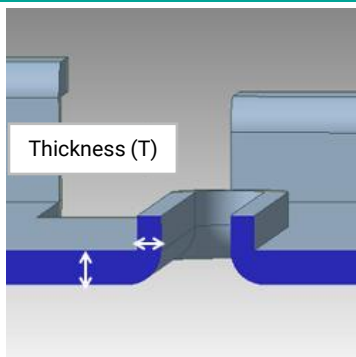
1. How to Check & Prerequisites

<How to Check>

Checks whether the sheet metal thickness is thicker / thinner than Standard thickness multiplied by the ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" / "Min. result to mode value ratio tol for sheet metal thickness" parameters.

<Prerequisites>

Geometry recognition is completed on "[Sheet metal thickness & Standard thickness](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the sheet metal thickness is thicker / thinner than Standard thickness multiplied by the ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" / "Min. result to mode value ratio tol for sheet metal thickness" parameters.

<Default Value>

- Standard thickness calculated with "[1.1 Check model: Sheet metal thickness](#)", and the thickness range specified with parameters

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

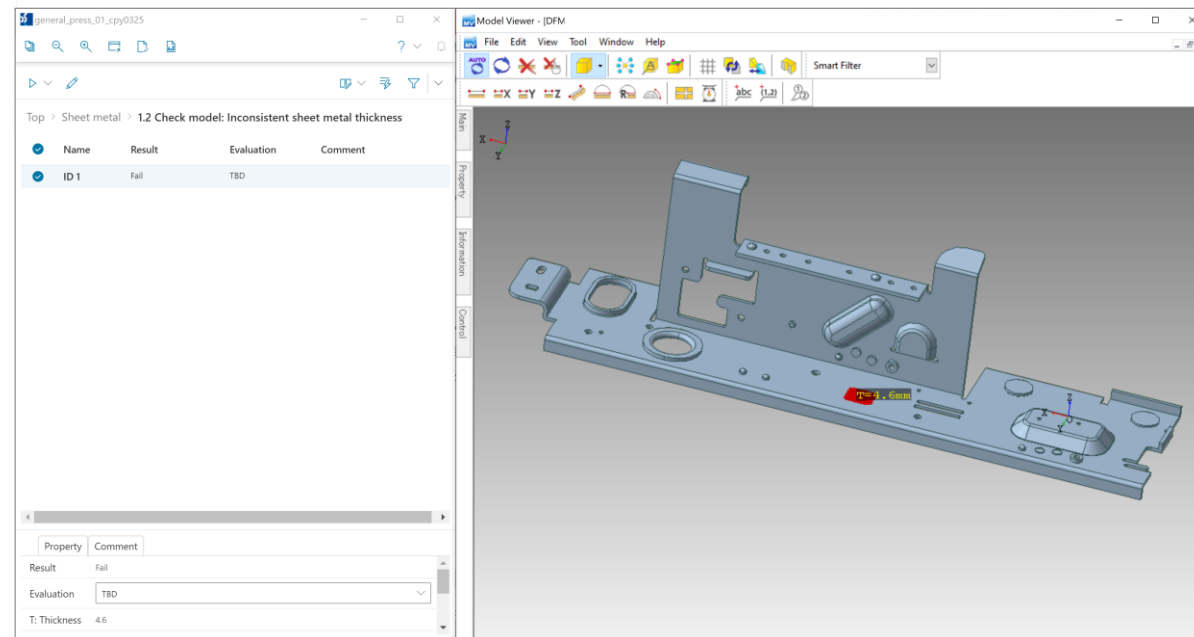
The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command

Description

Highlight Checked Area

- Red: [Fail] Area where the sheet metal thickness is thinner/thicker than the specified range.





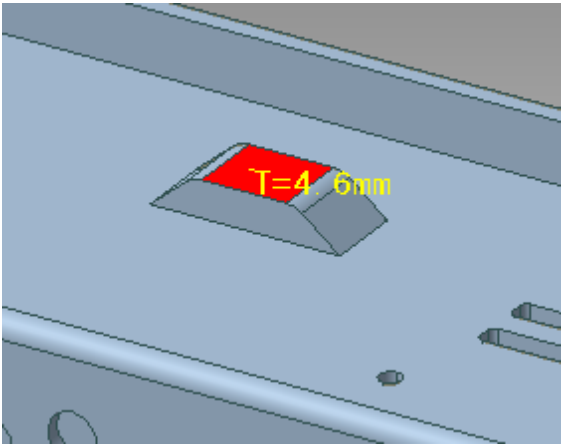
1.2 Check model: Inconsistent sheet metal thickness

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Area where the sheet metal thickness is thinner/thicker than the specified range.
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

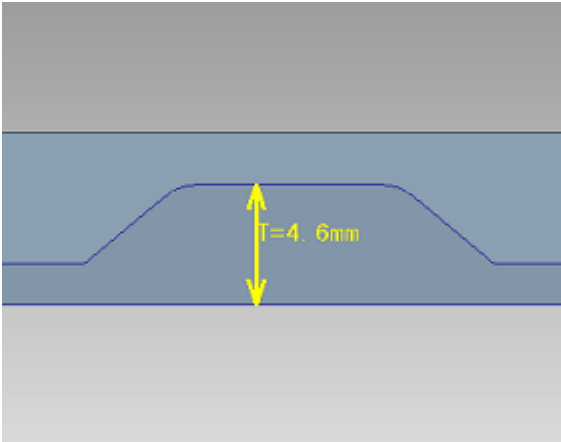
Highlight Checked Area - Fail



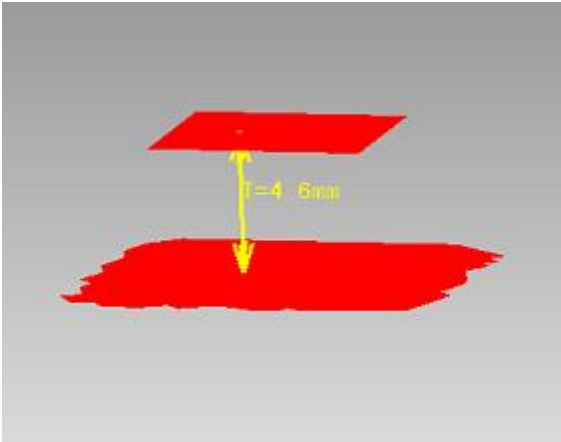
Highlight Checked Area - Pass

N/A

Section View



Hide Surroundings





1.3 Check model: Outward bend without fillet (Edge)

Description of Check Criterion & Possible Molding Defects

<Description>

Detects outward bends without fillet (=outward bend with a sharp corner).

Detected areas will not be recognized as Bend, and DFM check will be skipped on such areas. Please apply a fillet to such outward bends back in the CAD system, so that they will be recognized as the outward bend, and DFM check will be conducted appropriately.

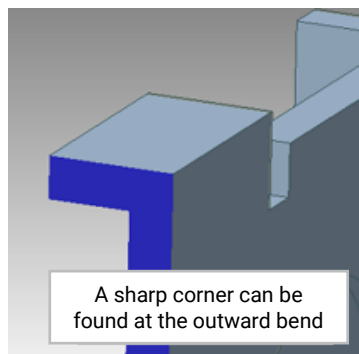
1. How to Check & Prerequisites

<How to Check>

Checks whether an outward bend has a fillet.

<Prerequisites>

Geometry recognition is completed on "[Sheet metal thickness & Standard thickness](#)", "[Simple bend](#)", and "[Hem bend](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the outward bend as Fail when it does not have a fillet.

<Default Value>

- N/A

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Red: [Fail] Outward bend without a fillet

The screenshot shows the Elysium software interface. On the left, the 'Inspector' window displays a table with the following data:

Name	Result	Evaluation	Comment
ID 1	Fail	TBD	

Below the table, the 'Property' tab is selected, showing 'Result: Fail', 'Evaluation: TBD', and 'A: Bend angle: 90'. The 'Model Viewer' window on the right shows a 3D model of a sheet metal part with a red highlight on the outward bend area, indicating a failure.



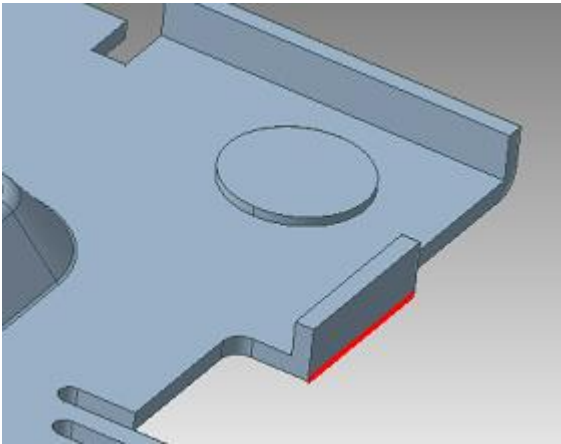
1.3 Check model: Outward bend without fillet (Edge)

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Outward bend without a fillet
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

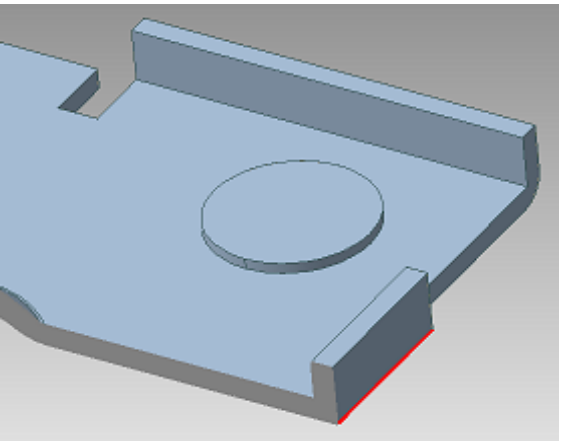
Highlight Checked Area - Fail



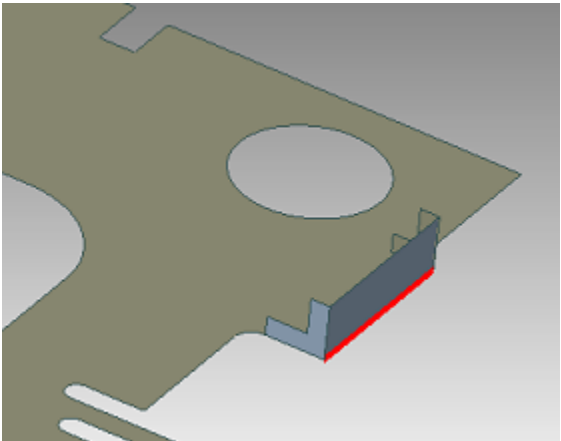
Highlight Checked Area - Pass

N/A

Section View



Hide Surroundings





1.4 Check model: Outward bend with invalid fillet (Sheet metal thickness)

Description of Check Criterion & Possible Molding Defects

<Description>

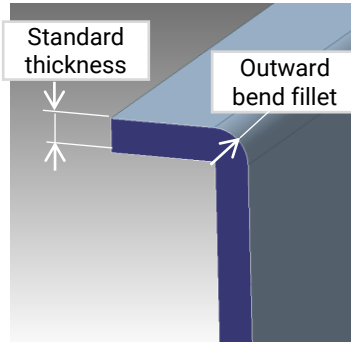
Detects outward bend with an invalid fillet (the sheet metal thickness does not match "Standard thickness").

This does not affect the DFM check result, but may be a modeling error. So please confirm that it meets the designer's intent.

1. How to Check & Prerequisites

<How to Check>

Checks whether the sheet metal thickness of the fillet on the outward bend is thicker / thinner than Standard thickness multiplied by the ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" / "Min. result to mode value ratio tol for sheet metal thickness" parameters.



<Prerequisites>

Geometry recognition is completed on "[Sheet metal thickness & Standard thickness](#)", "[Simple bend](#)", and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the fillet on the outward bend as Fail when the sheet metal thickness is outside the specified thickness range.

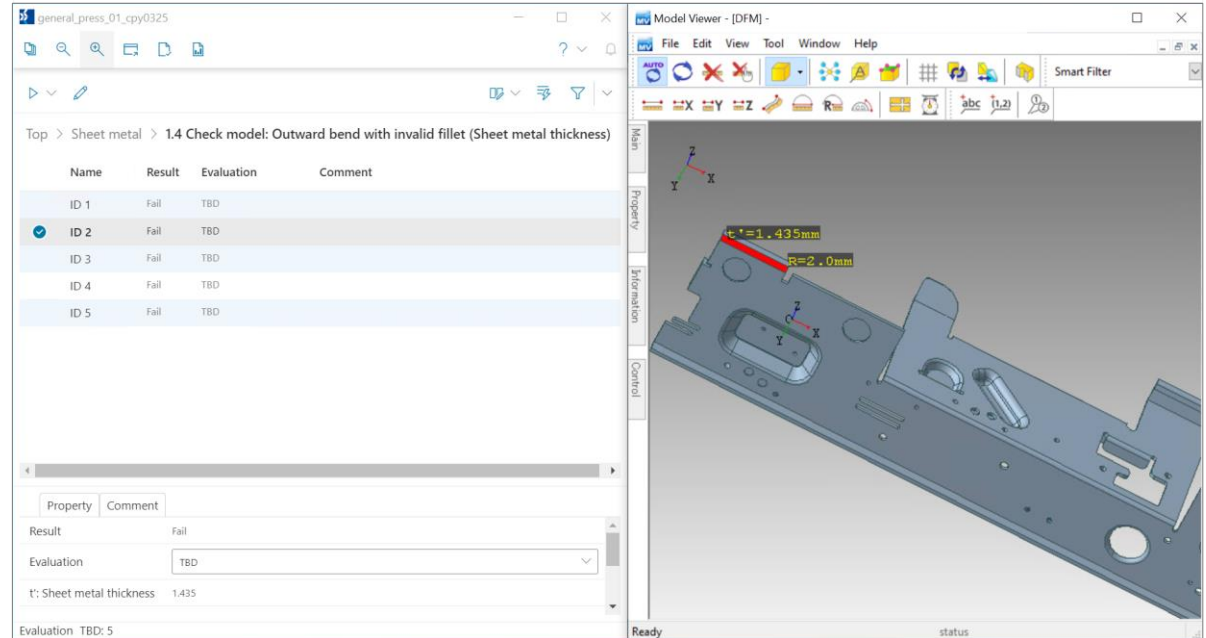
<Default Value>

- Standard thickness calculated with "[1.1 Check model: Sheet metal thickness](#)", and the thickness range specified with parameters

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Red: [Fail] Outward bend without a fillet whose thickness is outside the range



1.4 Check model: Outward bend with invalid fillet (Sheet metal thickness)

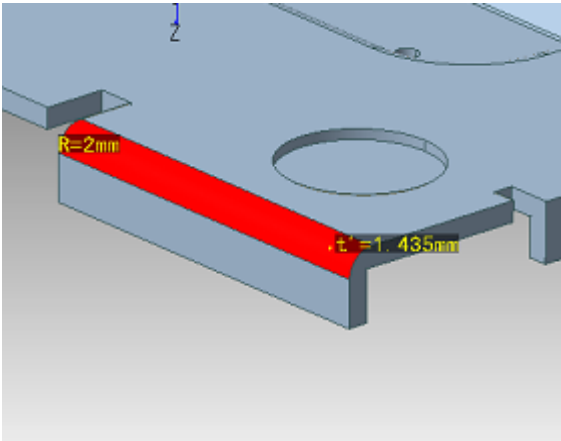


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Outward bend without a fillet whose thickness is outside the range
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

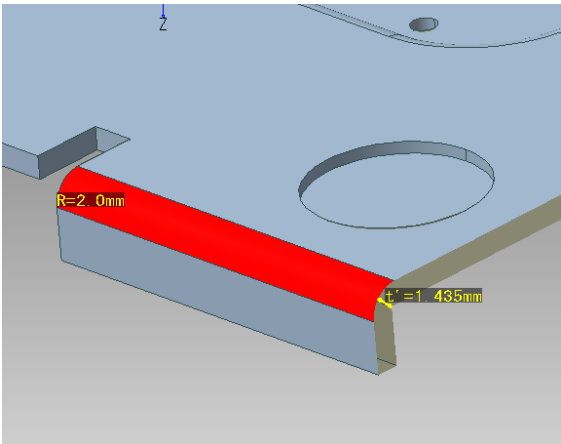
Highlight Checked Area - Fail



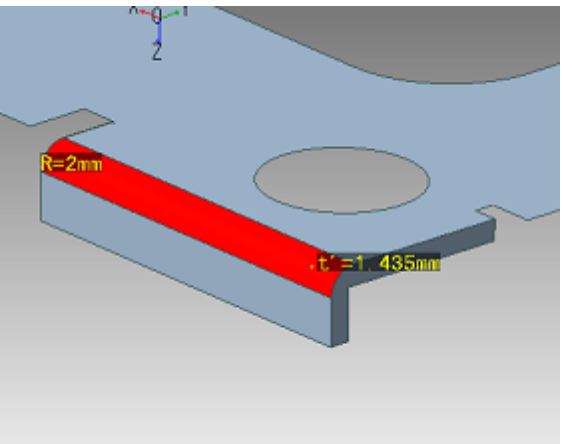
Highlight Checked Area - Pass

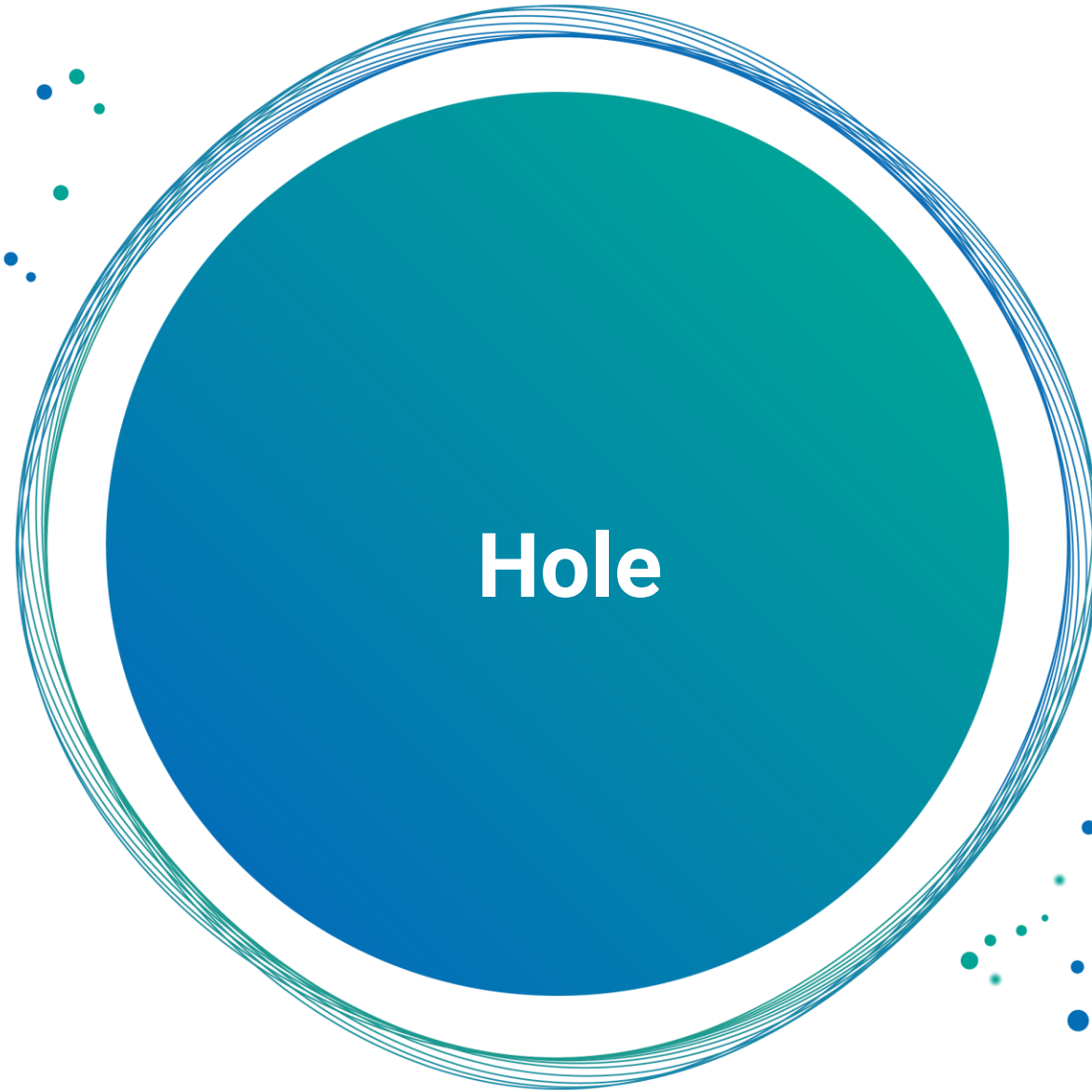
N/A

Section View



Hide Surroundings





2.1 Round hole diameter

Description of Check Criterion & Possible Molding Defects

<Description>

Detects round holes whose diameters are smaller than the specified tolerance.

<Possible Molding Defects>

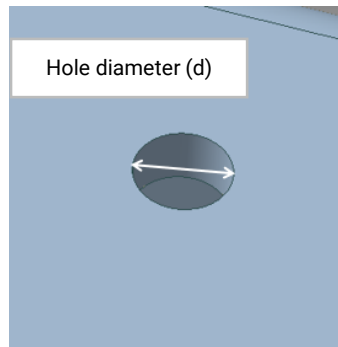


It may cause damage on the mold to make a small hole.
The minimum diameter tolerance differs depending on the material.

1. How to Check & Prerequisites

<How to Check>

Checks whether the diameter (d) of recognized round hole is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Punching hole](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the round hole as Fail when its diameter is smaller than the tolerance.

<Default Value>

- "Check round hole diameter | Min. diameter tol" parameter
PunchHoleDiam.Min=2.0 (Unit: mm)

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

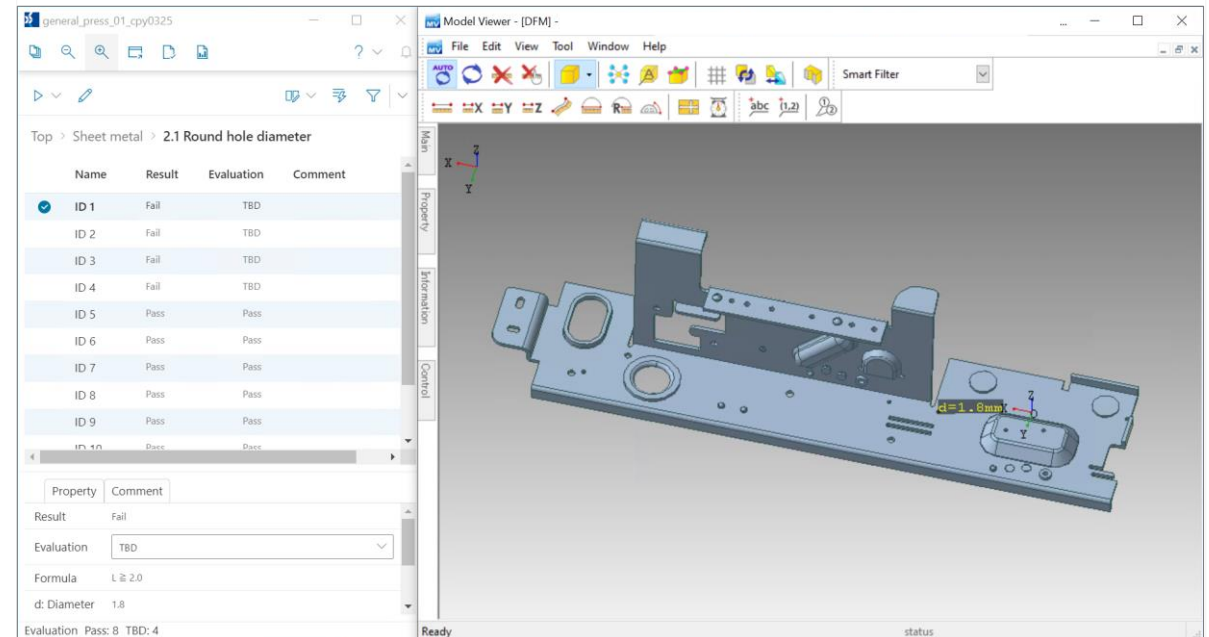
The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command

Description

Highlight Checked Area

- Green: [Pass] Round holes whose diameters are greater than the tolerance
- Red: [Fail] Round holes whose diameters are smaller than the tolerance



2.1 Round hole diameter

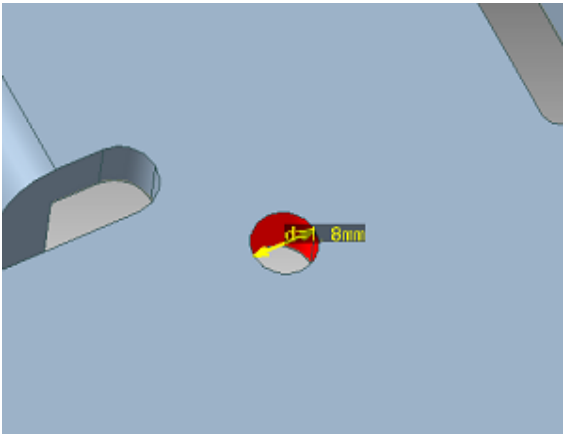


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Round holes whose diameters are greater than the toleranceRed: [Fail] Round holes whose diameters are smaller than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

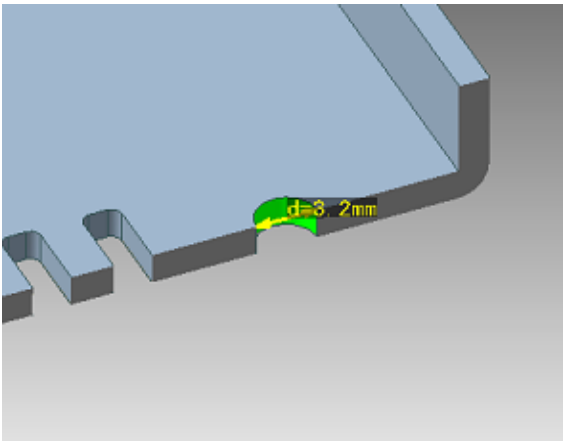
Highlight Checked Area - Fail



Highlight Checked Area - Pass



Section View



Hide Surroundings





2.2 Hole: Distance to hole or part end face

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a hole and another hole or a part end face is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause insufficient mold strength to make a hole too close to another hole or a part end face. It is recommended to make holes keeping an adequate distance from other holes and part end faces.

1. How to Check & Prerequisites

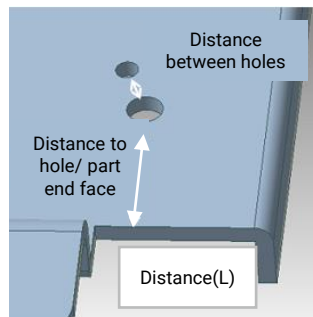
<How to Check>

Checks whether the distance (L) between a hole and another hole / part end face is greater than the tolerance.

Please note that measurement may be conducted at several points for a pair of a hole and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)" and "[Punching hole](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

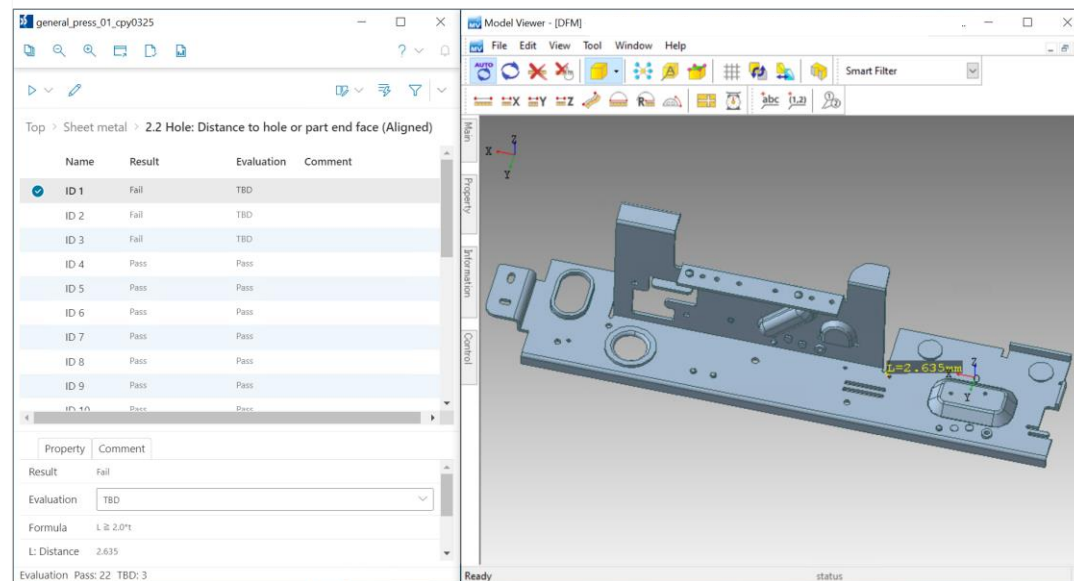
[A] When specifying by a fixed value with "Hole: Distance to hole or part end face | Min. distance tol by fixed value" parameter: HoleAndHoleDistance.Min=2.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Hole: Distance to hole or part end face | Min. distance tol by ratio to sheet metal thickness" parameter: HoleAndHoleDistance.Min.Ratio=2.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Hole and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Hole and hole / part end face pair which the distance in between is shorter than the tolerance





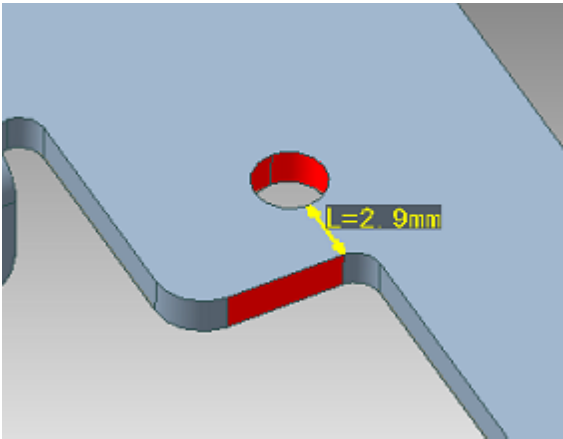
2.2 Hole: Distance to hole or part end face

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

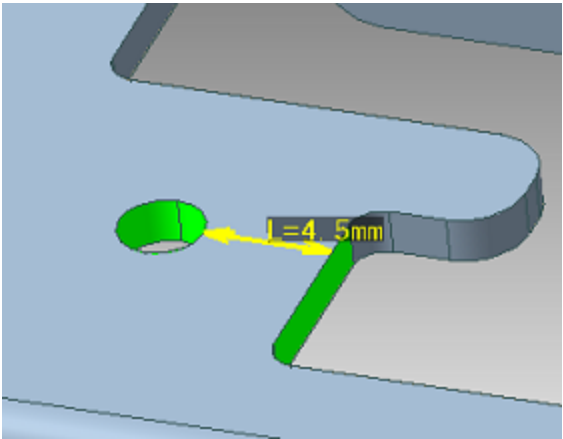
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Hole and hole / part end face pair which the distance in between is greater than the toleranceRed: [Fail] Hole and hole / part end face pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

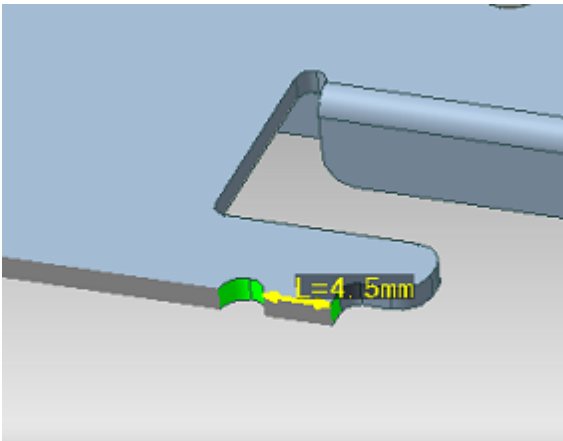
Highlight Checked Area - Fail



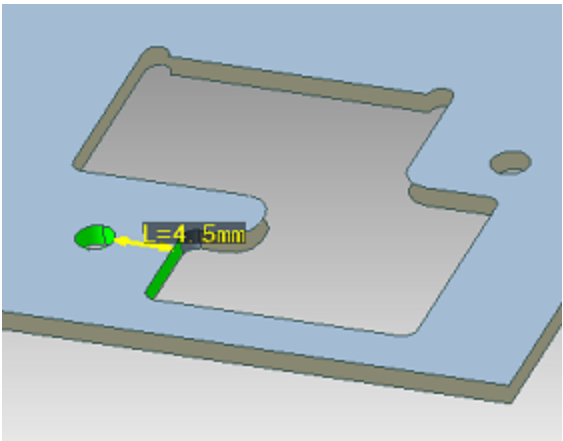
Highlight Checked Area - Pass



Section View



Hide Surroundings





2.3 Hole: Distance to hole or part end face (Parallel)

Description of Check Criterion & Possible Molding Defects

<Description>

When a hole has a straight-line edge, and another hole or part end face having a straight-line edge is parallel to it, detects areas where the distance between them is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause insufficient mold strength to make a hole too close to another hole or a part end face. It is recommended to make holes keeping an adequate distance from other holes and part end faces.

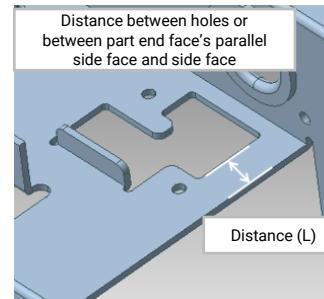
1. How to Check & Prerequisites

<How to Check>

For a hole and hole / part end face which have straight-line edges parallel to each other, checks whether the distance (L) between them is greater than the tolerance. Please note that measurement may be conducted at several points for a pair of a hole and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)" and "[Punching hole](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

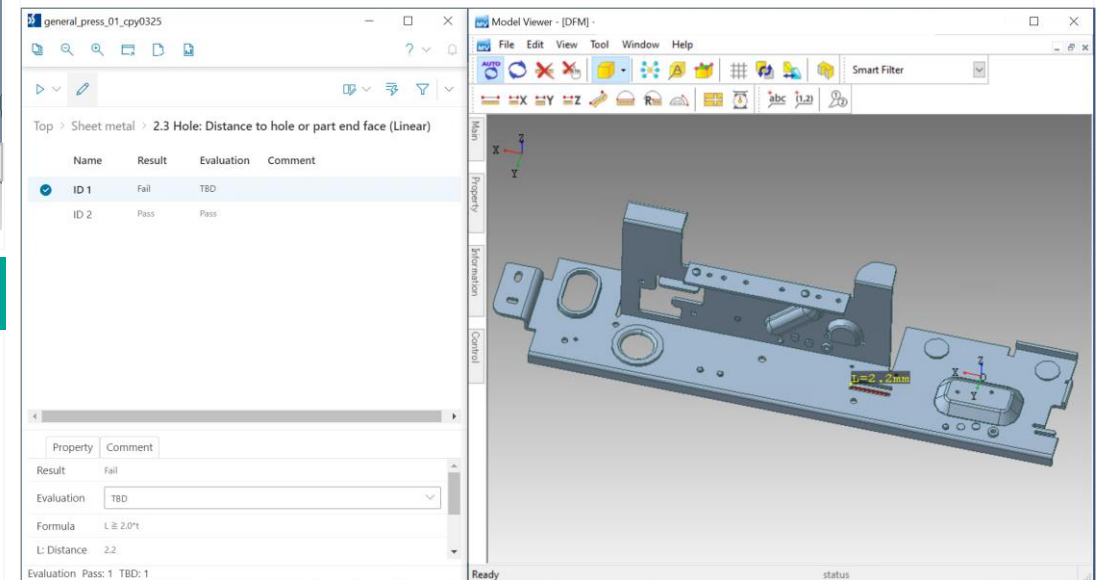
[A] When specifying by a fixed value with "Hole: Distance to hole or part end face (Parallel) | Min. distance tol by fixed value" parameter: HoleAndHoleDistance.Min=2.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Hole: Distance to hole or part end face (Parallel) | Min. distance tol by ratio to sheet metal thickness" parameter: HoleAndHoleDistance.Min.Ratio=2.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Hole and hole / part end face pair (Parallel) which the distance in between is greater than the tolerance Red: [Fail] Hole and hole / part end face pair (Parallel) which the distance in between is shorter than the tolerance





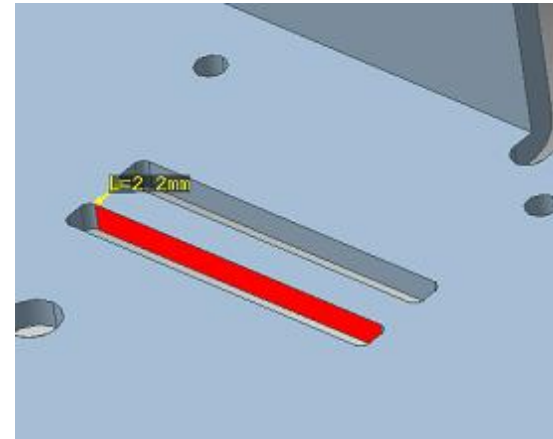
2.3 Hole: Distance to hole or part end face (Parallel)

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

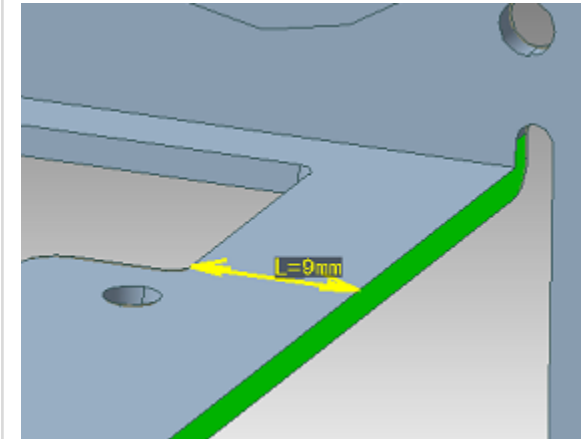
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Hole and hole / part end face pair (Parallel) which the distance in between is greater than the toleranceRed: [Fail] Hole and hole / part end face pair (Parallel) which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

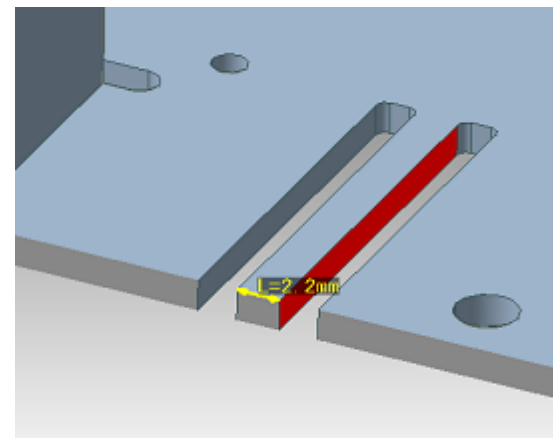
Highlight Checked Area - Fail



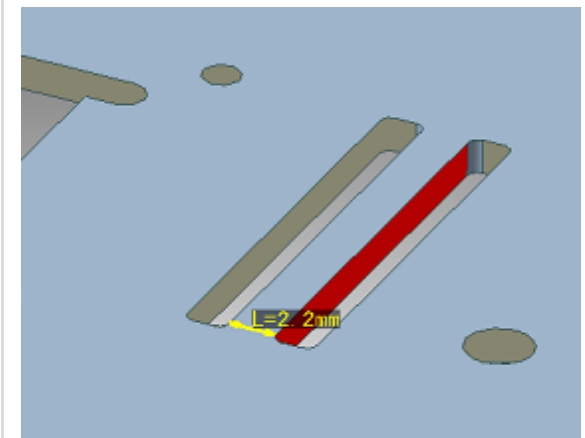
Highlight Checked Area - Pass



Section View



Hide Surroundings





Burring

3.1 Burring diameter and height

Description of Check Criterion & Possible Molding Defects

<Description>

Detects Burring which does not follow the specified standards (diameter and height).

<Possible Molding Defects>



It may cause insufficient thread length when the burring is not designed in conformity to the specified standards.

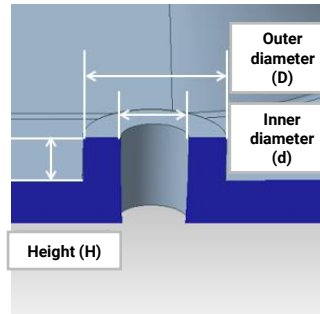
1. How to Check & Prerequisites

<How to Check>

Measures the inner diameter (d), the outer diameter (D), and the height (H) of recognized Burring, and then checks against the values in the specified table; first against "Diameter" to check whether the diameter matches one of the standards, second against "Min. height", and third against "Max. height" to check whether the height is within the range for the matched "Diameter".

<Prerequisites>

Geometry recognition is completed on "[Burring](#)".



2. How to Judge & Default Value

<How to Judge>

Marks Burring as Fail when its diameter does not match any of the standards, or its height is smaller than "Min. height" value, or greater than "Max. height" value.

"Formula" property will be shown as "-" (hyphen)" when the diameter does not match any of the standards.

<Default Value>

Specify the sets of burring diameter, min. height, and max. height.

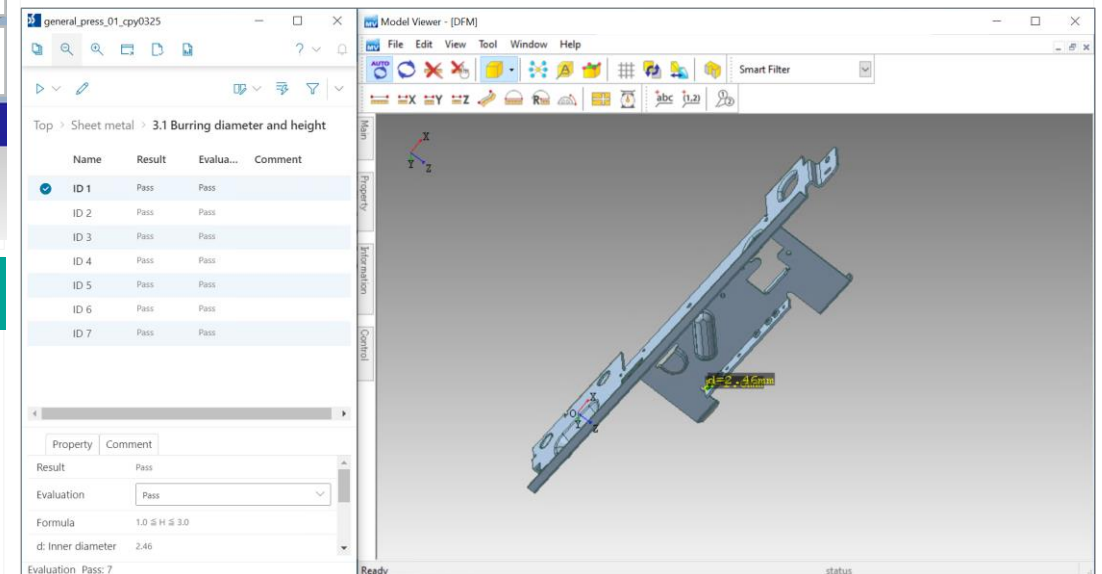
"Check burring diameter and height | Standard diameter and height sets by table" parameter:

BurringHeight.Table.String=line{2.4,1.0,3.0|2.7,1.0,3.0| 3.2,1.0,4.0|3.7,1.0,4.0}

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Burring whose diameter and height follows the standards Red: [Fail] Burring whose diameter and/or height do not follow the standards



3.1 Burring diameter and height

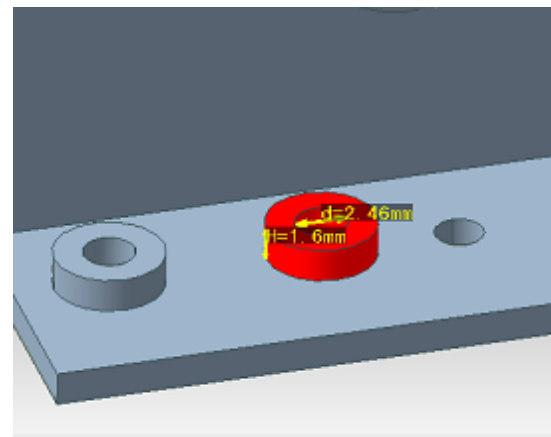


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

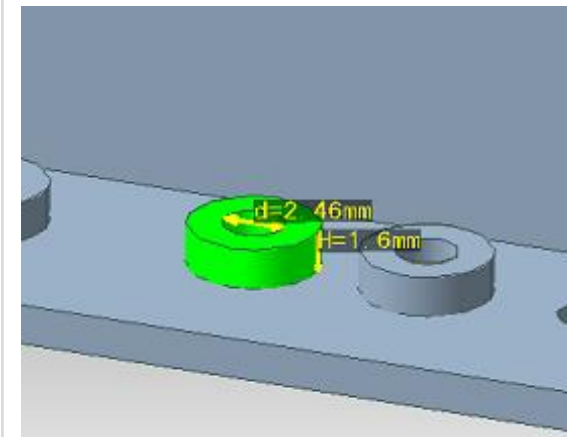
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Burring whose diameter and height follows the standards Red: [Fail] Burring whose diameter and/or height do not follow the standards
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

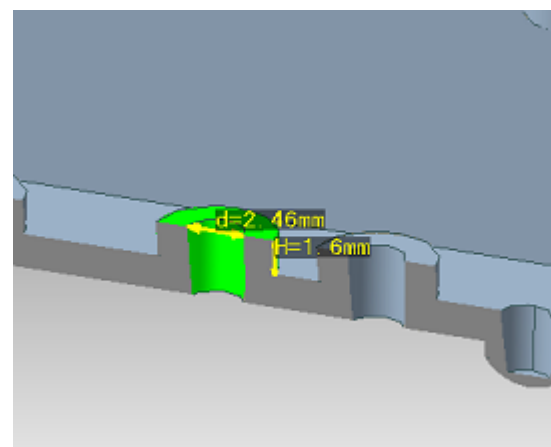
Highlight Checked Area - Fail



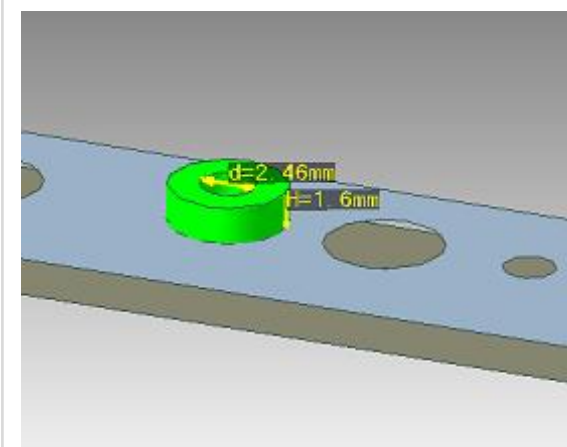
Highlight Checked Area - Pass



Section View



Hide Surroundings





3.2 Burring: Distance to hole or part end face

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a burring and a hole or a part end face is shorter than the specified tolerance.

<Possible Molding Defects>

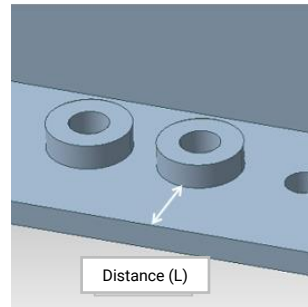


It may cause insufficient mold strength to make a burring too close to a hole or a part end face. It is recommended to make burring keeping an adequate distance from holes and part end faces.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a burring and a hole / part end face is greater than the tolerance. Please note that measurement may be conducted at several points for a pair of a burring and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.



<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", and "[Burring](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

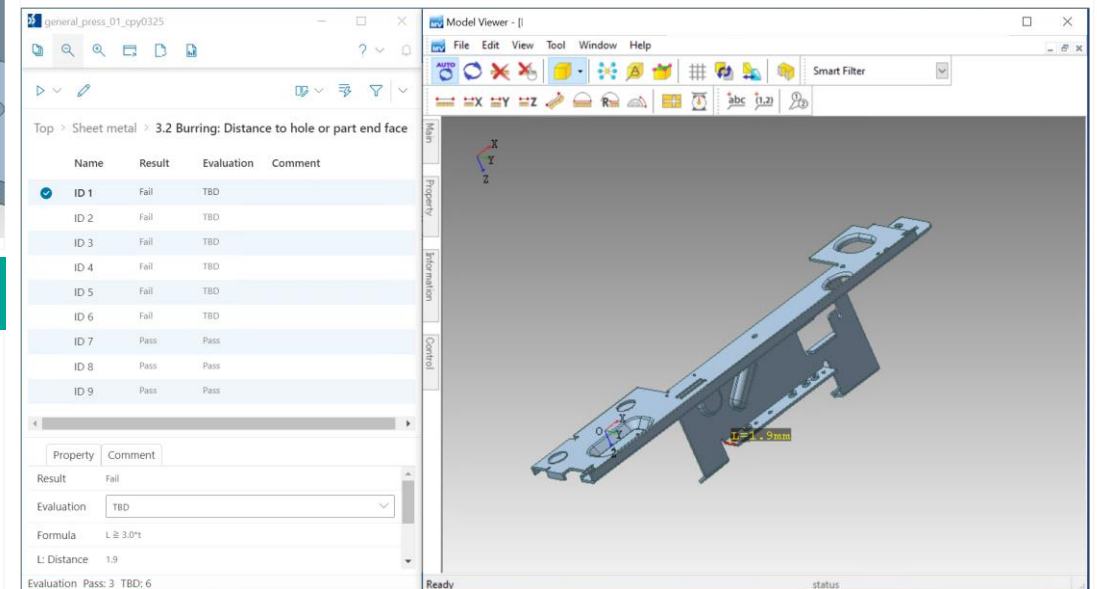
[A] When specifying by a fixed value with "Burring: Distance to hole or part end face | Min. distance tol by fixed value" parameter: BurringAndHoleDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Burring: Distance to hole or part end face | Min. distance tol by ratio to sheet metal thickness" parameter: BurringAndHoleDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Burring and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Burring and hole / part end face pair which the distance in between is shorter than the tolerance





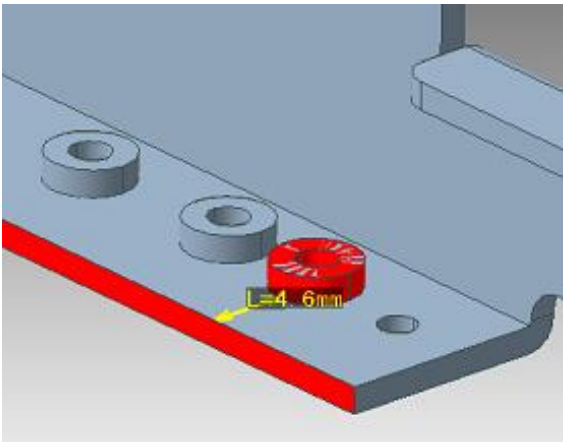
3.2 Burring: Distance to hole or part end face

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

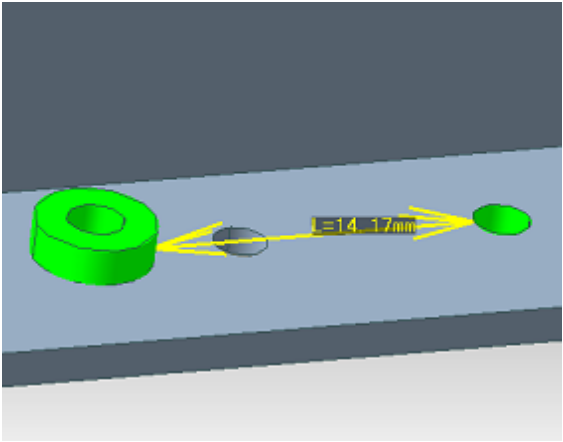
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Burring and hole / part end face pair which the distance in between is greater than the toleranceRed: [Fail] Burring and hole / part end face pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

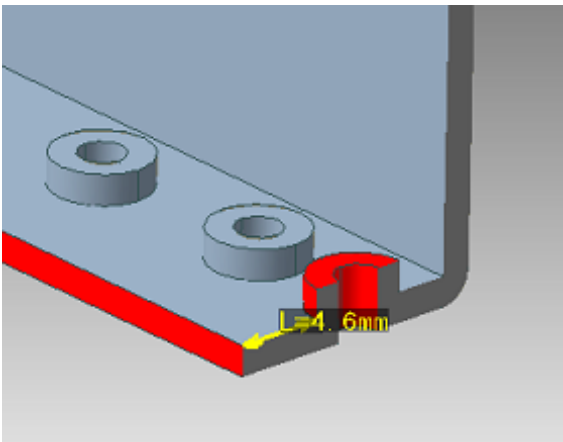
Highlight Checked Area - Fail



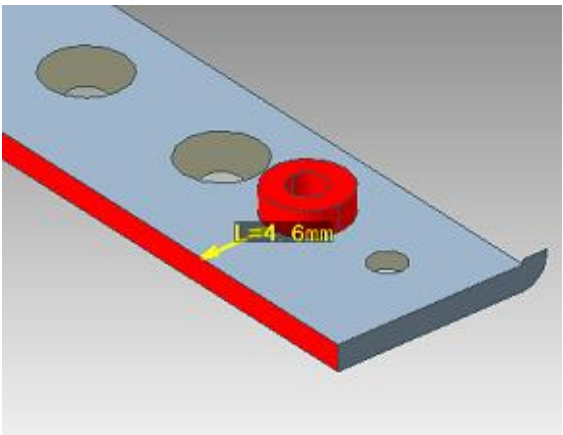
Highlight Checked Area - Pass



Section View



Hide Surroundings





3.3 Burring: Distance to burring

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a burring and another burring is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause restrictions for machining methods, and/or a rise in the cost to make a burring too close to another burring. It may also cause deformation in machining.

1. How to Check & Prerequisites

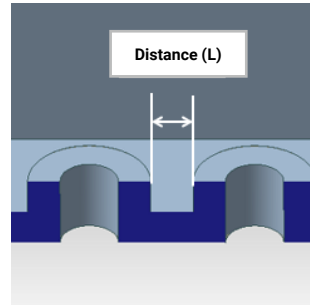
<How to Check>

Checks whether the distance (L) between a burring and another burring is greater than the tolerance.

Please note that measurement may be conducted at several points for a pair of two burring areas, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.

<Prerequisites>

Geometry recognition is completed on "[Burring](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

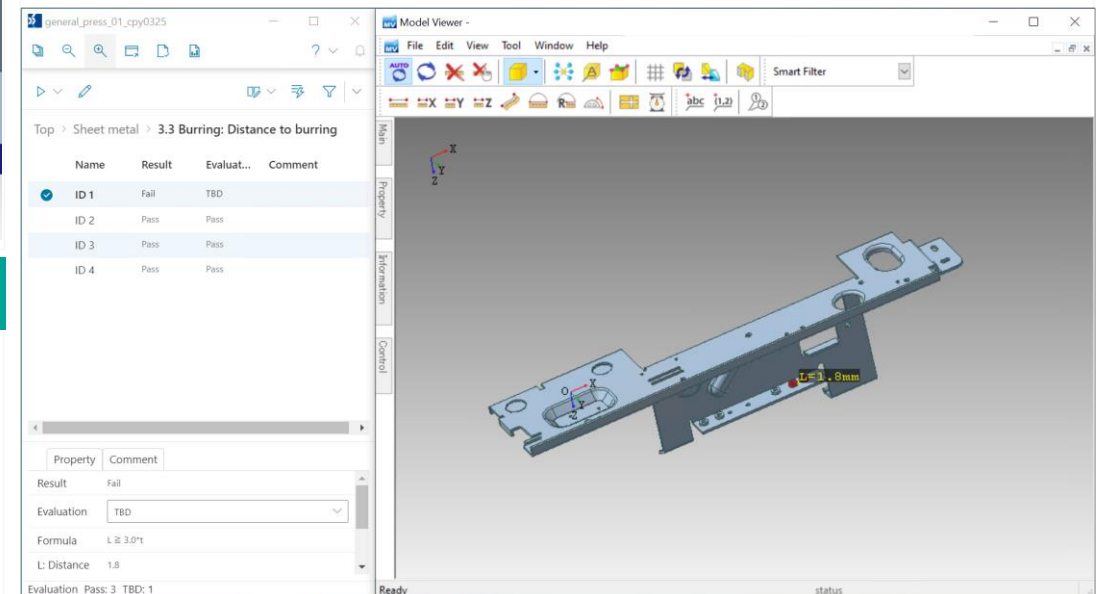
[A] When specifying by a fixed value with "Burring: Distance to burring | Min. distance tol by fixed value" parameter: BurringAndBurringDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Burring: Distance to burring | Min. distance tol by ratio to sheet metal thickness" parameter: BurringAndBurringDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Burring and another burring pair which the distance in between is greater than the tolerance Red: [Fail] Burring and another burring pair which the distance in between is shorter than the tolerance



3.3 Burring: Distance to burring

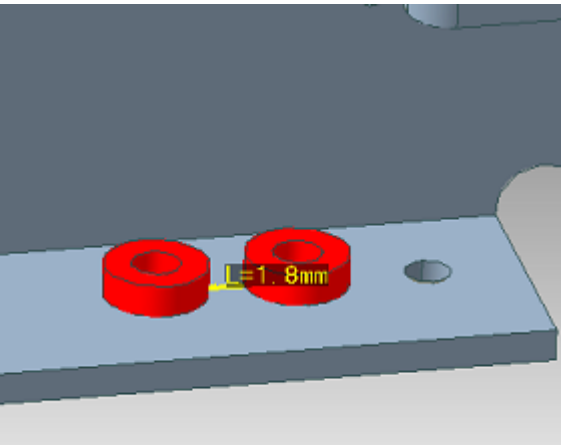


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

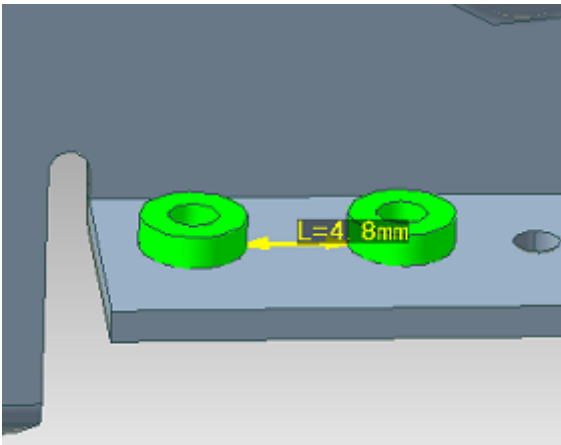
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Burring and another burring pair which the distance in between is greater than the toleranceRed: [Fail] Burring and another burring pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

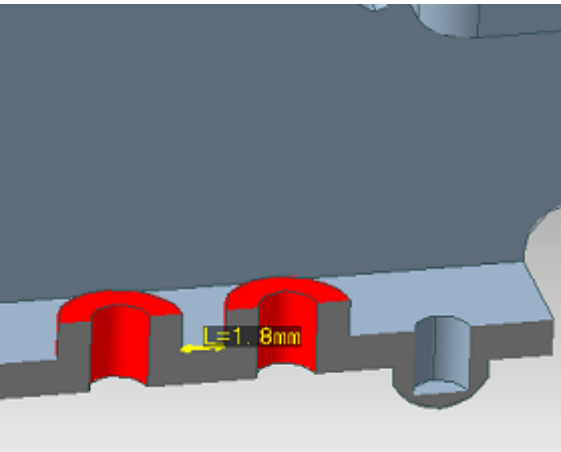
Highlight Checked Area - Fail



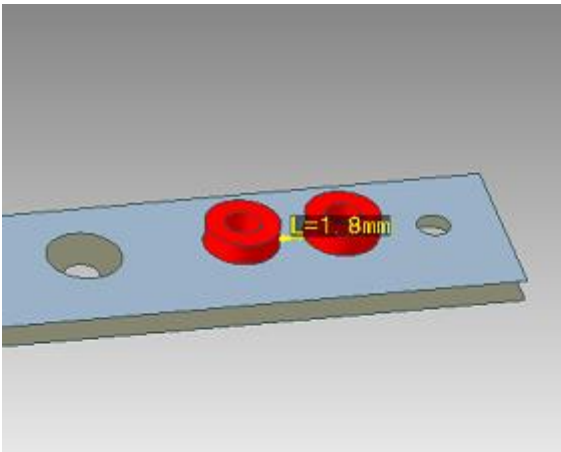
Highlight Checked Area - Pass

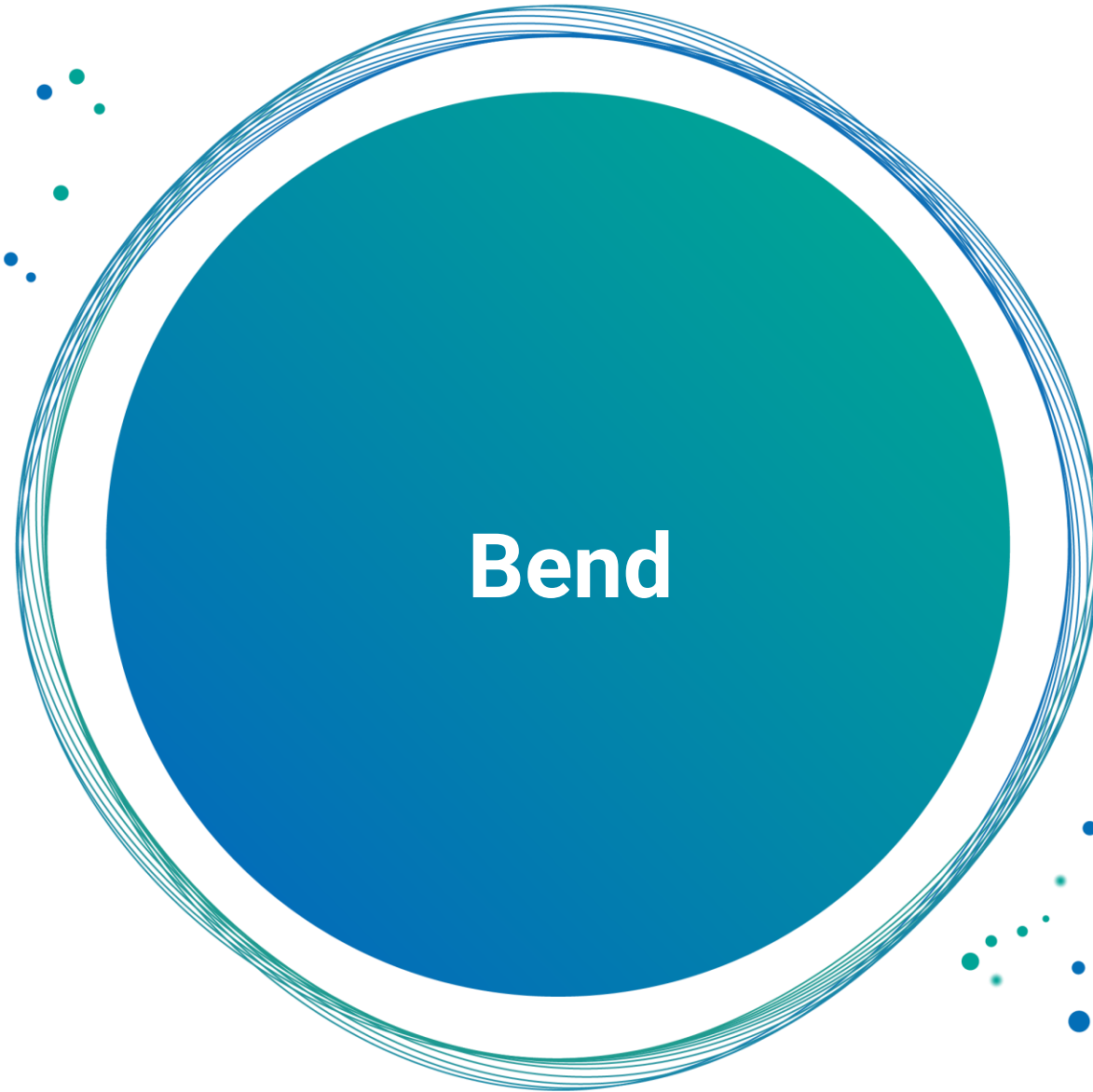


Section View



Hide Surroundings





4.1 Bend height




Description of Check Criterion & Possible Molding Defects

<Description>

Detects the area where the bend flange length (distance between an edge or a fillet on a bend and a part end face) is shorter than the specified tolerance.

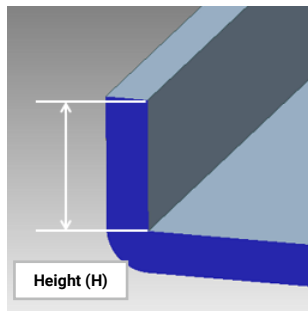
<Possible Molding Defects>

 It will increase the difficulty in machining which will cause the variance on the bend angle, etc. when the flange length is too short.

1. How to Check & Prerequisites

<How to Check>

Checks whether the bend flange length (H) (distance between an edge or a fillet on a bend and a part end face) is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)" and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the bend as Fail when the flange length (H) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

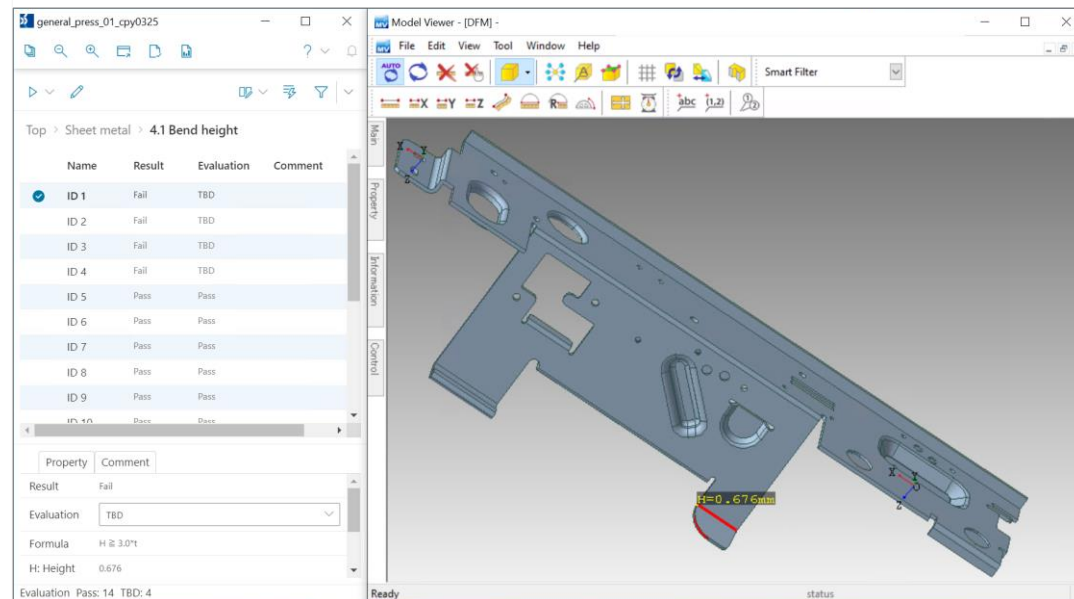
[A] When specifying by a fixed value with "Bend height | Min. height tol by fixed value" parameter: BendingEndDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend height | Min. height tol by ratio to sheet metal thickness" parameter: BendingEndDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Bends whose flange length are greater than the tolerance Red: [Fail] Bends whose flange length are shorter than the tolerance



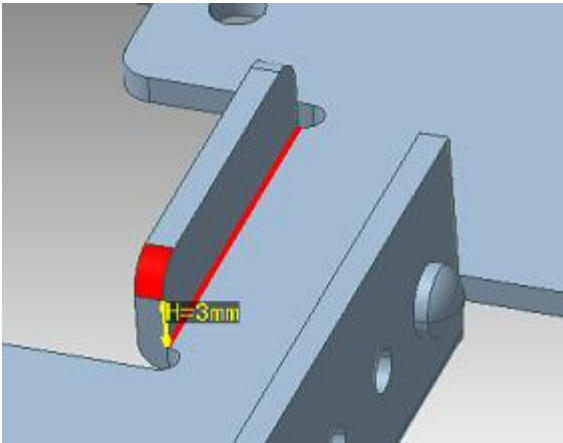
4.1 Bend height

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

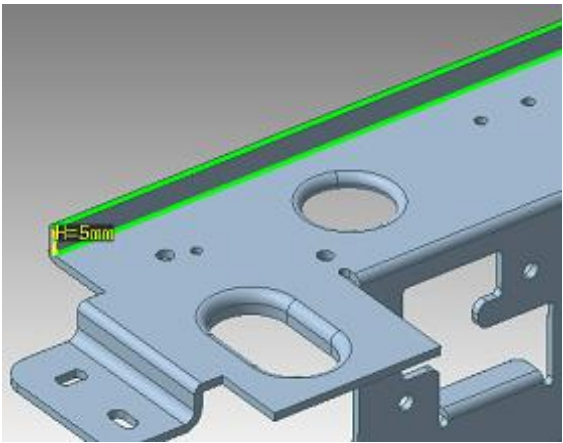
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Bends whose flange length are greater than the tolerance Red: [Fail] Bends whose flange length are shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

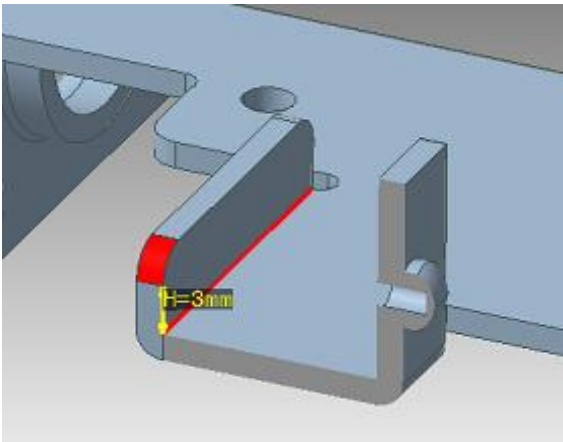
Highlight Checked Area - Fail



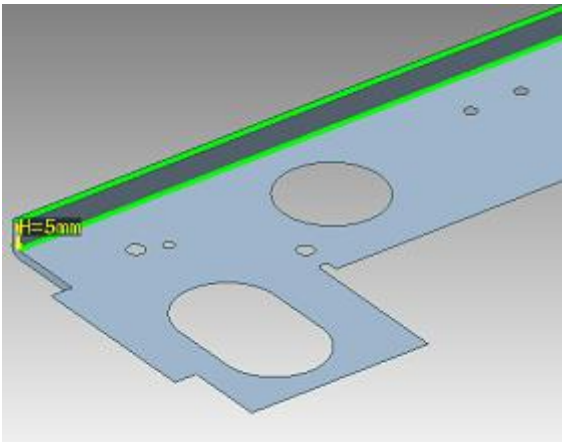
Highlight Checked Area - Pass



Section View



Hide Surroundings



4.2 Z bend flange length

Description of Check Criterion & Possible Molding Defects

<Description>

Detects the area where the Z bend flange length (distance between an edge or a fillet on an inward bend and a fillet on an outward bend) is shorter than the specified tolerance.

<Possible Molding Defects>

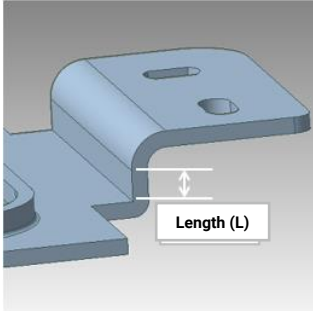


It will increase the difficulty in machining which will cause the variance on the bend angle, etc. when the Z bend flange length is too short.

1. How to Check & Prerequisites

<How to Check>

Checks whether the Z bend flange length (L) (distance between an edge or a fillet on an inward bend and a fillet on an outward bend) is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)" and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the Z bend as Fail when the flange length (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

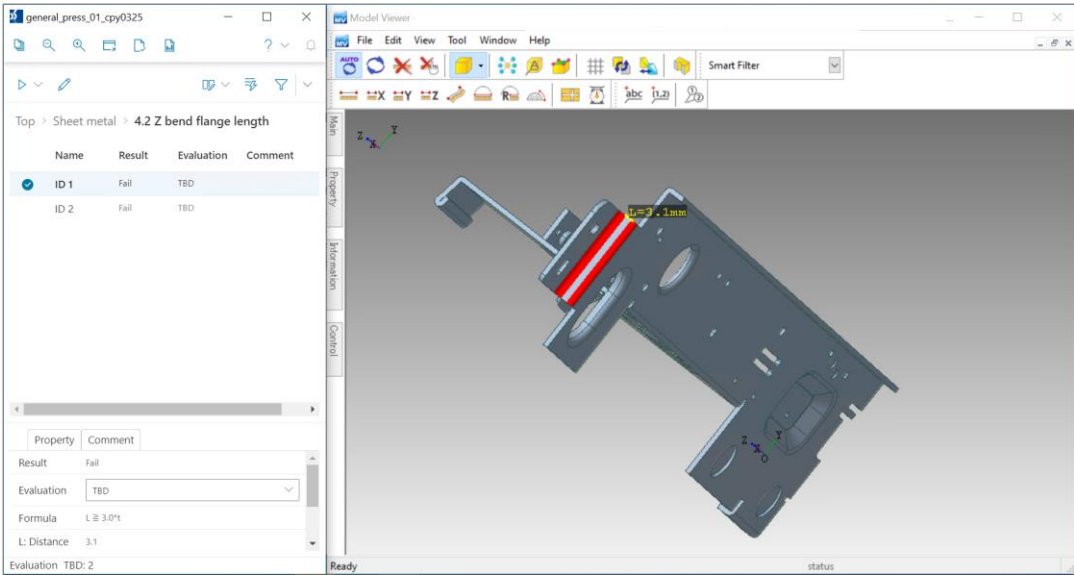
[A] When specifying by a fixed value with "Z bend flange length | Min. length tol by fixed value" parameter:
BendingAndBendingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Z bend flange length | Min. length tol by ratio to sheet metal thickness" parameter: BendingAndBendingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Z bends whose flange length are greater than the toleranceRed: [Fail] Z bends whose flange length are shorter than the tolerance



4.2 Z bend flange length

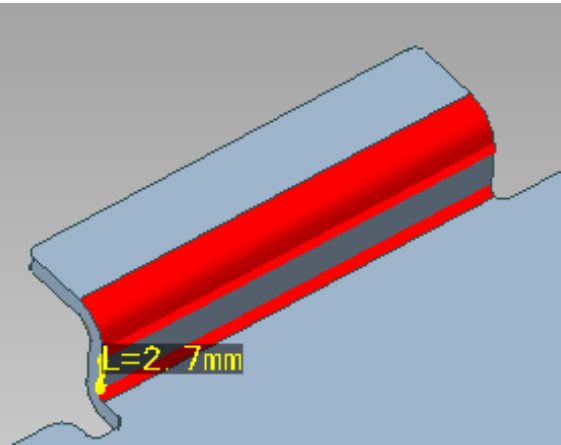


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

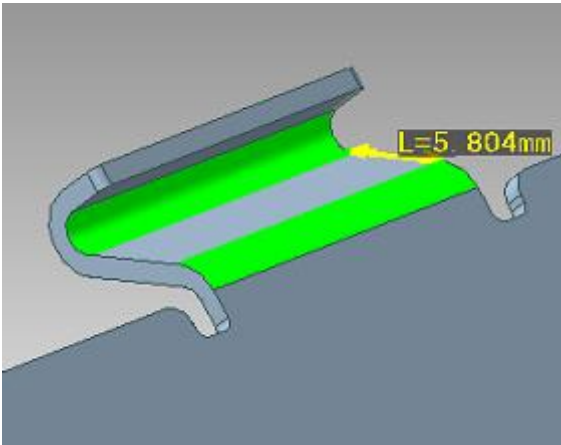
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Z bends whose flange length are greater than the toleranceRed: [Fail] Z bends whose flange length are shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

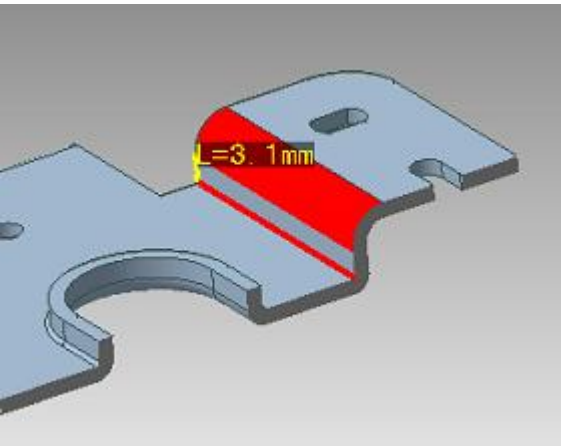
Highlight Checked Area - Fail



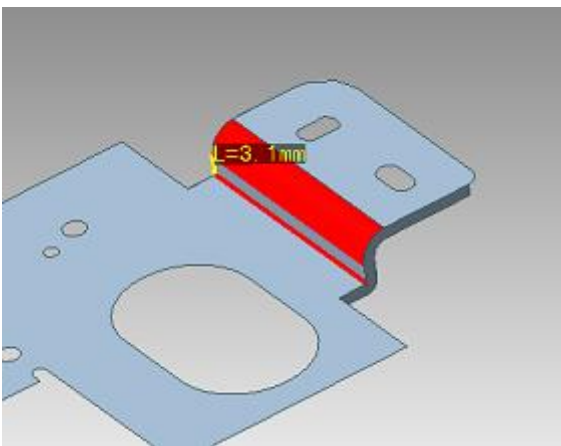
Highlight Checked Area - Pass



Section View



Hide Surroundings



4.3 U bend flange length




Description of Check Criterion & Possible Molding Defects

<Description>

Detects the area where the U bend flange length (distance between an edge or a fillet on an inward bend and an edge or a fillet on an inward bend) is shorter than the specified tolerance.

<Possible Molding Defects>

 It will increase the difficulty in machining which will cause the variance on the bend angle, etc. when the U bend flange length is too short.

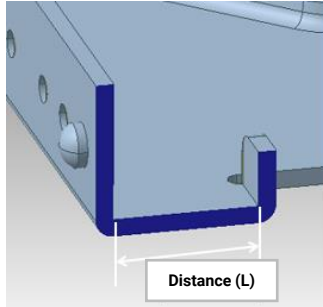
1. How to Check & Prerequisites

<How to Check>

Checks whether the U bend flange length (L) (distance between an edge or a fillet on an inward bend and an edge or a fillet on an inward bend) is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)" and "[Hem bend](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the U bend as Fail when the flange length (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

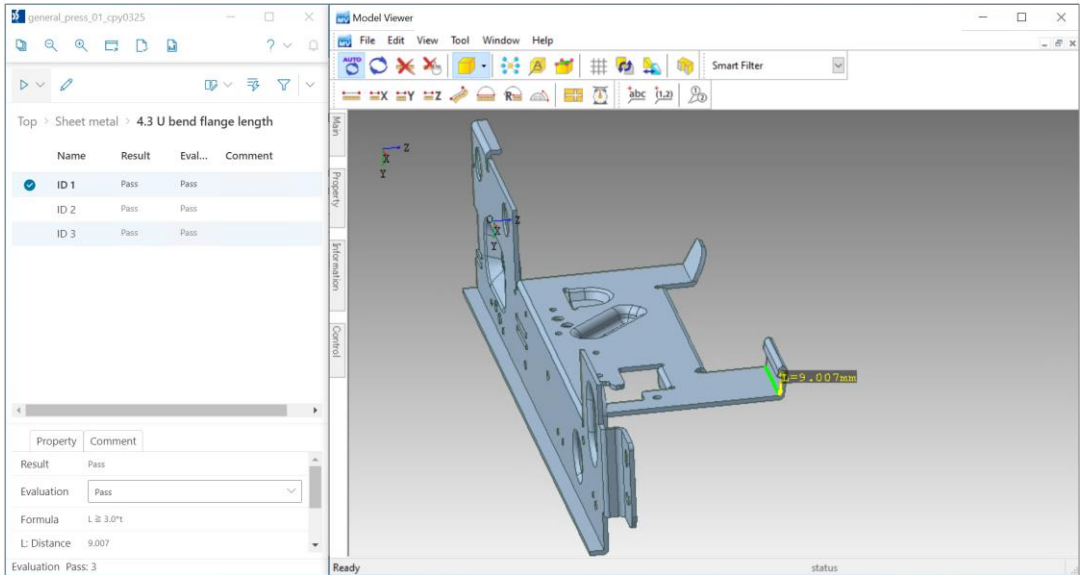
[A] When specifying by a fixed value with "U bend flange length | Min. length tol by fixed value" parameter:
BendingAndBendingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "U bend flange length | Min. length tol by ratio to sheet metal thickness" parameter: BendingAndBendingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] U bends whose flange length are greater than the toleranceRed: [Fail] U bends whose flange length are shorter than the tolerance



4.3 U bend flange length

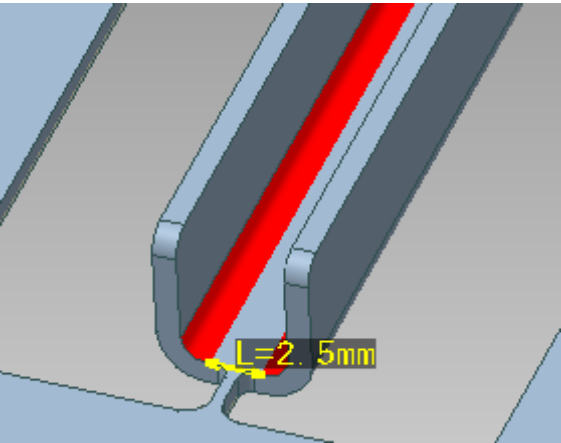


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

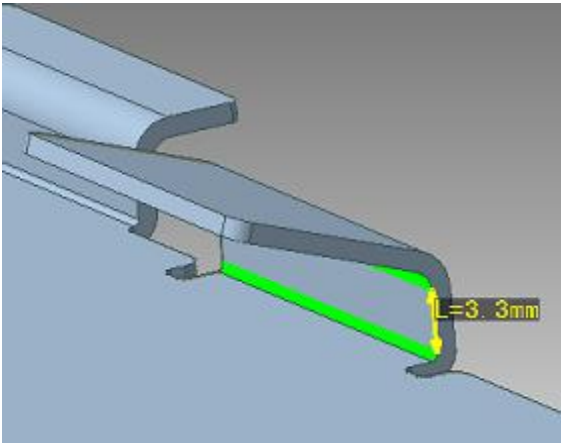
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] U bends whose flange length are greater than the toleranceRed: [Fail] U bends whose flange length are shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

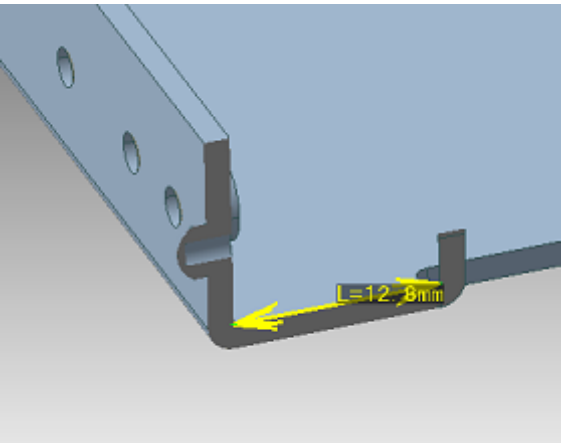
Highlight Checked Area - Fail



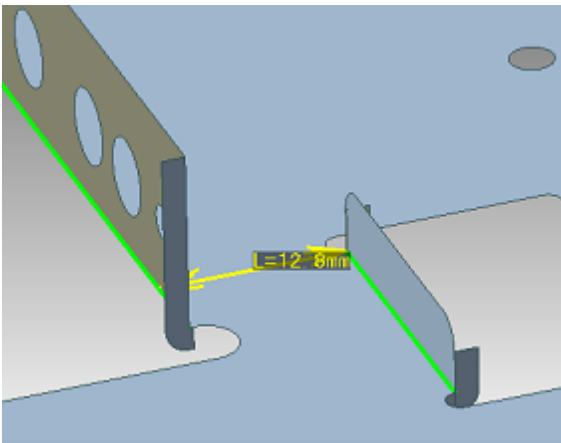
Highlight Checked Area - Pass



Section View



Hide Surroundings






4.4 Bend: Distance to hole

Description of Check Criterion & Possible Molding Defects

<Description>

Detects the area where the distance between an edge or a fillet on an inward bend and a hole is shorter than the specified tolerance.

<Possible Molding Defects>

 It may cause insufficient mold strength to make holes too close to a bend. It also often deforms the hole to bend after making holes. It is recommended to make bends keeping an adequate distance from holes.

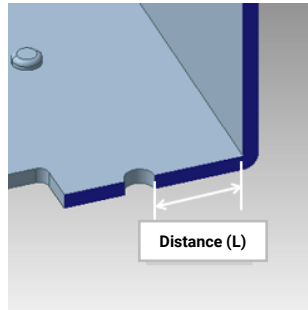
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between an edge or a fillet on an inward bend and a hole is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)", "[Hem bend](#)", and "[Punching hole](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

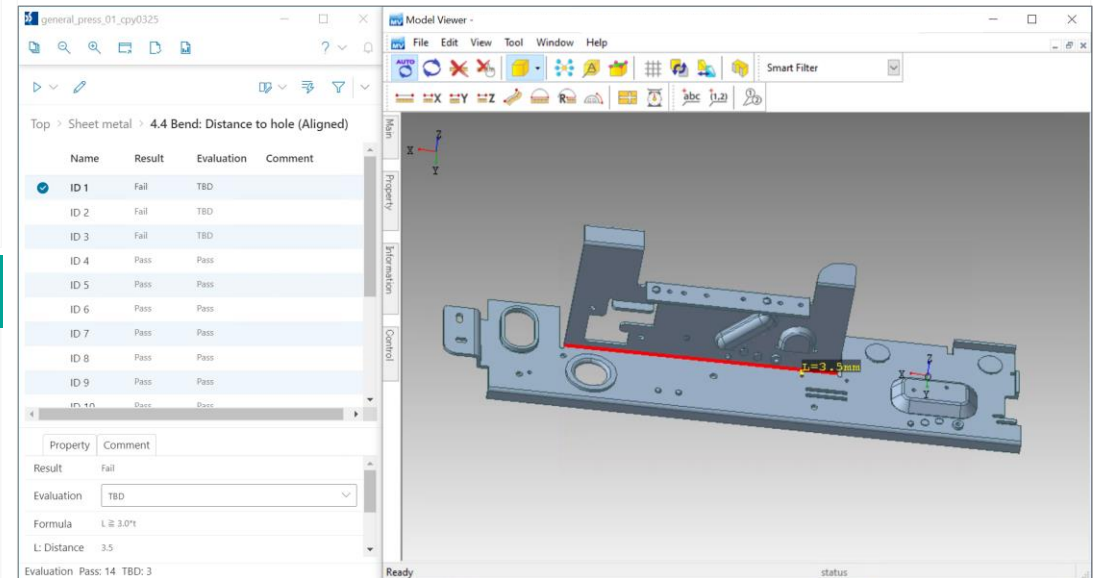
[A] When specifying by a fixed value with "Bend: Distance to hole | Min. distance tol by fixed value" parameter: BendingAndHoleDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend: Distance to hole | Min. distance tol by ratio to sheet metal thickness" parameter: BendingAndHoleDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Bend and hole pair which the distance in between is greater than the tolerance Red: [Fail] Bend and hole pair which the distance in between is shorter than the tolerance





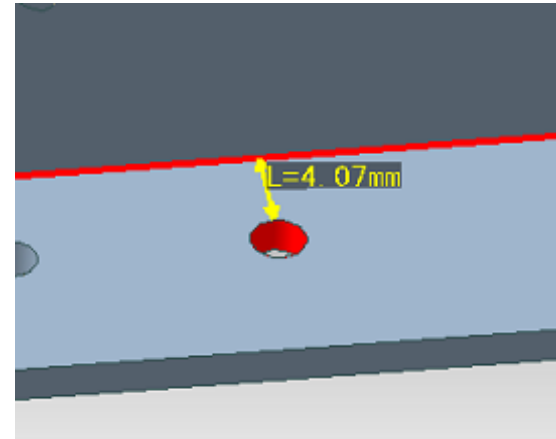
4.4 Bend: Distance to hole

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

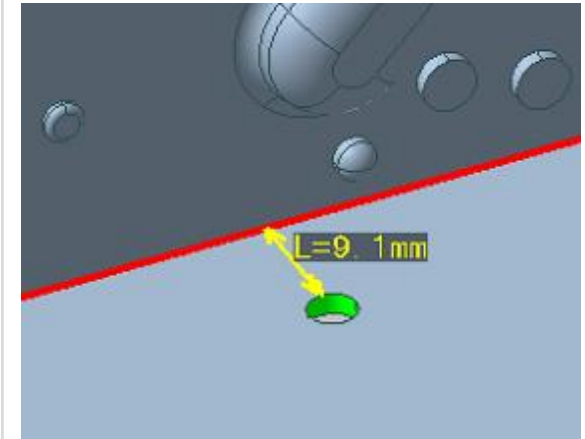
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend and hole pair which the distance in between is greater than the toleranceRed: [Fail] Bend and hole pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

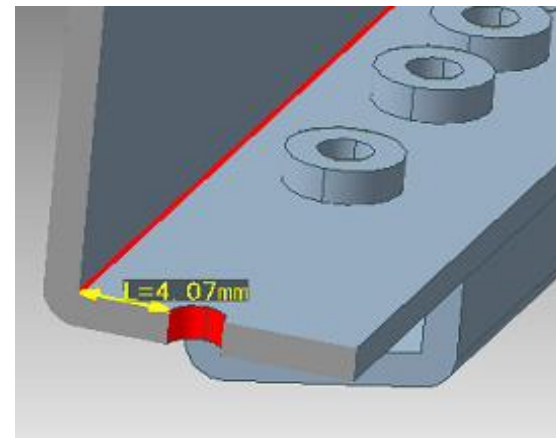
Highlight Checked Area - Fail



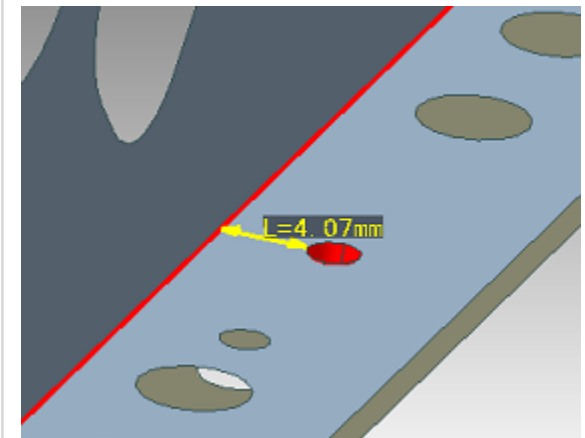
Highlight Checked Area - Pass



Section View



Hide Surroundings



4.5 Bend: Distance to hole (Parallel)

Description of Check Criterion & Possible Molding Defects

<Description>

When there is a hole which has straight-line edge(s) near the bend, detects the area where the distance between an edge or a fillet on an inward bend and a straight-line edge of a hole is shorter than the specified tolerance.

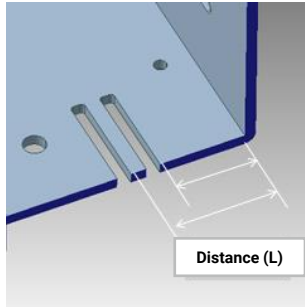
<Possible Molding Defects>

It may cause insufficient mold strength to make holes too close to a bend. It also often deforms the hole to bend after making holes. It is recommended to make bends keeping an adequate distance from holes.

1. How to Check & Prerequisites

<How to Check>

For a bend and a hole which have straight-line edges parallel to the bend, checks whether the distance (L) between an edge or a fillet on an inward bend and a straight-line edge of a hole is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)", "[Hem bend](#)", and "[Punching hole](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

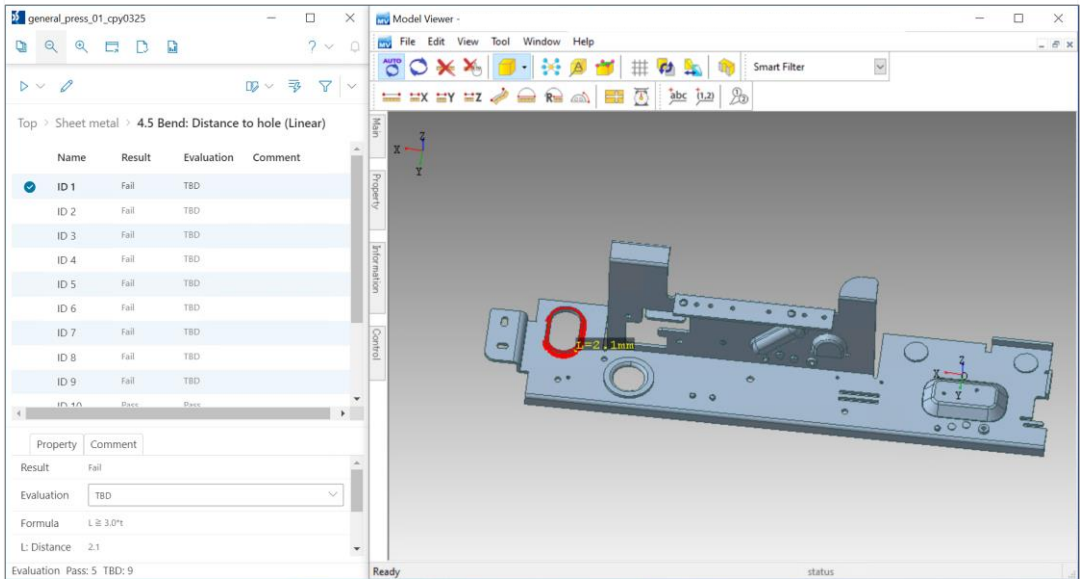
[A] When specifying by a fixed value with "Bend: Distance to hole (Parallel) | Min. distance tol by fixed value" parameter: BendingAndHoleDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend: Distance to hole (Parallel) | Min. distance tol by ratio to sheet metal thickness" parameter: BendingAndHoleDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend and hole pair which the distance in between is greater than the toleranceRed: [Fail] Bend and hole pair which the distance in between is shorter than the tolerance





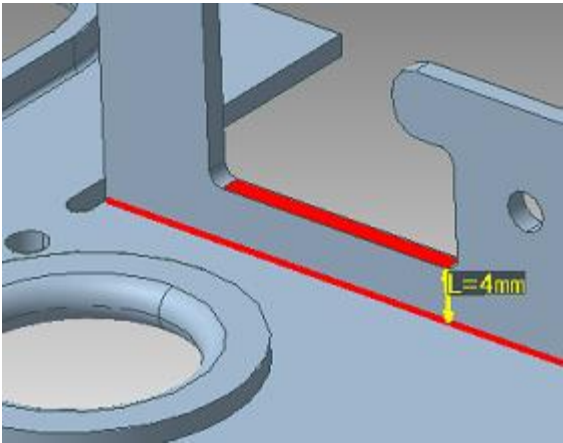
4.5 Bend: Distance to hole (Parallel)

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

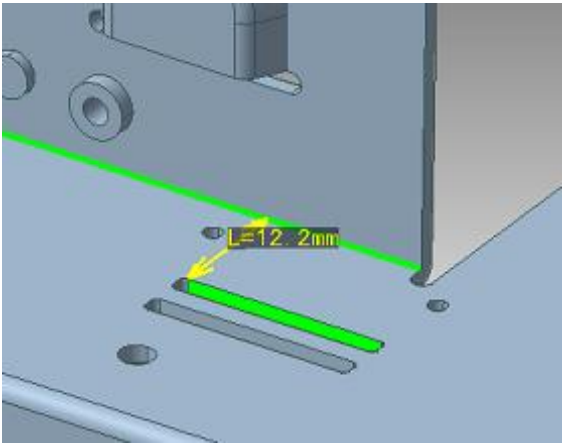
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend and hole pair which the distance in between is greater than the toleranceRed: [Fail] Bend and hole pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

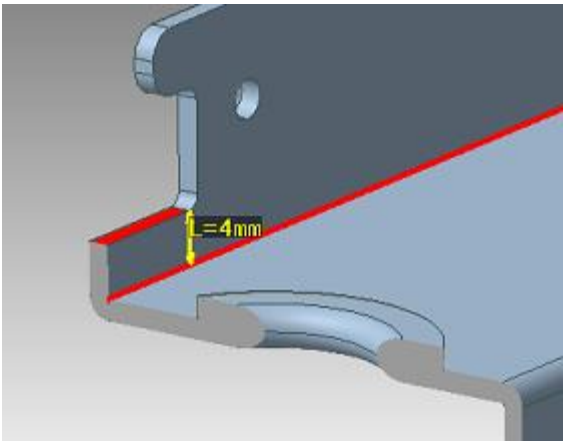
Highlight Checked Area - Fail



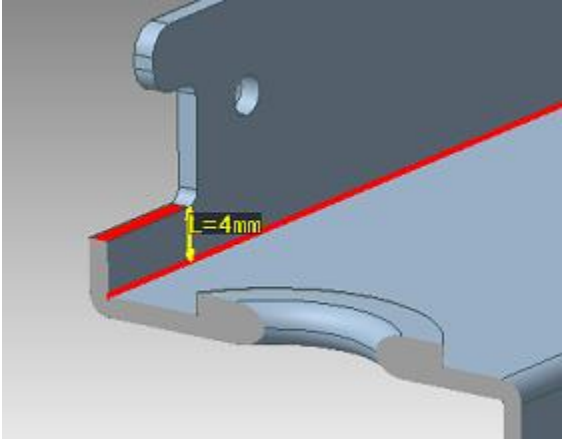
Highlight Checked Area - Pass

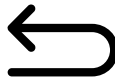


Section View



Hide Surroundings





4.6 Bend: Distance to burring

Description of Check Criterion & Possible Molding Defects

<Description>

Detects the area where the distance between an edge or a fillet on an inward bend and burring is shorter than the specified tolerance.

<Possible Molding Defects>



It may deform the burring to bend after burring sheet metal.
It is recommended to make bends keeping an adequate distance from burring.

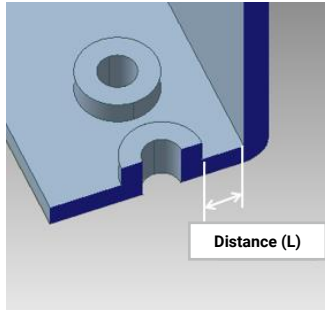
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between an edge or a fillet on an inward bend and burring is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Simple bend](#)", "[Hem bend](#)", and "[Burring](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

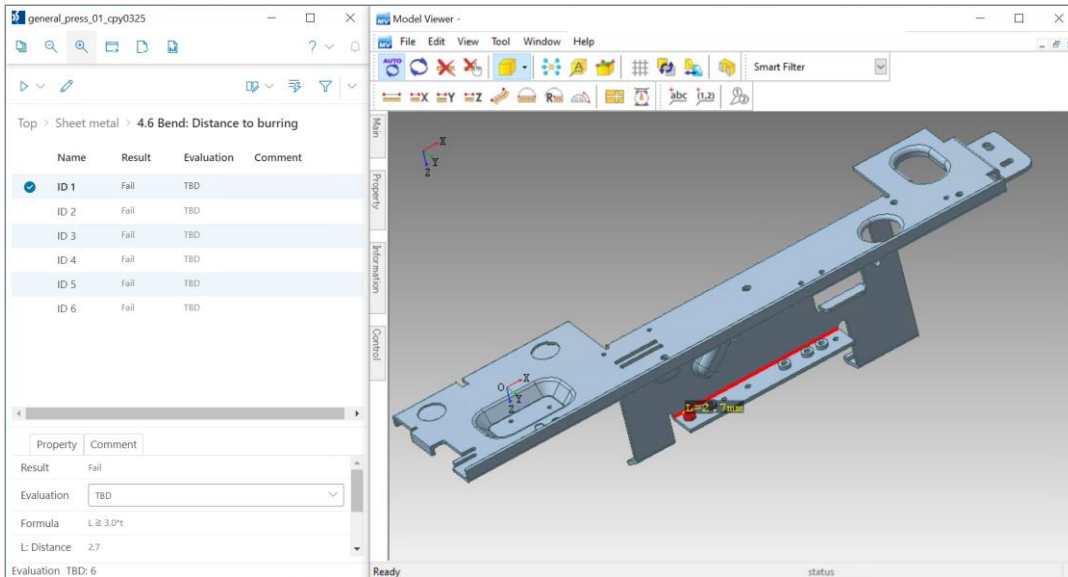
[A] When specifying by a fixed value with "Bend: Distance to burring | Min. distance tol by fixed value" parameter: BendingAndBurringDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend: Distance to burring | Min. distance tol by ratio to sheet metal thickness" parameter: BendingAndBurringDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Bend and burring pair which the distance in between is greater than the tolerance Red: [Fail] Bend and burring pair which the distance in between is shorter than the tolerance





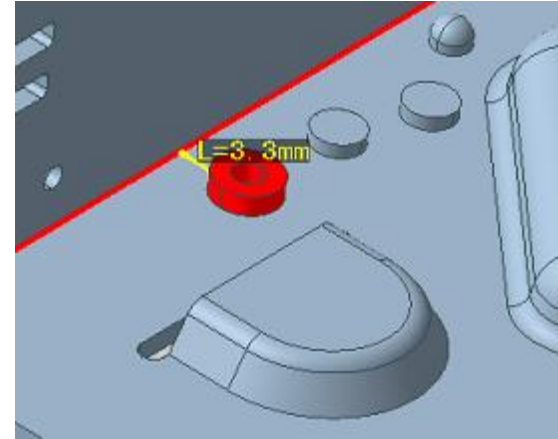
4.6 Bend: Distance to burring

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

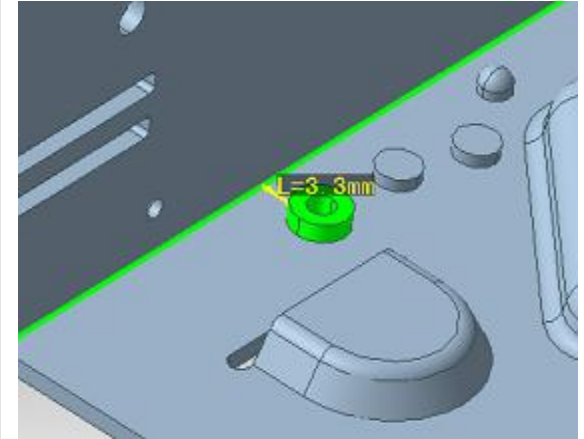
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend and burring pair which the distance in between is greater than the toleranceRed: [Fail] Bend and burring pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

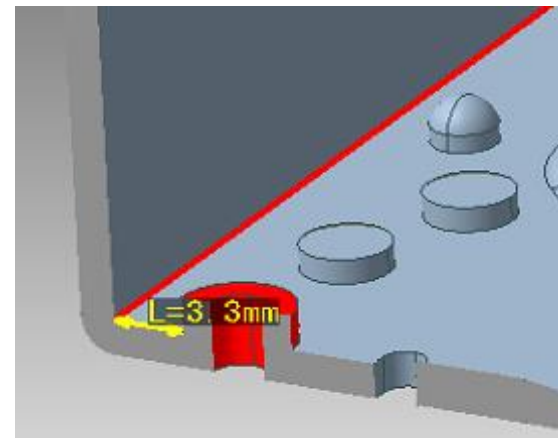
Highlight Checked Area - Fail



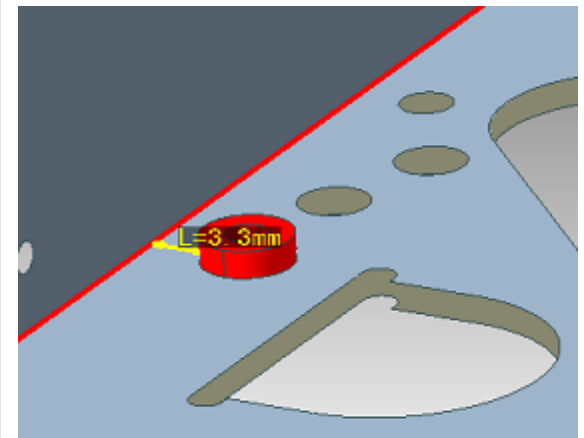
Highlight Checked Area - Pass



Section View



Hide Surroundings





Groove / Thin area



5.1 Groove width (Thin area)

Description of Check Criterion & Possible Molding Defects

<Description>

Detects the grooves whose width (=distance between its Side faces) are smaller than the specified tolerance.

<Possible Molding Defects>



It may deform in machining to make thin grooves.
It may also damage the tools, and/or cause a decrease in durability of the tool.

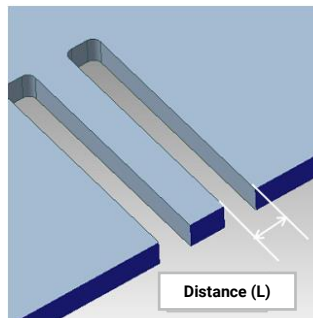
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between two Side faces which are facing each other is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", and "[Punching hole](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

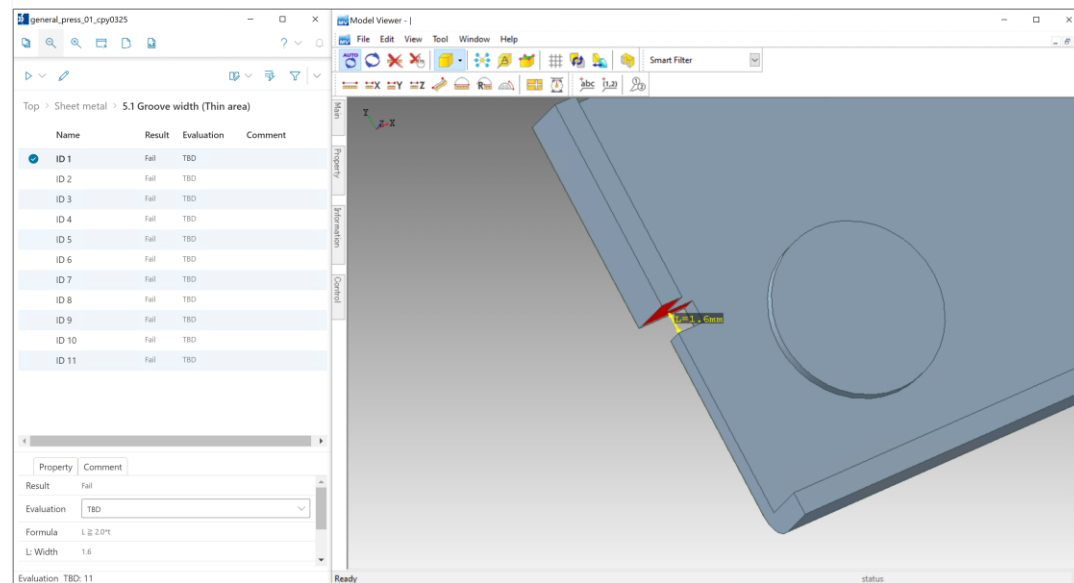
[A] When specifying by a fixed value with "Groove width (Thin area) | Min. width tol by fixed value" parameter: SideFaceConcaveDistance.Min=2.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Groove width (Thin area) | Min. width tol by ratio to sheet metal thickness" parameter: SideFaceConcaveDistanceMin.Ratio=2.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Red: [Fail] Grooves whose width are smaller than the tolerance <i>Note) This detects Fail areas only (=does not mark Pass areas)</i>



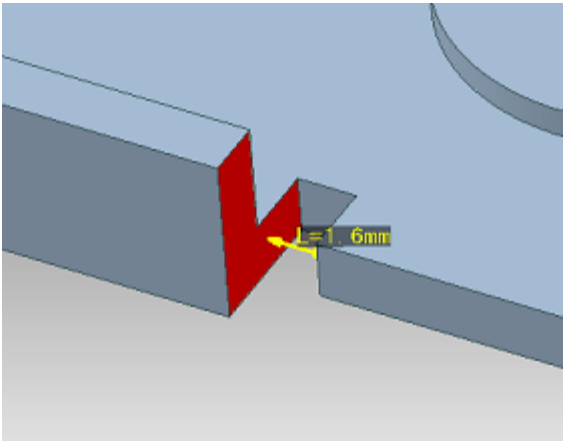
5.1 Groove width (Thin area)

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Grooves whose width are smaller than the tolerance <i>Note) This detects Fail areas only (=does not mark Pass areas)</i>
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

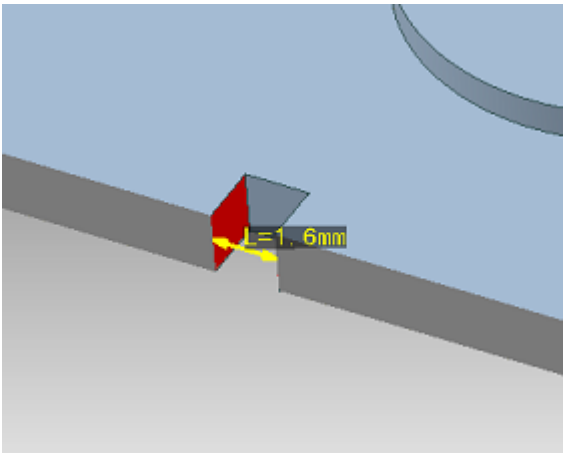
Highlight Checked Area - Fail



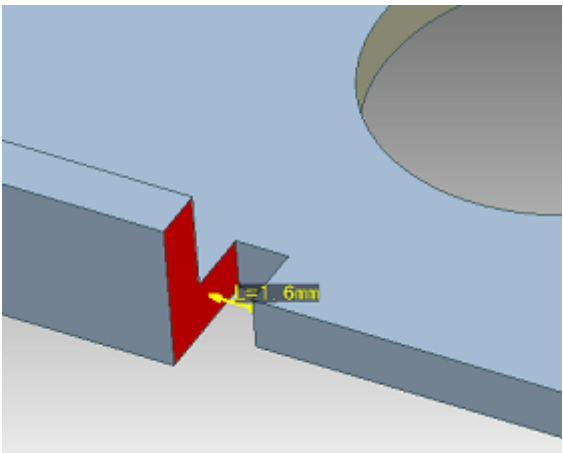
Highlight Checked Area - Pass

N/A

Section View



Hide Surroundings



5.2 Thin area width



Description of Check Criterion & Possible Molding Defects

<Description>

Detects the thin area where the width (=distance between its Side faces) is smaller than the specified tolerance.

<Possible Molding Defects>



It may deform in machining to have thin areas.

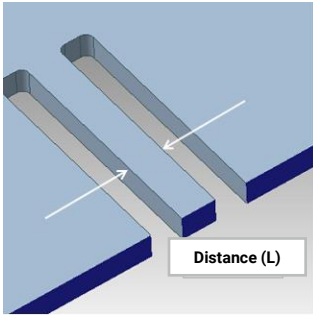
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between two Side faces which are not facing each other is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", and "[Punching hole](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

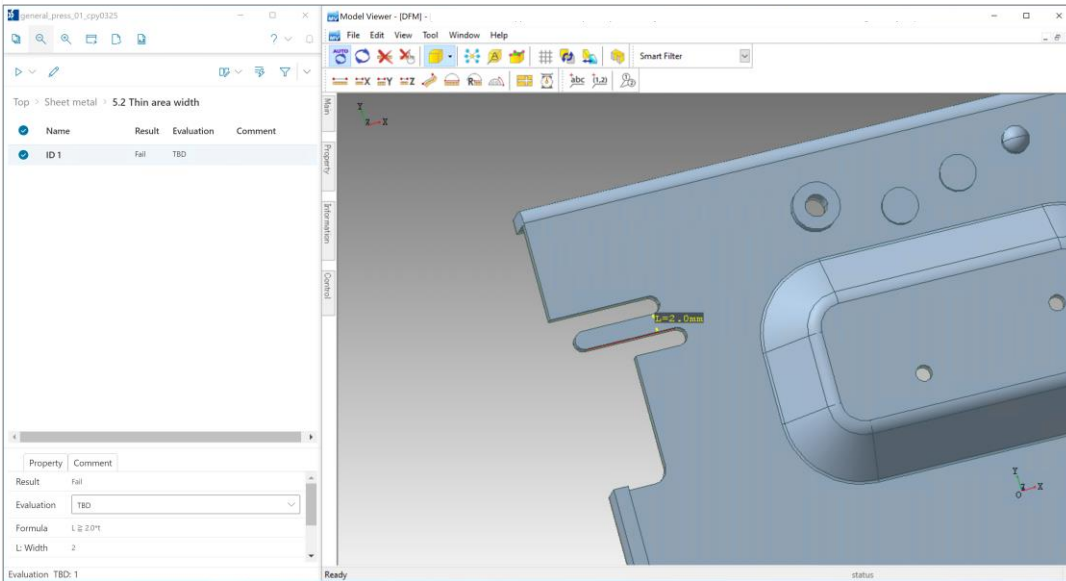
[A] When specifying by a fixed value with "Thin area width | Min. width tol by fixed value" parameter:
SideFaceConvexDistance.Min=2.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Thin area width | Min. width tol by ratio to sheet metal thickness" parameter: SideFaceConvexDistanceMin.Ratio=2.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Thin area whose width is smaller than the tolerance <i>Note) This detects Fail areas only (=does not mark Pass areas)</i>



5.2 Thin area width

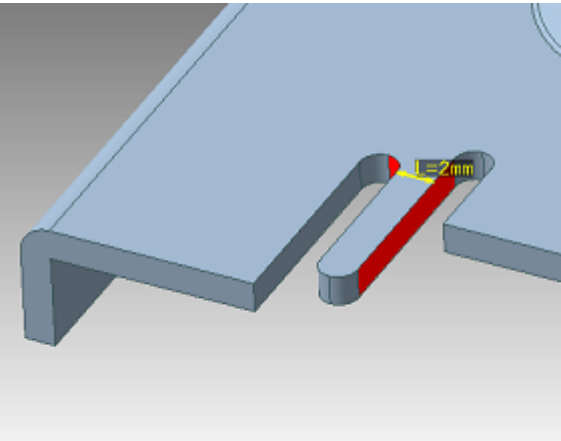


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Red: [Fail] Thin area whose width is smaller than the tolerance <i>Note) This detects Fail areas only (=does not mark Pass areas)</i>
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

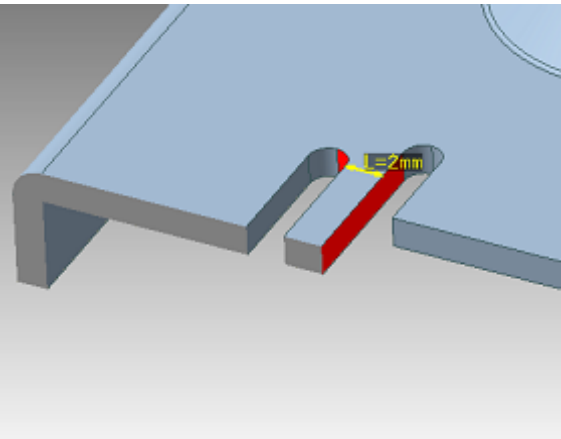
Highlight Checked Area - Fail



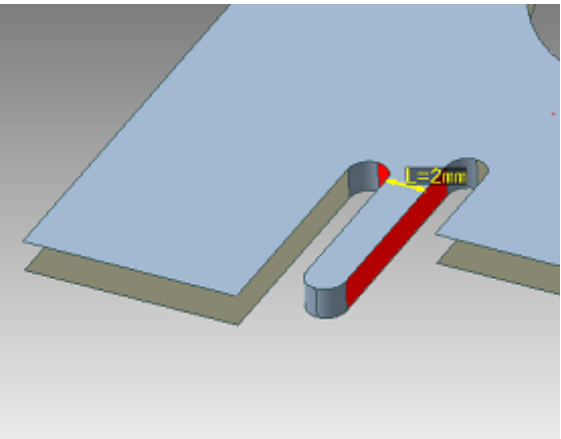
Highlight Checked Area - Pass

N/A

Section View



Hide Surroundings



5.3 Bend relief cut depth




Description of Check Criterion & Possible Molding Defects

<Description>

Detects the bend relief cuts whose depth are smaller than the specified tolerance.

<Possible Molding Defects>

 It may deform in machining when the bend relief cut depth is insufficient.

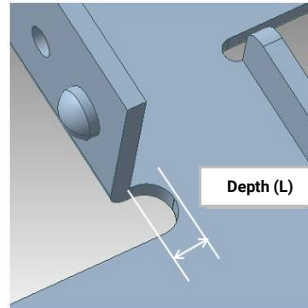
1. How to Check & Prerequisites

<How to Check>

Checks whether the bend relief cut depth (L) is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", "[Simple bend](#)", and "[Hem bend](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the depth (L) is smaller than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

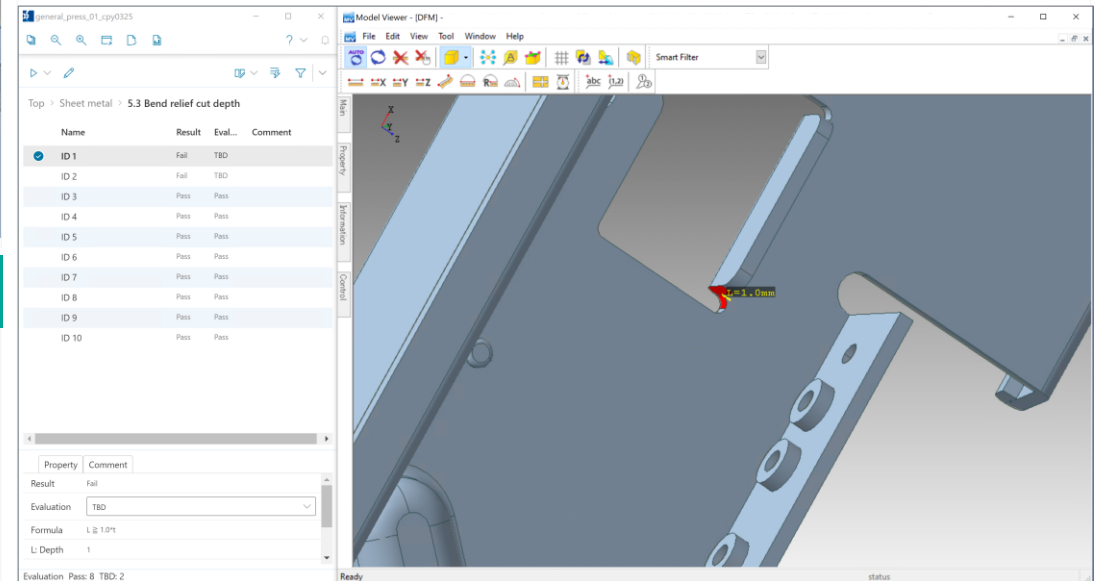
[A] When specifying by a fixed value with "Bend relief cut depth | Min. depth tol by fixed value" parameter: BendingReliefDepth.Min=1.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend relief cut depth | Min. distance tol by ratio to sheet metal thickness" parameter: BendingReliefDepth.Min.Ratio=1.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Bend relief cuts whose depth are greater than the tolerance Red: [Fail] Bend relief cuts whose depth are smaller than the tolerance



5.3 Bend relief cut depth

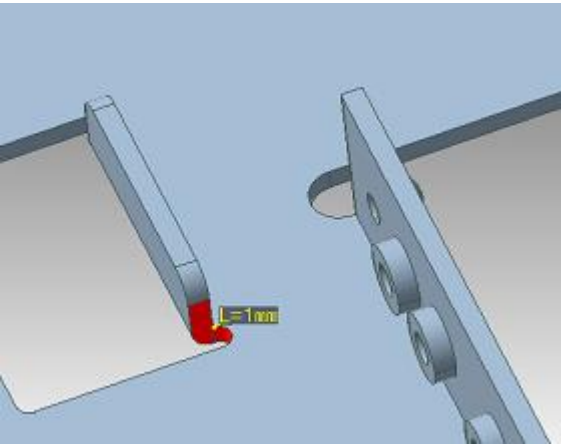


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

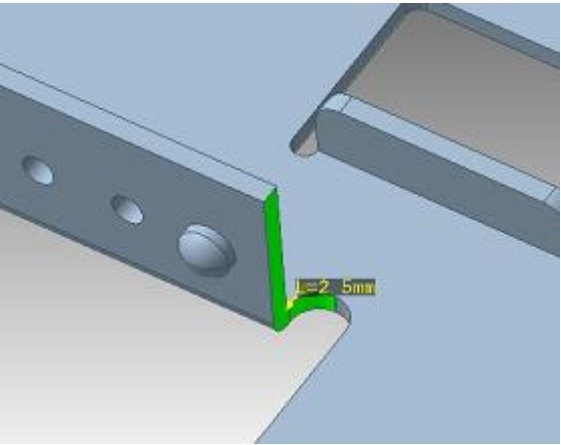
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend relief cuts whose depth are greater than the toleranceRed: [Fail] Bend relief cuts whose depth are smaller than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

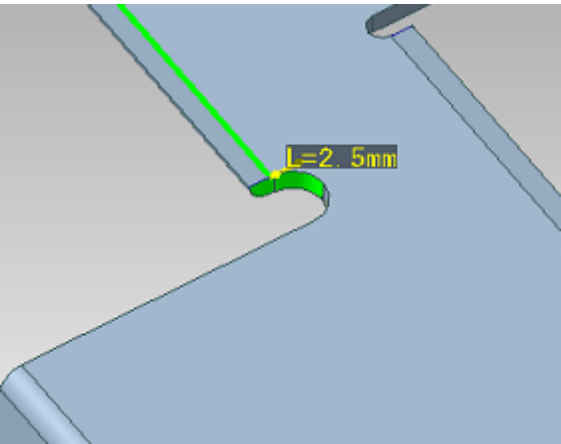
Highlight Checked Area - Fail



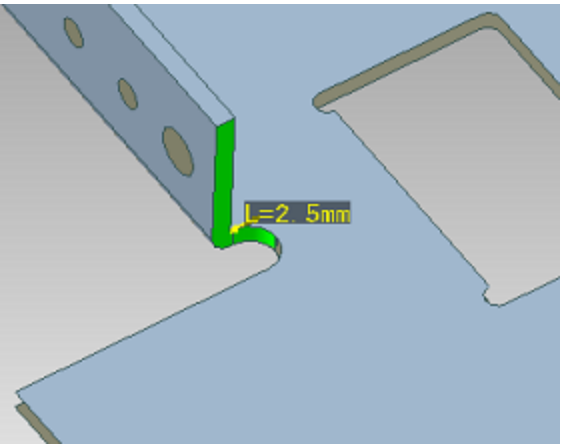
Highlight Checked Area - Pass



Section View



Hide Surroundings



5.4 Bend relief cut width



Description of Check Criterion & Possible Molding Defects

<Description>

Detects the bend relief cuts whose width are smaller than the specified tolerance.

<Possible Molding Defects>

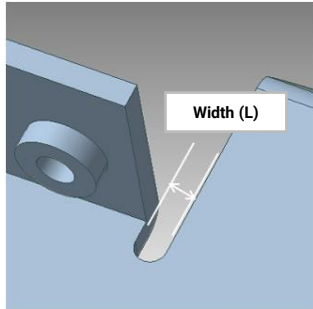


It may deform in machining when the bend relief cut width is insufficient.
It may also damage the tools, and/or cause a decrease in durability of the tool.

1. How to Check & Prerequisites

<How to Check>

Checks whether the bend relief cut width (L) is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", "[Simple bend](#)", and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the width (L) is smaller than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

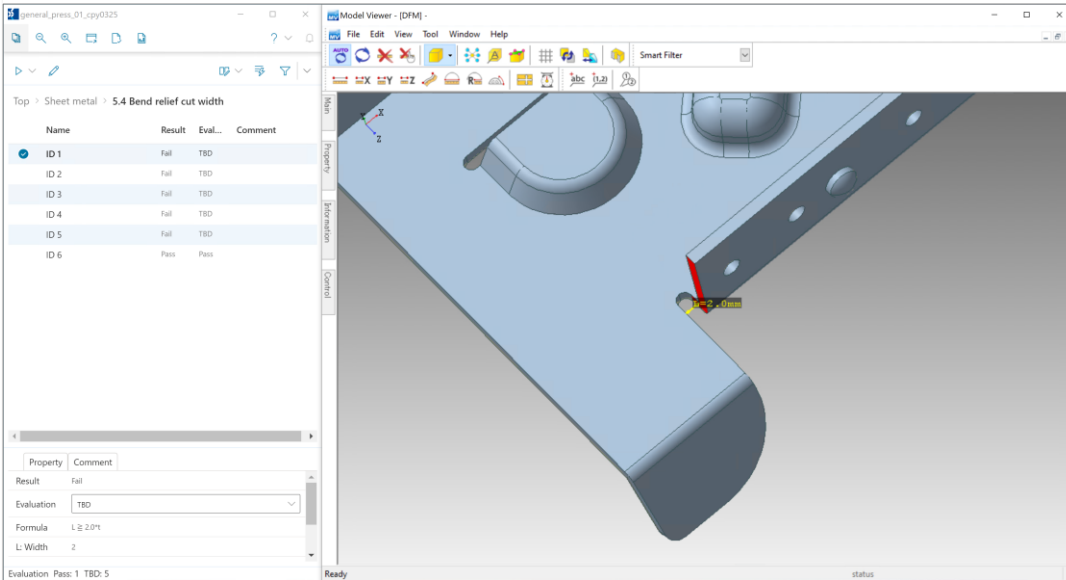
[A] When specifying by a fixed value with "Bend relief cut width | Min. width tol by fixed value" parameter:
BendingReliefWidth.Min=2.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Bend relief cut width | Min. width tol by ratio to sheet metal thickness" parameter: BendingReliefWidth.Min.Ratio=2.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend relief cuts whose width are greater than the toleranceRed: [Fail] Bend relief cuts whose width are smaller than the tolerance



5.4 Bend relief cut width

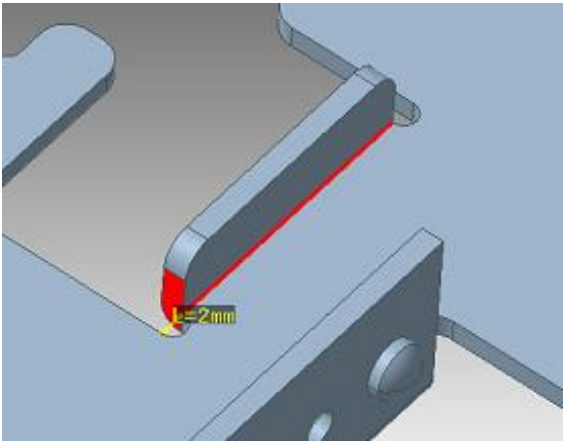


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

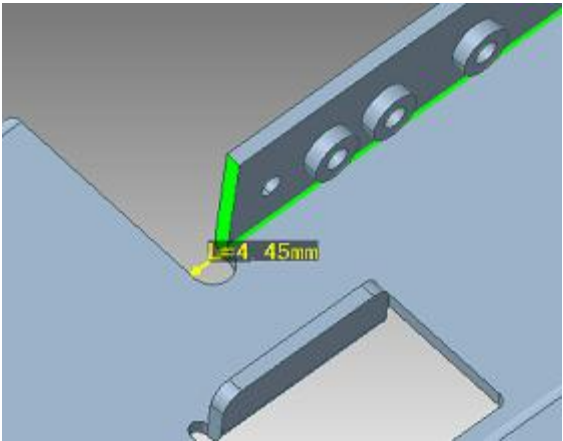
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Bend relief cuts whose width are greater than the toleranceRed: [Fail] Bend relief cuts whose width are smaller than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

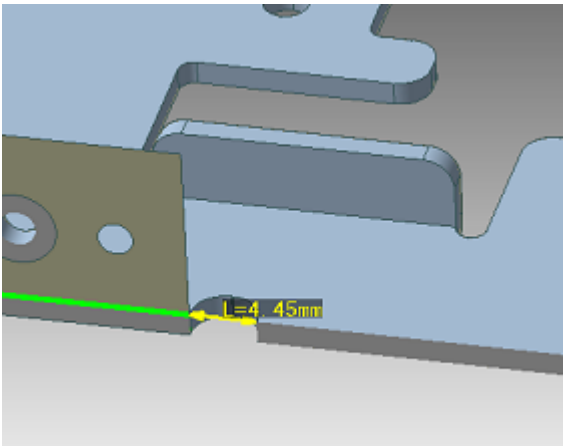
Highlight Checked Area - Fail



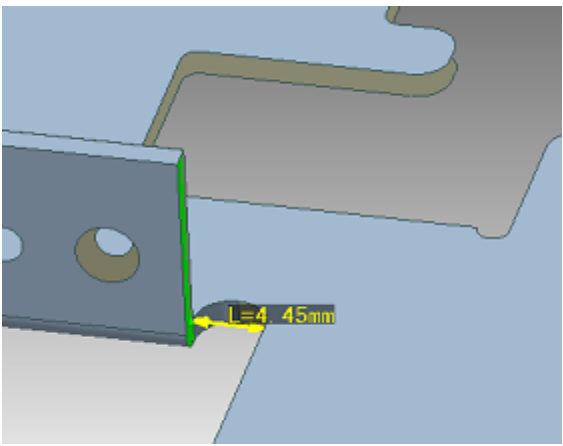
Highlight Checked Area - Pass



Section View



Hide Surroundings





Dowel



6.1 Dowel outer diameter and height

Description of Check Criterion & Possible Molding Defects

<Description>

Detects Dowel which does not follow the specified standards (diameter and height).

<Possible Molding Defects>



It may cause crack on the material when the dowel is not designed in conformity to the specified standards.

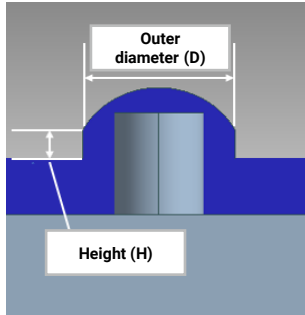
1. How to Check & Prerequisites

<How to Check>

Measures the outer diameter (D) and the height (H) of recognized Dowel, and then checks against the values in the specified table; first against "Diameter" to check whether the diameter matches one of the standards, second against "Min. height", and third against "Max. height" to check whether the height is within the range for the matched "Diameter".

<Prerequisites>

Geometry recognition is completed on "[Dowel](#)".



2. How to Judge & Default Value

<How to Judge>

Marks Dowel as Fail when its diameter does not match any of the standards, or its height is smaller than "Min. height" value, or greater than "Max. height" value.

* "Formula" property will be shown as "-" (hyphen)" when the diameter does not match any of the standards.

<Default Value>

Specify the sets of dowel diameter, min. height, and max. height.

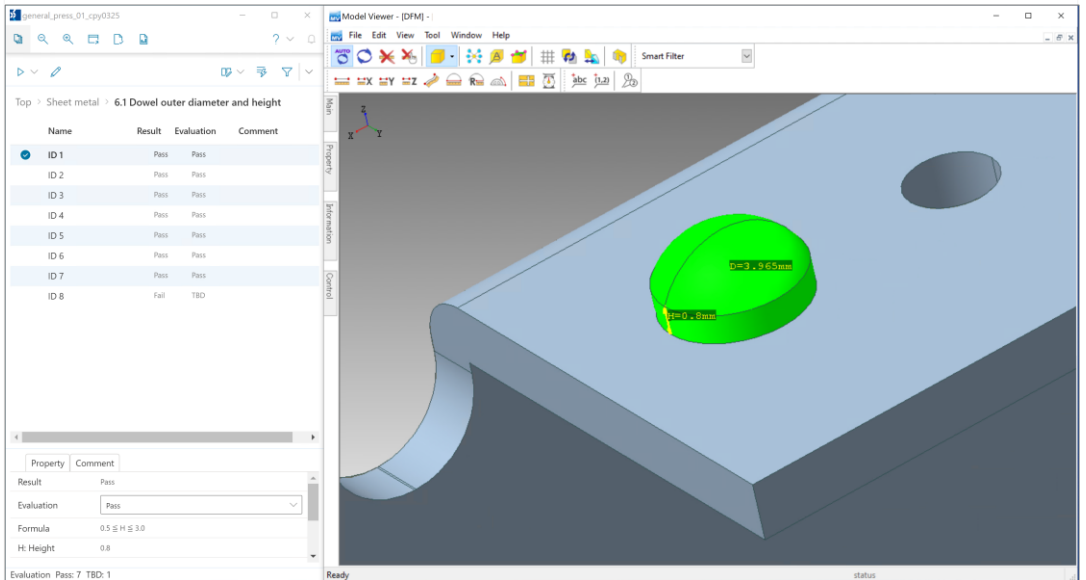
"Dowel outer diameter and height | Standard outer diameter and height sets by table" parameter:

DowelHeight.Table.String=line{3.0,0.5,2.0|4.0,0.5,3.0}

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Dowel whose diameter and height follows the standards Red: [Fail] Dowel whose diameter and/or height do not follow the standards





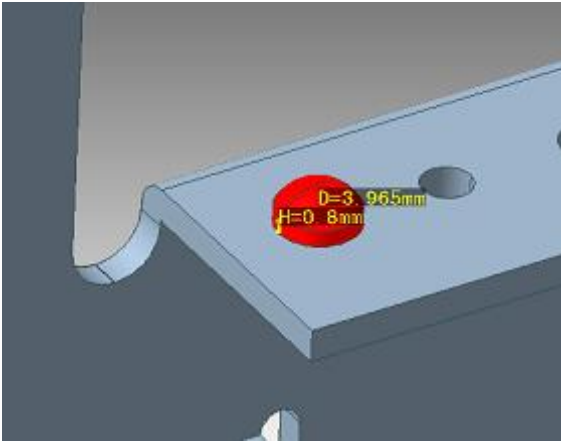
6.1 Dowel outer diameter and height

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

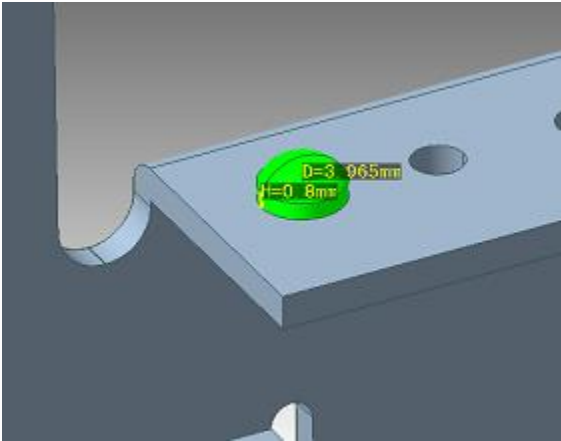
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel whose diameter and height follows the standardsRed: [Fail] Dowel whose diameter and/or height do not follow the standards
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

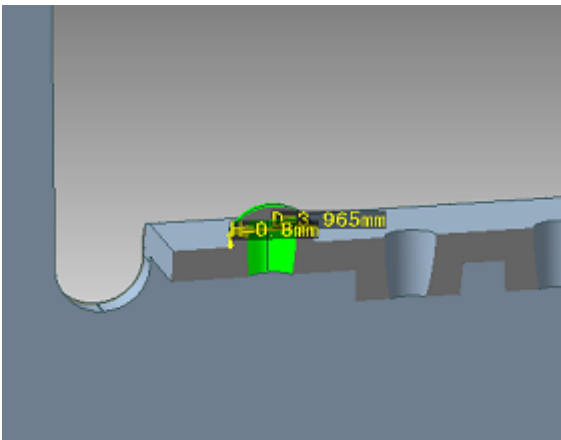
Highlight Checked Area - Fail



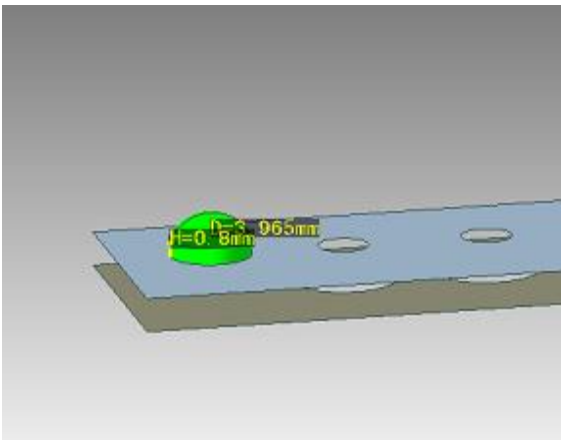
Highlight Checked Area - Pass



Section View



Hide Surroundings





6.2 Dowel: Distance to hole or part end face

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a dowel and a hole or a part end face is shorter than the specified tolerance.

<Possible Molding Defects>

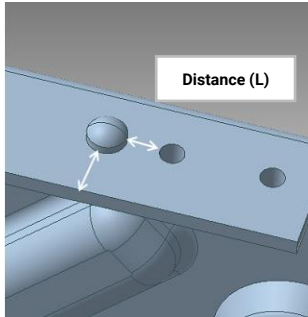


It may cause insufficient mold strength to make a dowel too close to a hole or a part end face. It is recommended to make dowel keeping an adequate distance from holes and part end faces.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a dowel and a hole / part end face is greater than the tolerance. Please note that measurement may be conducted at several points for a pair of a dowel and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.



<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", and "[Dowel](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

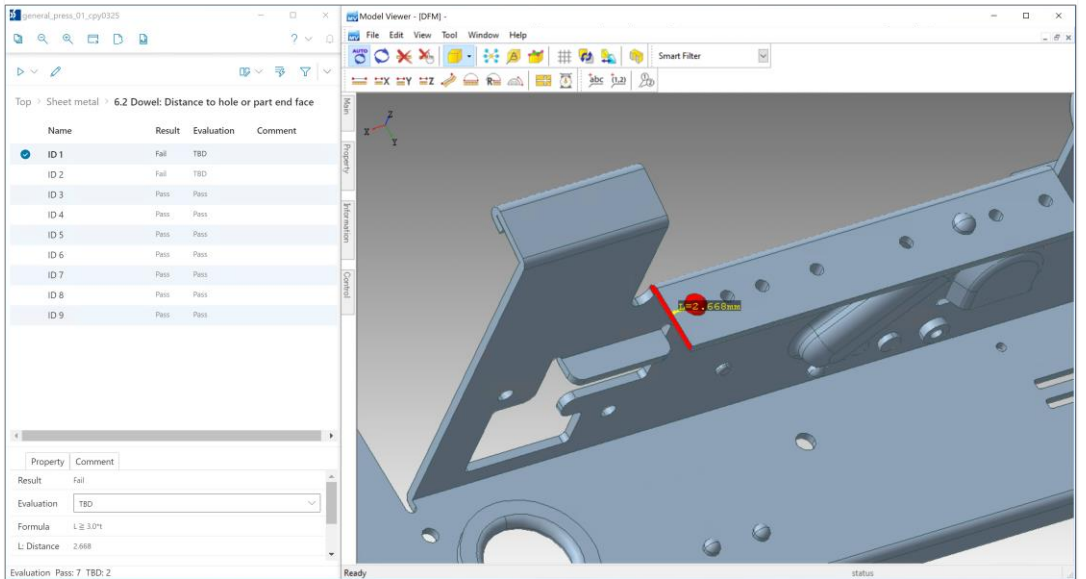
[A] When specifying by a fixed value with "Dowel: Distance to hole or part end face | Min. distance tol by fixed value" parameter: SideAndDowelDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Dowel: Distance to hole or part end face | Min. distance tol by ratio to sheet metal thickness" parameter: SideAndDowelDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Dowel and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Dowel and hole / part end face pair which the distance in between is shorter than the tolerance



6.2 Dowel: Distance to hole or part end face

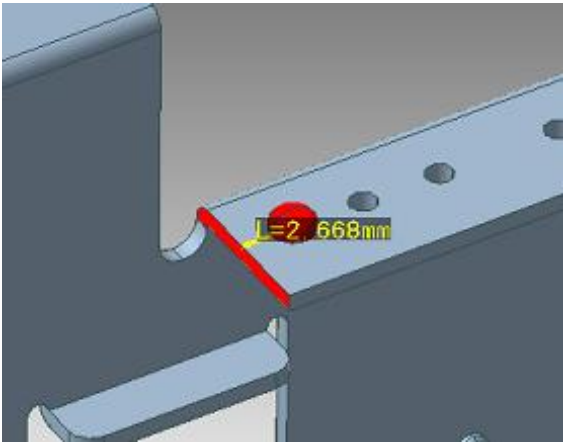


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

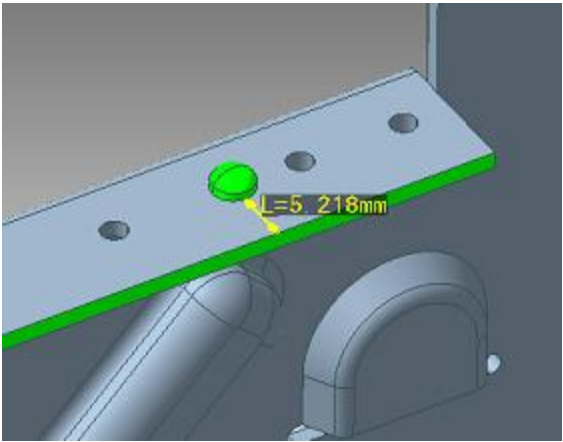
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel and hole / part end face pair which the distance in between is greater than the toleranceRed: [Fail] Dowel and hole / part end face pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

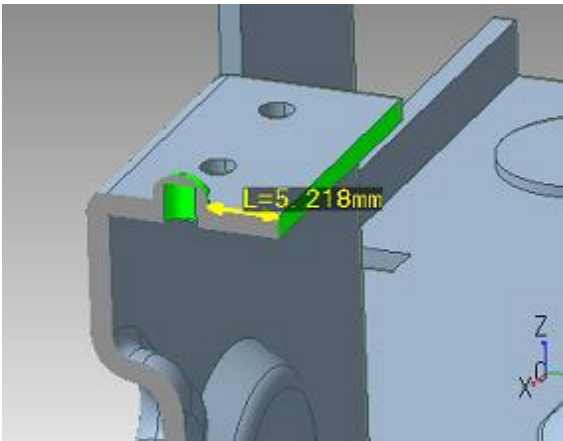
Highlight Checked Area - Fail



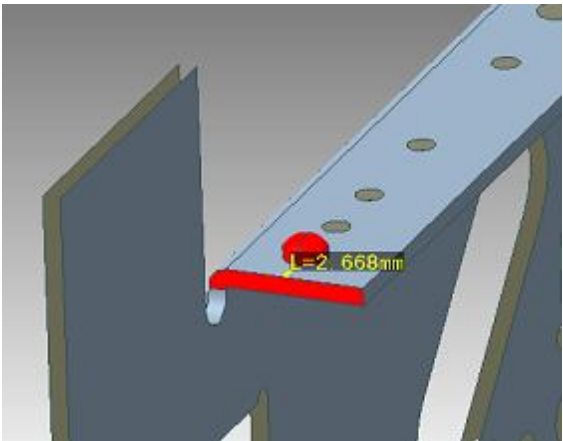
Highlight Checked Area - Pass



Section View



Hide Surroundings





6.3 Dowel: Distance to dowel

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a dowel and another dowel is shorter than the specified tolerance.

<Possible Molding Defects>

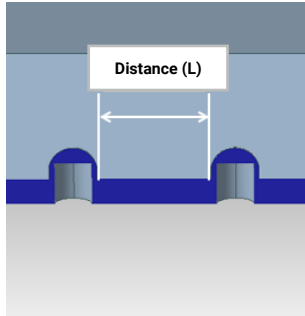


It may cause restrictions for machining methods, and/or a rise in the cost to make a dowel too close to another dowel. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a dowel and another dowel is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Dowel](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

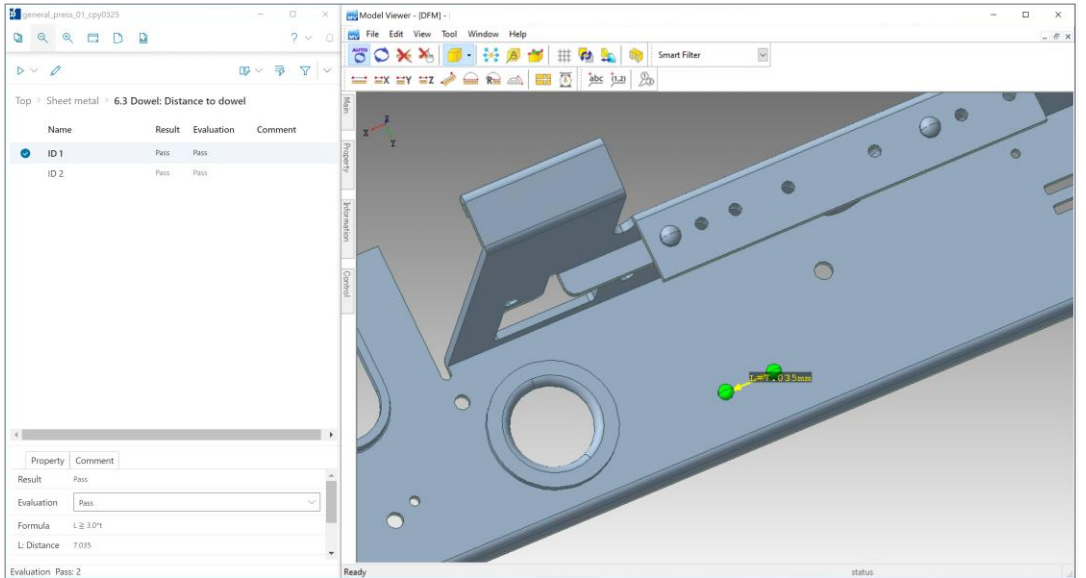
[A] When specifying by a fixed value with "Dowel: Distance to dowel | Min. distance tol by fixed value" parameter: DowelAndDowelDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Dowel: Distance to dowel | Min. distance tol by ratio to sheet metal thickness" parameter: DowelAndDowelDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel and another dowel pair which the distance in between is greater than the toleranceRed: [Fail] Dowel and another dowel pair which the distance in between is shorter than the tolerance





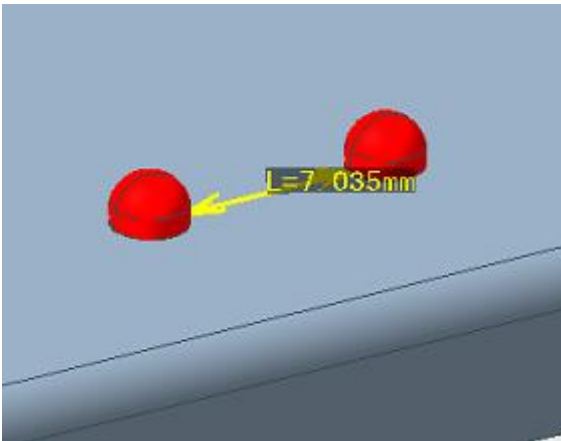
6.3 Dowel: Distance to dowel

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

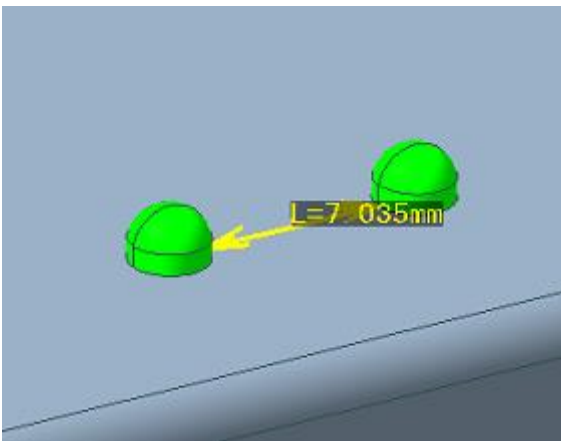
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Dowel and another dowel pair which the distance in between is greater than the tolerance Red: [Fail] Dowel and another dowel pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

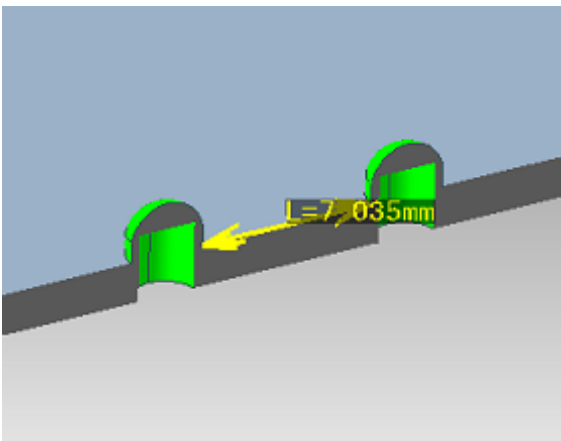
Highlight Checked Area - Fail



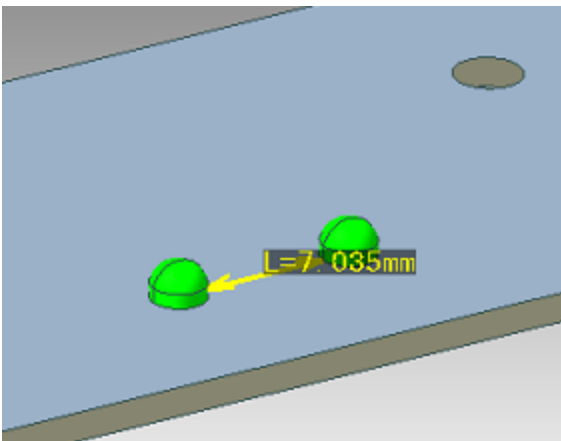
Highlight Checked Area - Pass

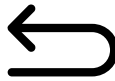


Section View



Hide Surroundings





6.4 Dowel: Distance to bend

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a dowel and an edge or a fillet on an inward/outward bend is shorter than the specified tolerance.

<Possible Molding Defects>

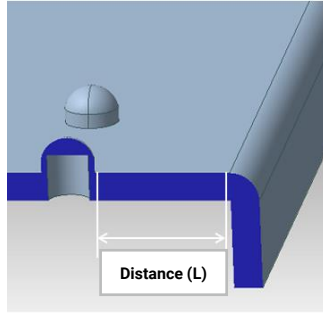


It may cause deformation in machining to make a dowel too close to a bend.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a dowel and an edge or a fillet on an inward/outward bend is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Dowel](#)", "[Simple bend](#)", and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

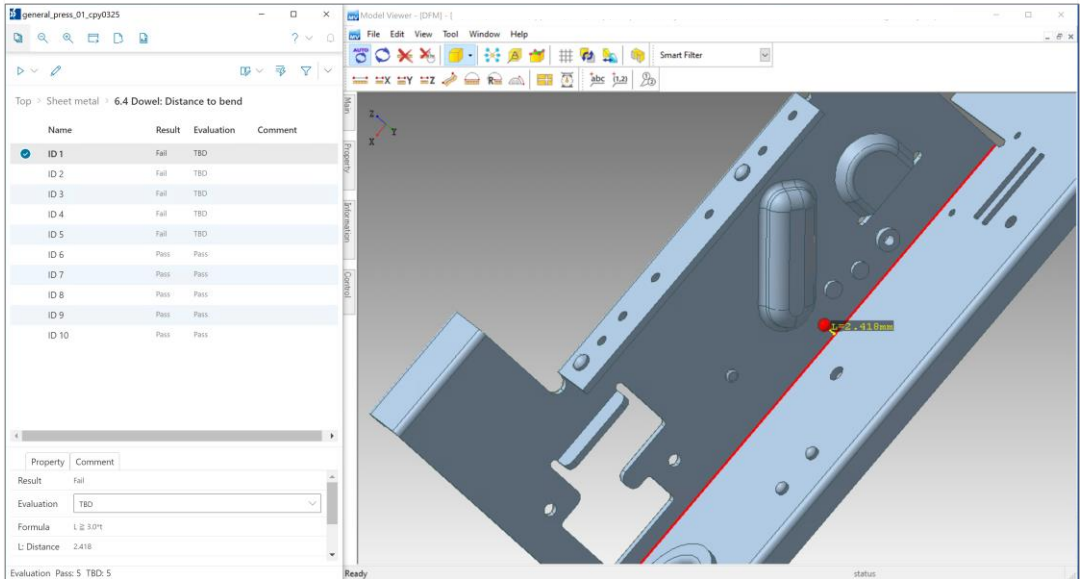
[A] When specifying by a fixed value with "Dowel: Distance to bend | Min. distance tol by fixed value" parameter: DowelAndBendingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Dowel: Distance to bend | Min. distance tol by ratio to sheet metal thickness" parameter: DowelAndBendingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Dowel and a bend pair which the distance in between is greater than the tolerance Red: [Fail] Dowel and a bend pair which the distance in between is shorter than the tolerance





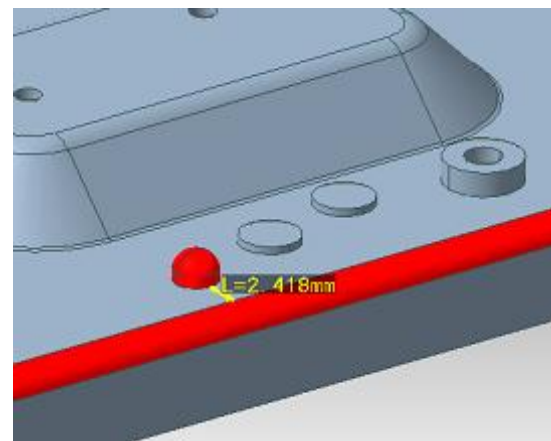
6.4 Dowel: Distance to bend

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

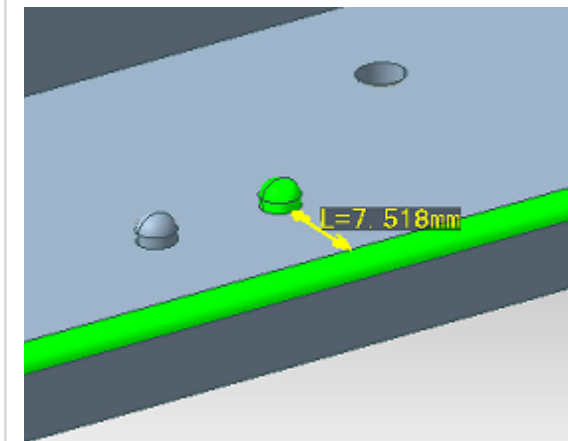
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel and a bend pair which the distance in between is greater than the toleranceRed: [Fail] Dowel and a bend pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

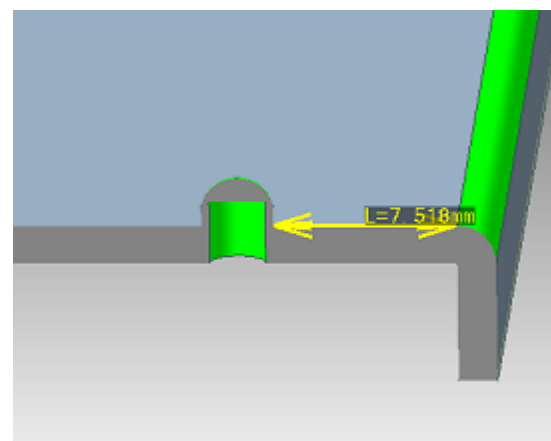
Highlight Checked Area - Fail



Highlight Checked Area - Pass

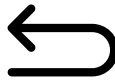


Section View



Hide Surroundings





6.5 Dowel: Distance to burring

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a dowel and a burring is shorter than the specified tolerance.

<Possible Molding Defects>

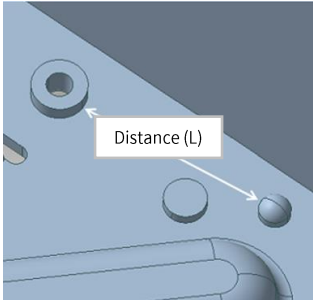


It may cause restrictions for machining methods, and/or a rise in the cost to make a dowel too close to a burring. It may also cause deformation in machining to make a dowel too close to a burring.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a dowel and a burring is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Dowel](#)" and "[Burring](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

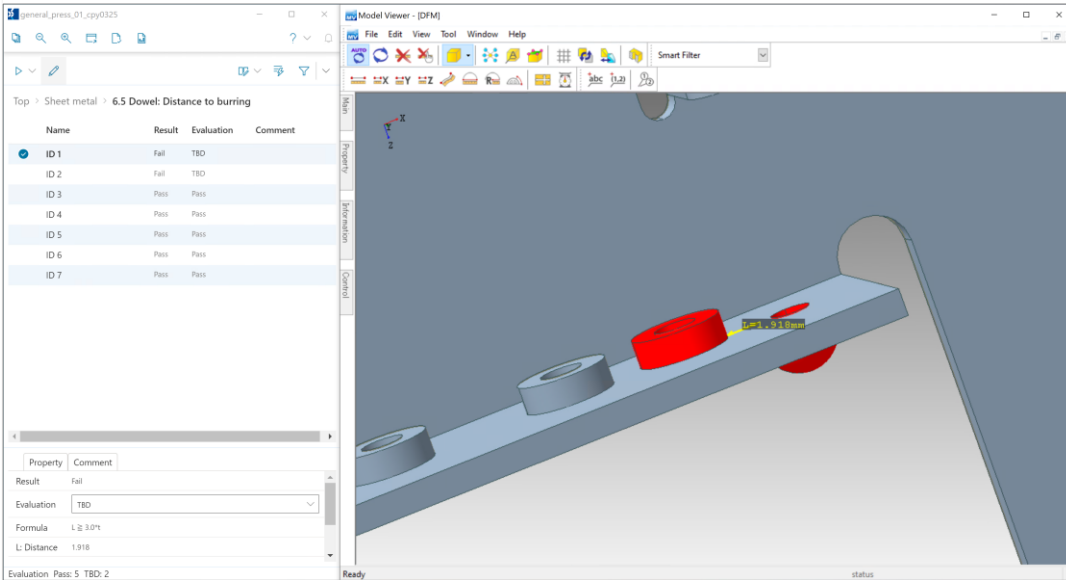
[A] When specifying by a fixed value with "Dowel: Distance to burring | Min. distance tol by fixed value" parameter: DowelAndBendingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Dowel: Distance to burring | Min. distance tol by ratio to sheet metal thickness" parameter: DowelAndBendingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel and a burring pair which the distance in between is greater than the toleranceRed: [Fail] Dowel and a burring pair which the distance in between is shorter than the tolerance





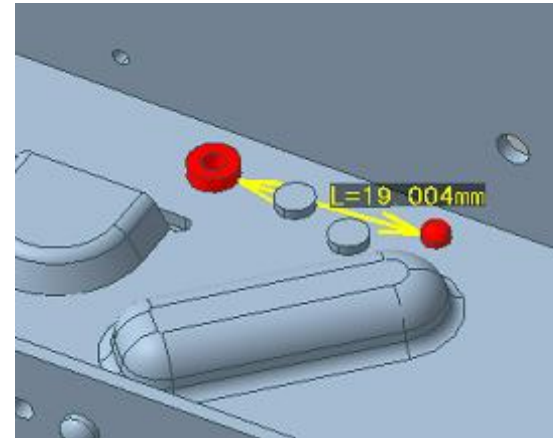
6.5 Dowel: Distance to burring

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

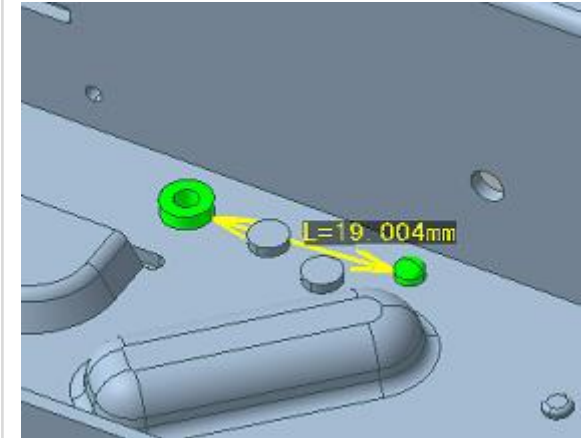
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Dowel and a burring pair which the distance in between is greater than the toleranceRed: [Fail] Dowel and a burring pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

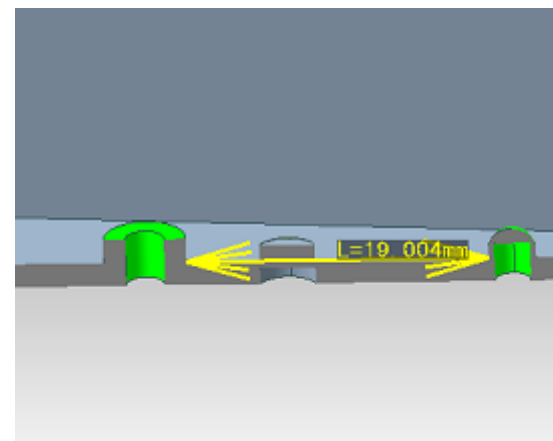
Highlight Checked Area - Fail



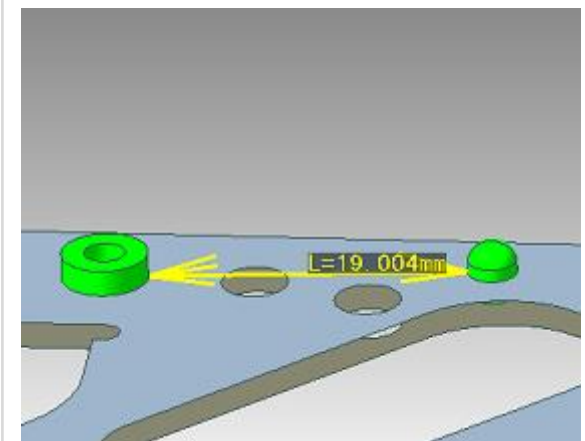
Highlight Checked Area - Pass



Section View



Hide Surroundings





Half-punching

7.1 Half-punching height



Description of Check Criterion & Possible Molding Defects

<Description>

Detects half-punching whose height is greater than the specified tolerance.

<Possible Molding Defects>

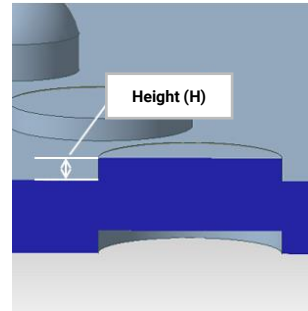


Half-punching may get damaged due to weighting when it is too high.

1. How to Check & Prerequisites

<How to Check>

Checks whether the height (H) of recognized half-punching is within the tolerance (Standard thickness multiplied by the "HalfPunchHeight.Max.Ratio").



<Prerequisites>

Geometry recognition is completed on "[Half-punching](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the half-punching as Fail when its height is greater than the tolerance.

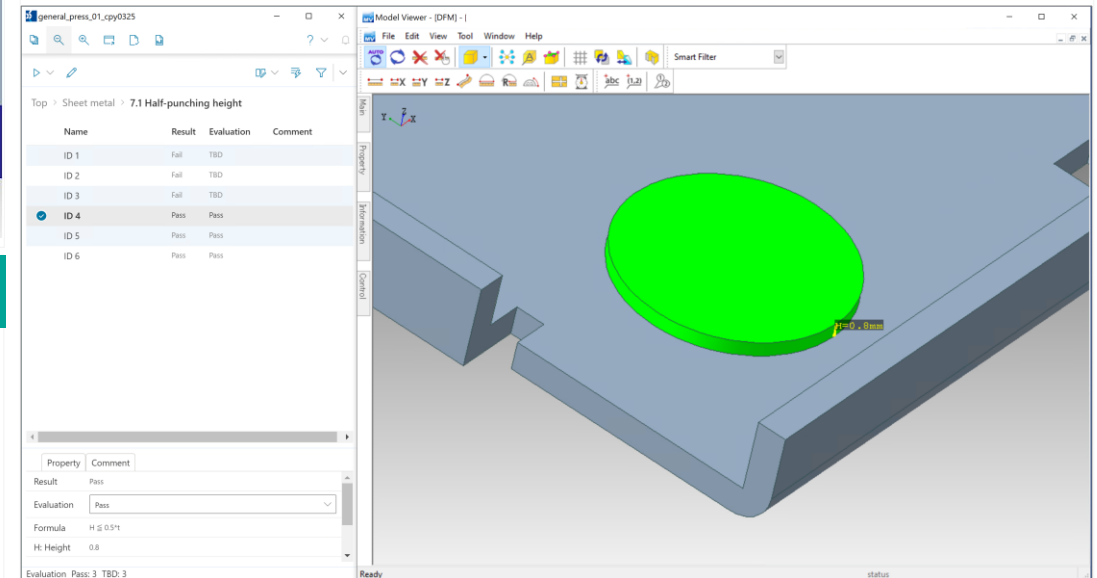
<Default Value>

"Half-punching height | Max. height tol by ratio to sheet metal thickness" parameter:
HalfPunchHeight.Max.Ratio=0.5

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Half-punching whose height is smaller than the tolerance Red: [Fail] Half-punching whose height is greater than the tolerance



7.1 Half-punching height

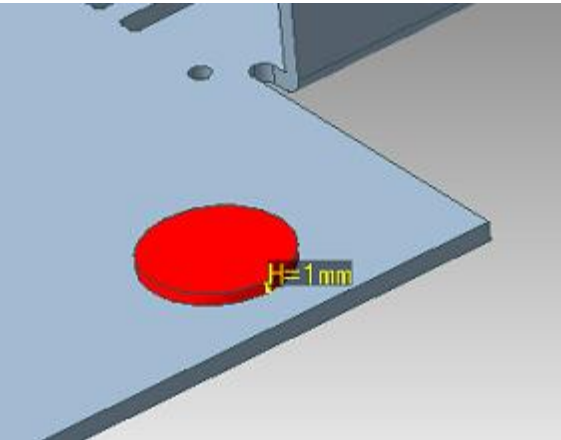


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

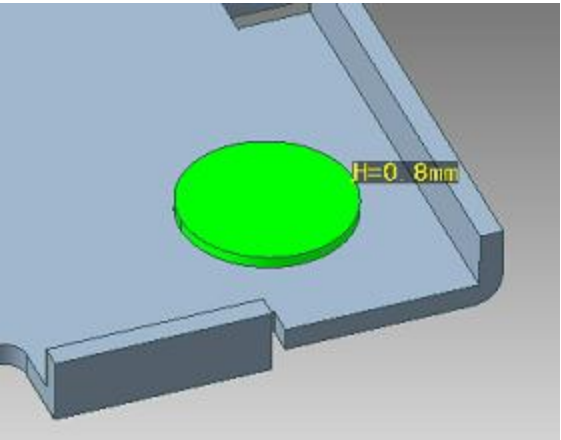
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching whose height is smaller than the toleranceRed: [Fail] Half-punching whose height is greater than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

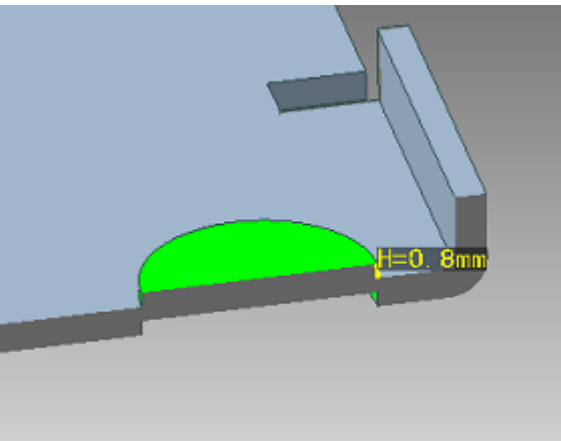
Highlight Checked Area - Fail



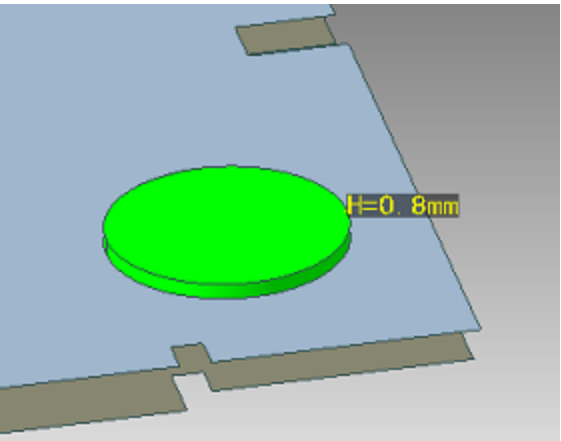
Highlight Checked Area - Pass



Section View



Hide Surroundings





7.2 Half-punching: Distance to hole or part end face

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a half-punching and a hole or a part end face is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause insufficient mold strength to make a half-punching too close to a hole or a part end face. It is recommended to make half-punching keeping an adequate distance from holes and part end faces.

1. How to Check & Prerequisites

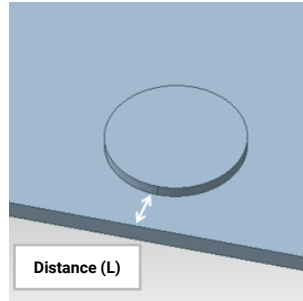
<How to Check>

Checks whether the distance (L) between a half-punching and a hole / part end face is greater than the tolerance.

Please note that measurement may be conducted at several points for a pair of a dowel and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", and "[Half-punching](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

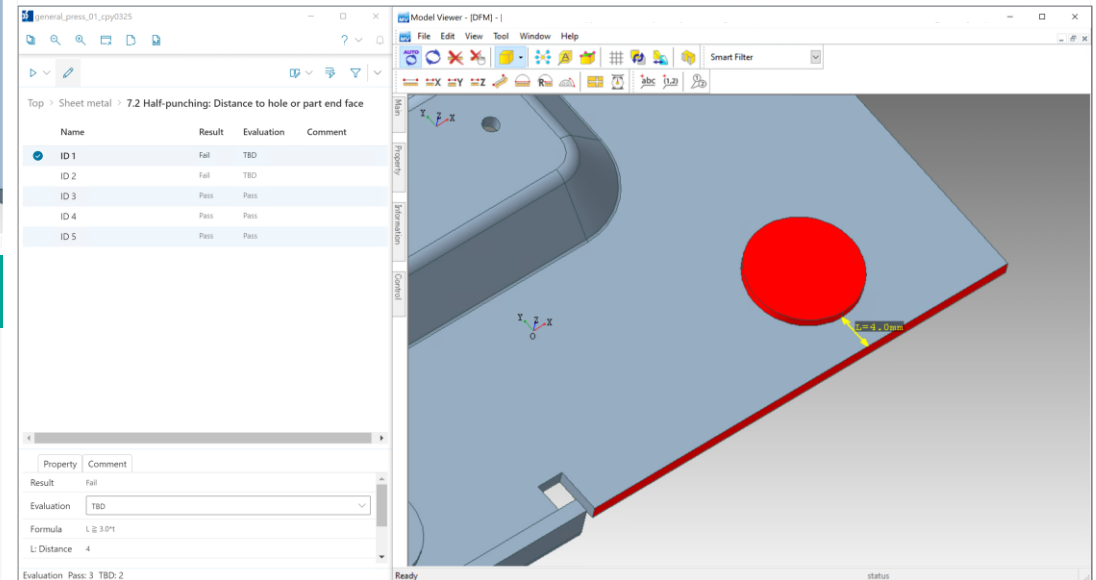
[A] When specifying by a fixed value with "Half-punching: Distance to hole or part end face | Min. distance tol by fixed value" parameter: SideAndHalfPunchDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Half-punching: Distance to hole or part end face | Min. distance tol by ratio to sheet metal thickness" parameter: SideAndHalfPunchDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Half-punching and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Half-punching and hole / part end face pair which the distance in between is shorter than the tolerance





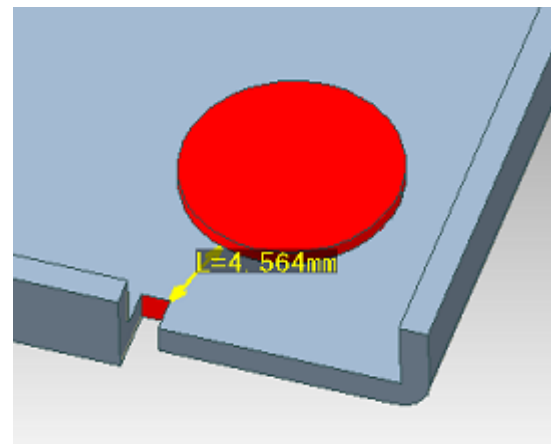
7.2 Half-punching: Distance to hole or part end face

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

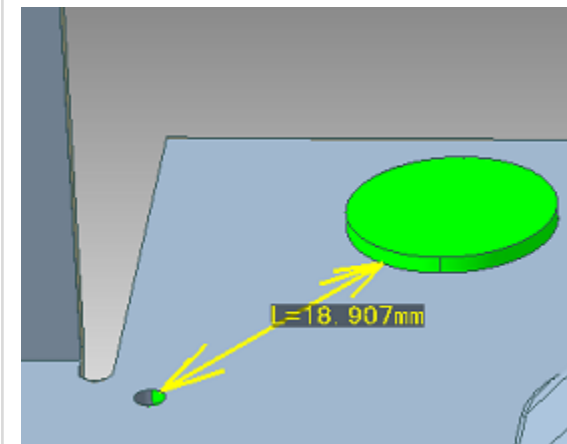
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Half-punching and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Half-punching and hole / part end face pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

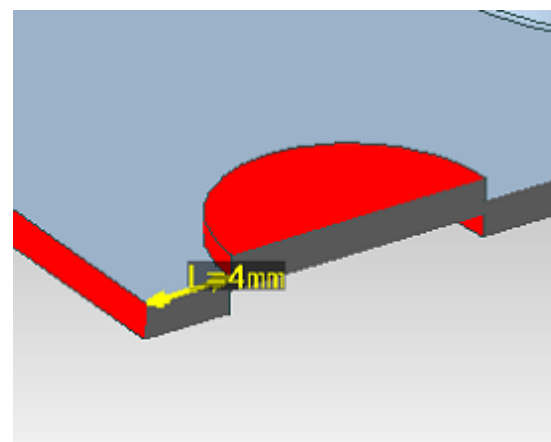
Highlight Checked Area - Fail



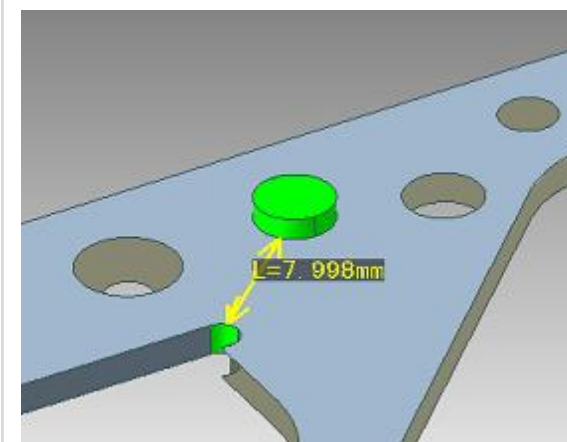
Highlight Checked Area - Pass



Section View



Hide Surroundings





7.3 Half-punching: Distance to burring

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a half-punching and a burring is shorter than the specified tolerance.

<Possible Molding Defects>

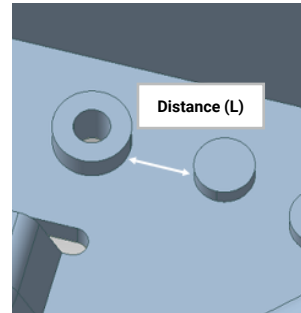


It may cause restrictions for machining methods, and/or a rise in the cost to make a half-punching too close to a burring. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a half-punching and a burring is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Burring](#)" and "[Half-punching](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

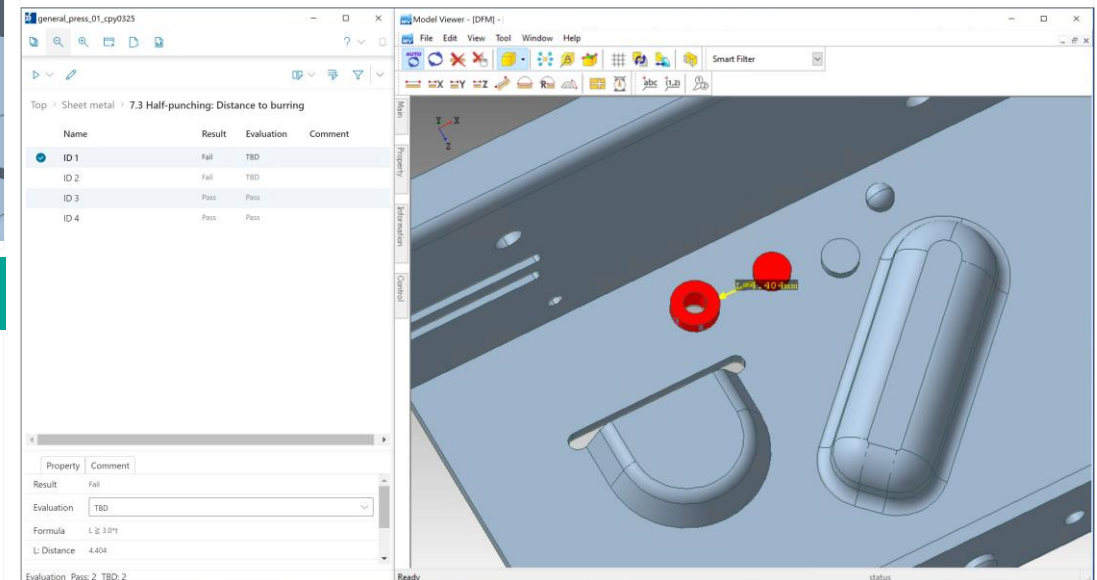
[A] When specifying by a fixed value with "Half-punching: Distance to burring | Min. distance tol by fixed value" parameter: HalfPunchAndBurringDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Half-punching: Distance to burring | Min. distance tol by ratio to sheet metal thickness" parameter: HalfPunchAndBurringDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Half-punching and burring pair which the distance in between is greater than the tolerance Red: [Fail] Half-punching and burring pair which the distance in between is shorter than the tolerance





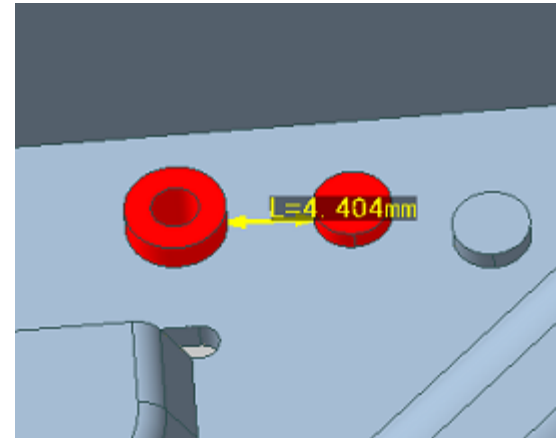
7.3 Half-punching: Distance to burring

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

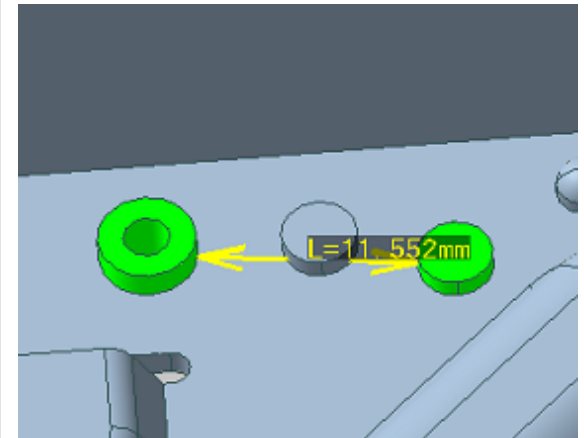
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and burring pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and burring pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

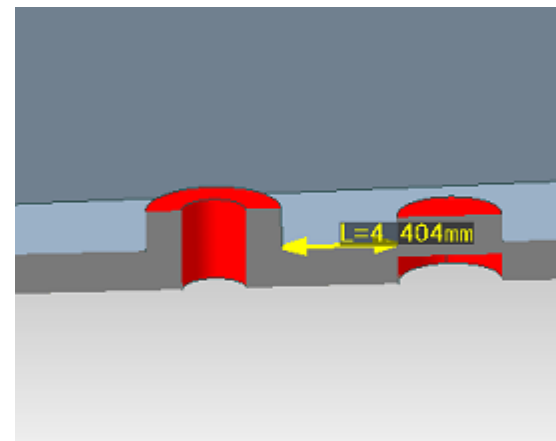
Highlight Checked Area - Fail



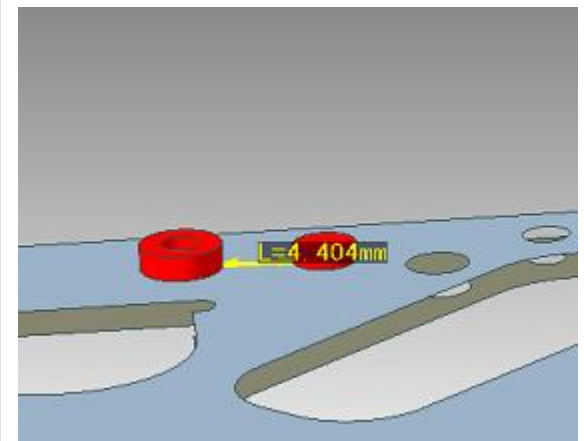
Highlight Checked Area - Pass

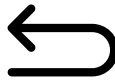


Section View



Hide Surroundings






7.4 Half-punching: Distance to half-punching

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a half-punching and another half-punching is shorter than the specified tolerance.

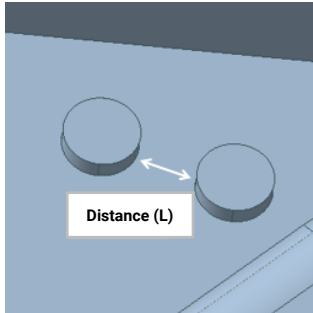
<Possible Molding Defects>

 It may cause restrictions for machining methods, and/or a rise in the cost to make a half-punching too close to another half-punching. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a half-punching and another half-punching is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Half-punching](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

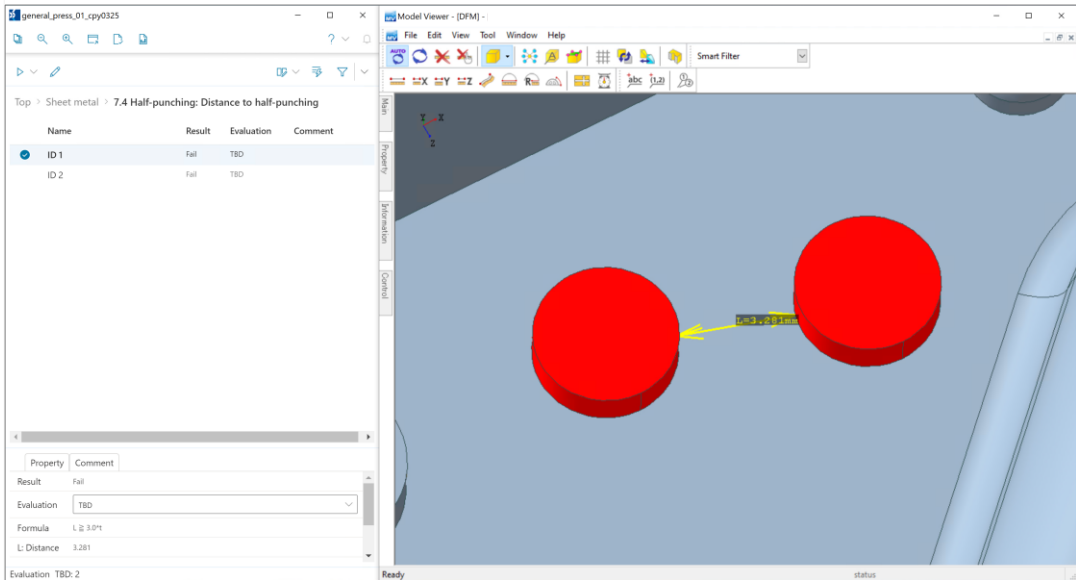
[A] When specifying by a fixed value with "Half-punching: Distance to half-punching | Min. distance tol by fixed value" parameter: HalfPunchAndHalfPunchDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Half-punching: Distance to half-punching | Min. distance tol by ratio to sheet metal thickness" parameter: HalfPunchAndHalfPunchDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and another half-punching pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and another half-punching pair which the distance in between is shorter than the tolerance





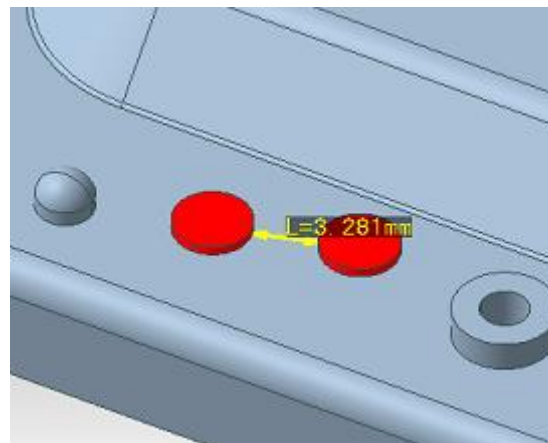
7.4 Half-punching: Distance to half-punching

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

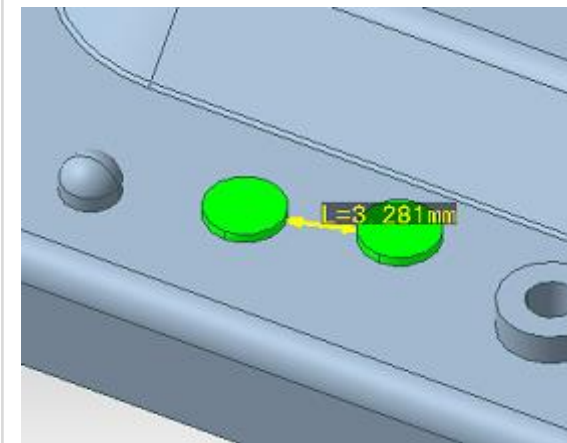
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and another half-punching pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and another half-punching pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

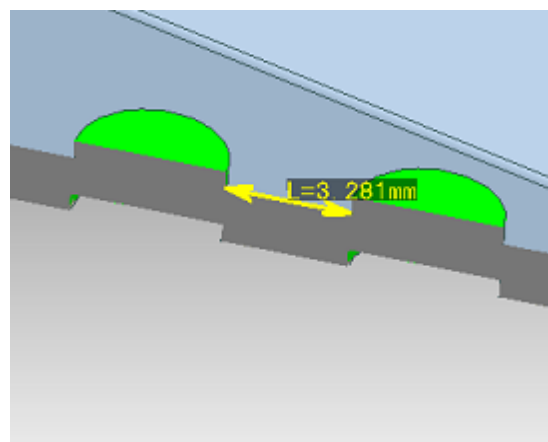
Highlight Checked Area - Fail



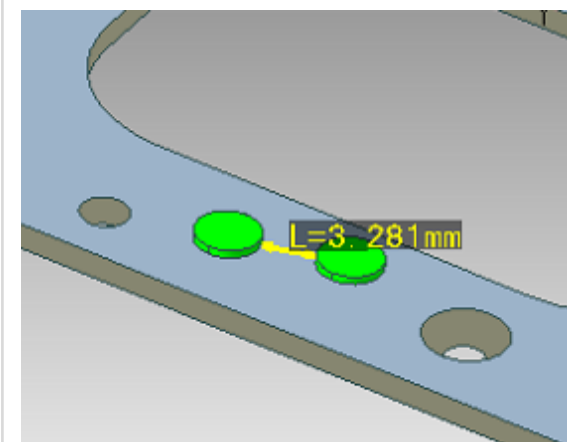
Highlight Checked Area - Pass



Section View



Hide Surroundings



7.5 Half-punching: Distance to bend



Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a half-punching and an edge or a fillet on an inward/outward bend is shorter than the specified tolerance.

<Possible Molding Defects>

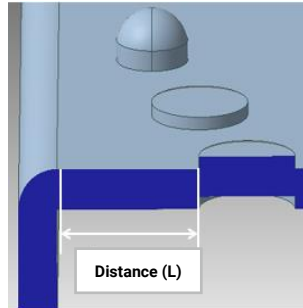


It may deform the half-punching to bend after half-punching the sheet metal. It is recommended to make bends keeping an adequate distance from half-punching.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a half-punching and an edge or a fillet on an inward/outward bend is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Half-punching](#)", "[Simple bend](#)", and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

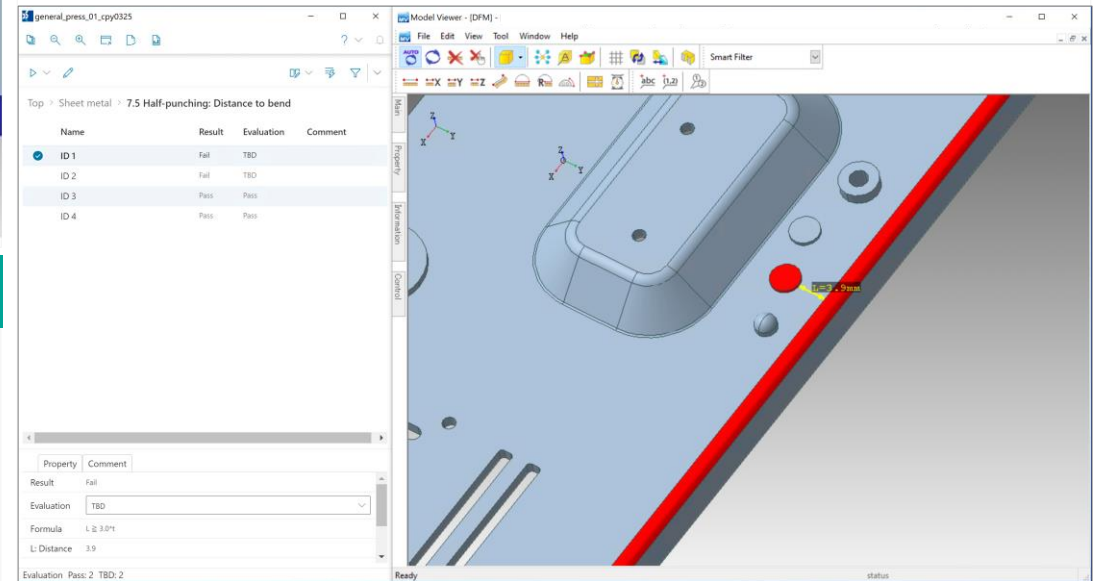
[A] When specifying by a fixed value with "Half-punching: Distance to bend | Min. distance tol by fixed value" parameter: BendingAndHalfPunchDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Half-punching: Distance to bend | Min. distance tol by ratio to sheet metal thickness" parameter: BendingAndHalfPunchDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Half-punching and bend pair which the distance in between is greater than the tolerance Red: [Fail] Half-punching and bend pair which the distance in between is shorter than the tolerance





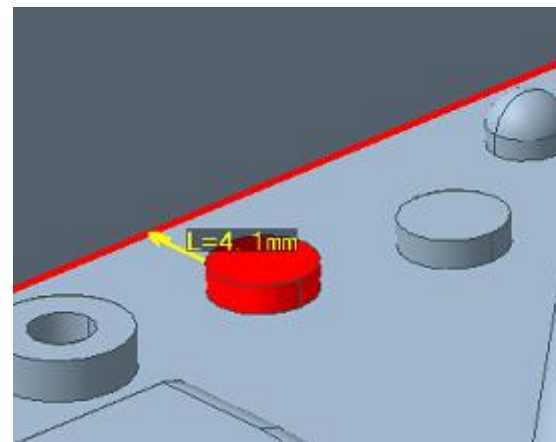
7.5 Half-punching: Distance to bend

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

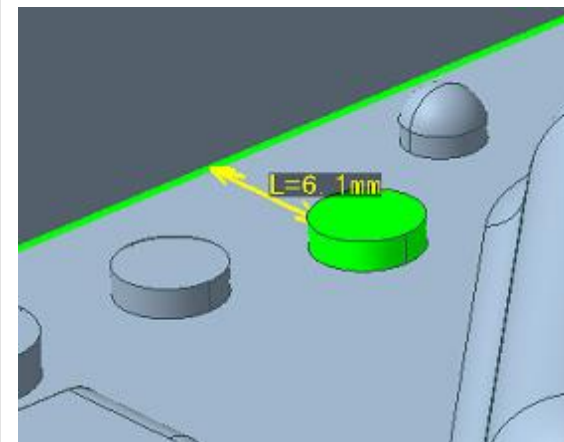
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and bend pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and bend pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

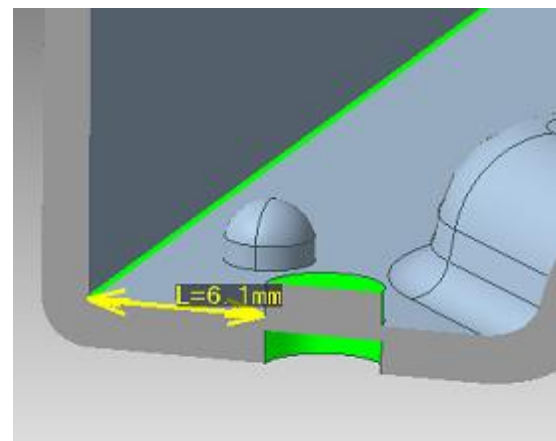
Highlight Checked Area - Fail



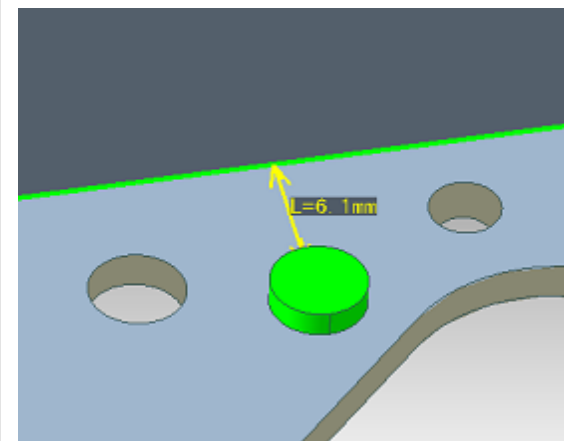
Highlight Checked Area - Pass



Section View



Hide Surroundings



7.6 Half-punching: Distance to dowel



Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a half-punching and a dowel is shorter than the specified tolerance.

<Possible Molding Defects>

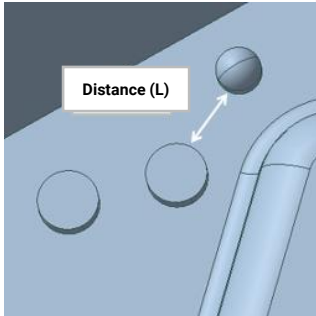


It may cause restrictions for machining methods, and/or a rise in the cost to make a half-punching too close to a dowel. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a half-punching and a dowel is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Half-punching](#)", "[Dowel](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

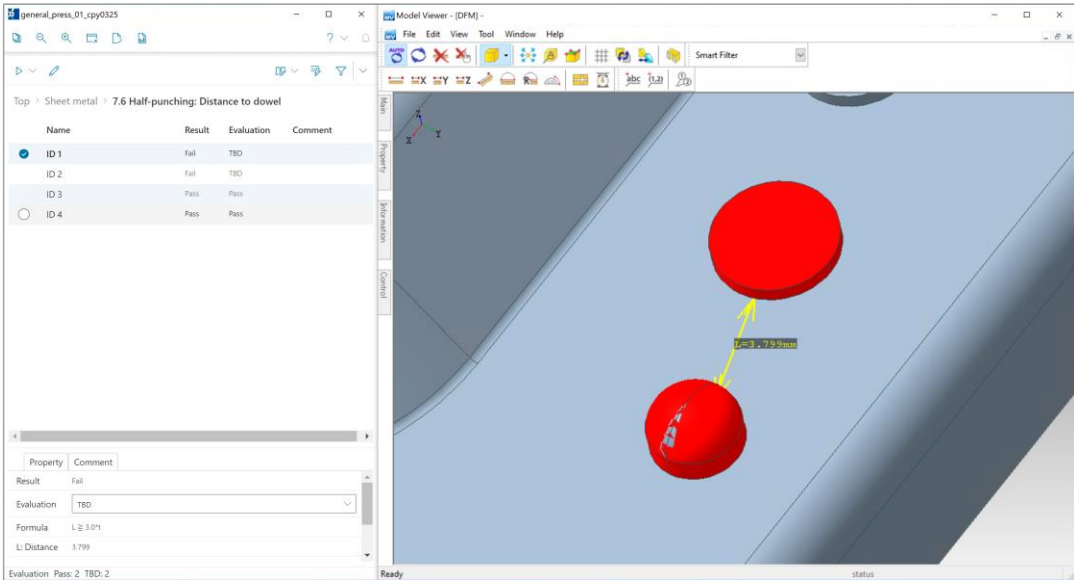
[A] When specifying by a fixed value with "Half-punching: Distance to dowel | Min. distance tol by fixed value" parameter: HalfPunchAndDowelDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Half-punching: Distance to dowel | Min. distance tol by ratio to sheet metal thickness" parameter: HalfPunchAndDowelDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and dowel pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and dowel pair which the distance in between is shorter than the tolerance



7.6 Half-punching: Distance to dowel

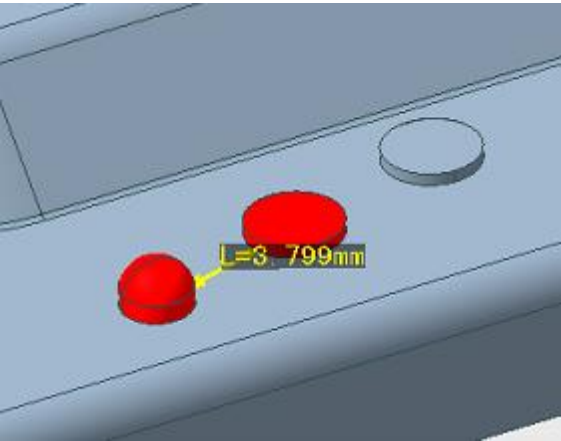


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

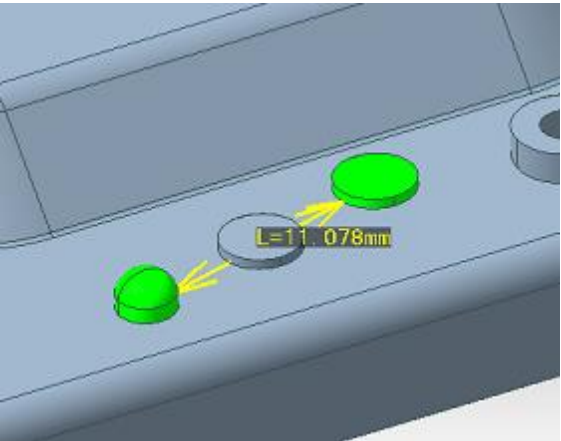
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Half-punching and dowel pair which the distance in between is greater than the toleranceRed: [Fail] Half-punching and dowel pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

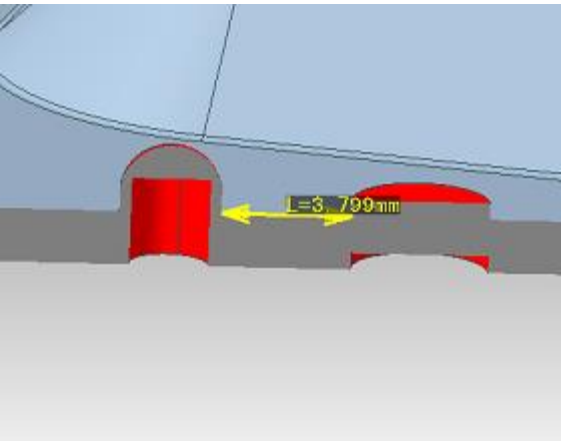
Highlight Checked Area - Fail



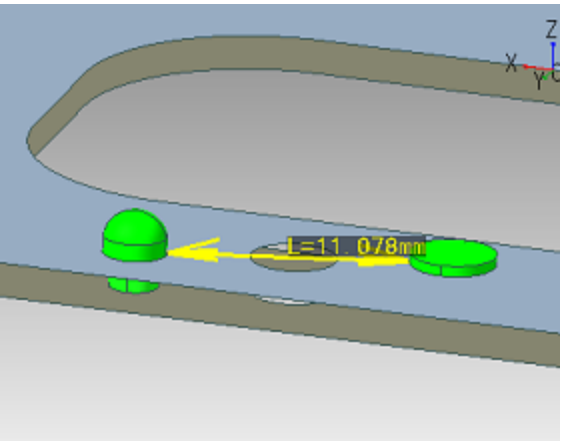
Highlight Checked Area - Pass



Section View



Hide Surroundings





Drawing



8.1 Drawing height

Description of Check Criterion & Possible Molding Defects

<Description>

Detects drawing whose height is greater than the specified tolerance.

<Possible Molding Defects>



It may cause the crack on the material when the drawing is too high.

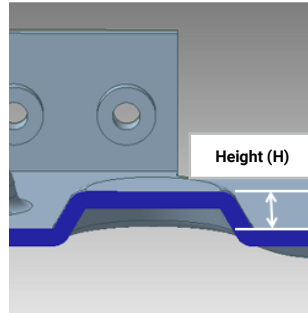
1. How to Check & Prerequisites

<How to Check>

Checks whether the height (H) of recognized drawing is smaller than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Drawing](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the height (H) is greater than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

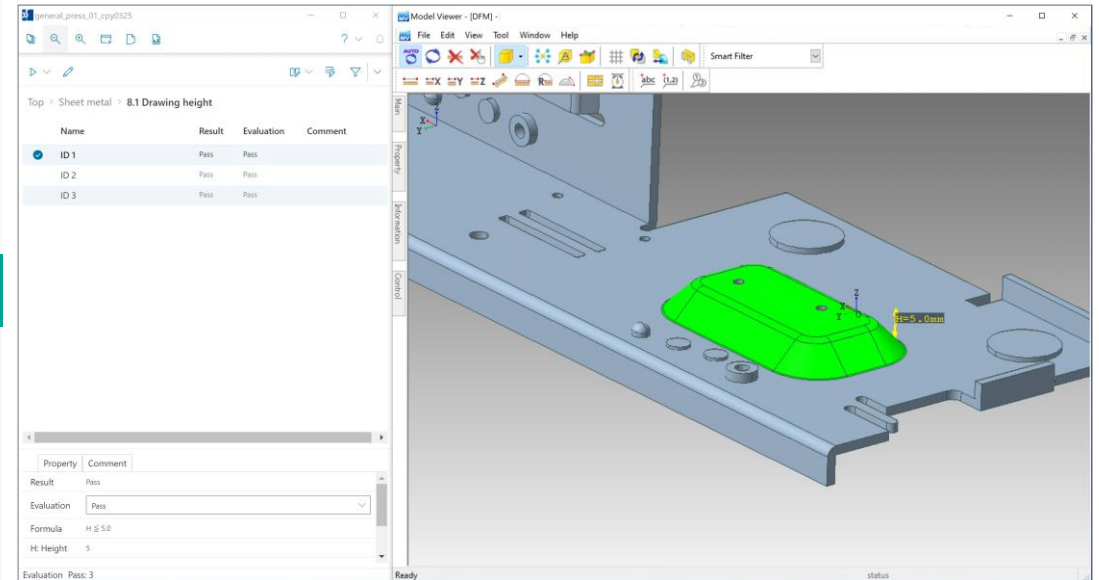
[A] When specifying by a fixed value with "Drawing height | Max. height tol by fixed value" parameter:
DrawingCompoundHeight.Max=5.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing height | Max. height tol by ratio to sheet metal thickness" parameter: DrawingCompoundHeight.Max.Ratio=5.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing whose height is smaller than the tolerance Red: [Fail] Drawing whose height is greater than the tolerance



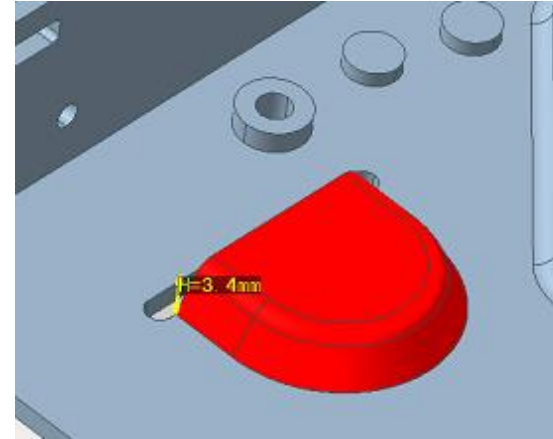
8.1 Drawing height

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

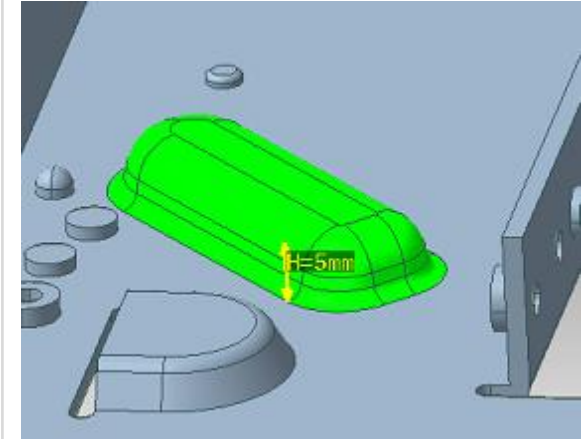
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing whose height is smaller than the toleranceRed: [Fail] Drawing whose height is greater than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

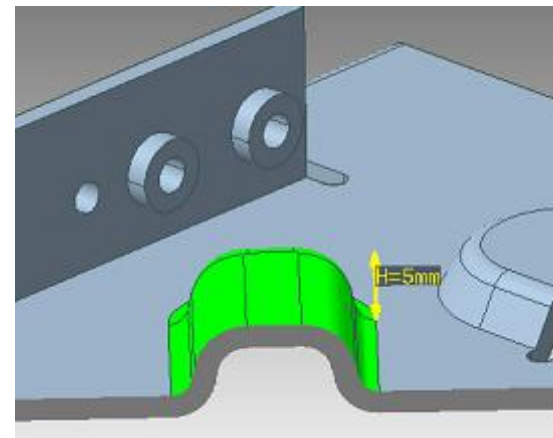
Highlight Checked Area - Fail



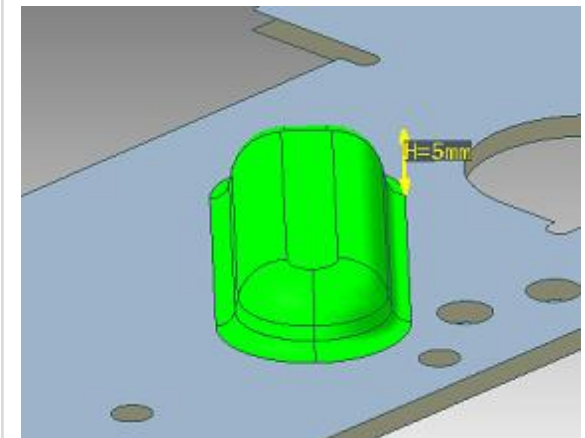
Highlight Checked Area - Pass



Section View



Hide Surroundings





8.2 Drawing: Distance to hole or part end face

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and a hole or a part end face is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause insufficient mold strength to make a drawing too close to a hole or a part end face. It is recommended to make drawing keeping an adequate distance from holes and part end faces.

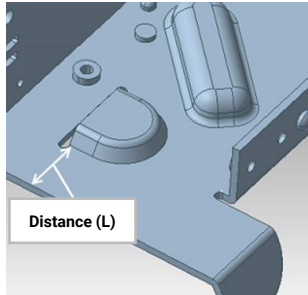
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and a hole / part end face is greater than the tolerance. Please note that measurement may be conducted at several points for a pair of a dowel and a hole / part end face, and only the shortest distance will be used as the check result even when the measurement result contains two or more Fail results.

<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)", "[Punching hole](#)", and "[Drawing](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance. A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

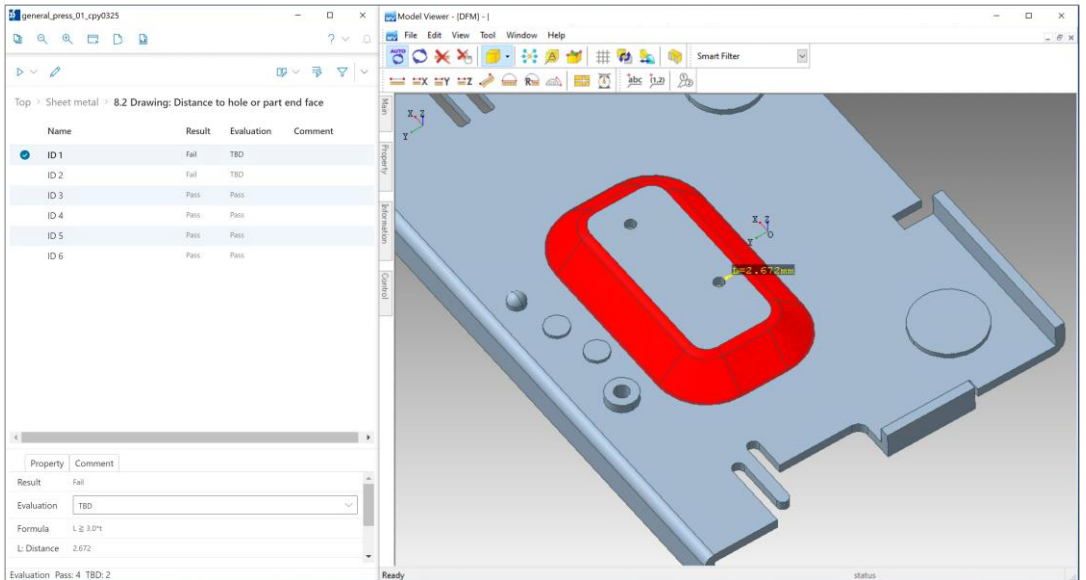
[A] When specifying by a fixed value with "Drawing: Distance to hole or part end face | Min. distance tol by fixed value" parameter: SideAndDrawingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to hole or part end face | Min. distance tol by ratio to sheet metal thickness" parameter: SideAndDrawingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing and hole / part end face pair which the distance in between is greater than the tolerance Red: [Fail] Drawing and hole / part end face pair which the distance in between is shorter than the tolerance





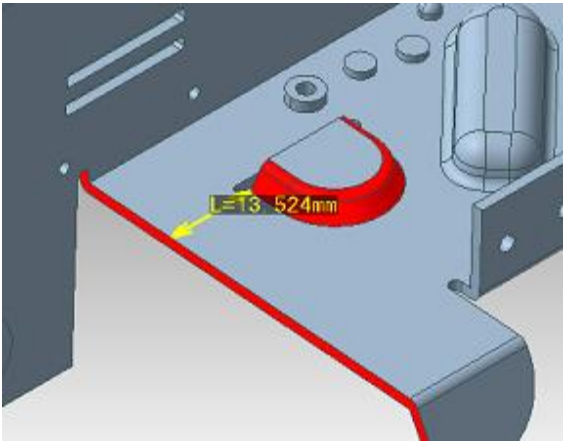
8.2 Drawing: Distance to hole or part end face

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

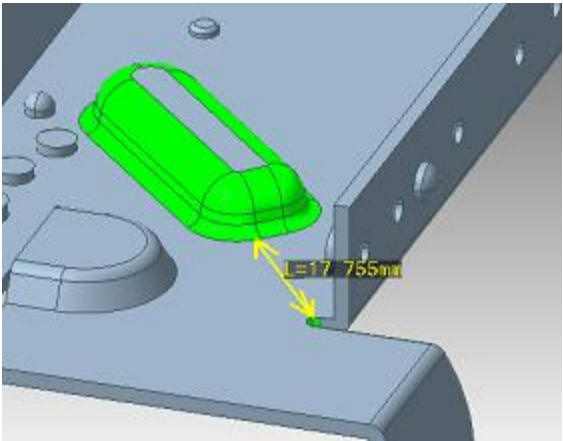
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and hole / part end face pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and hole / part end face pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

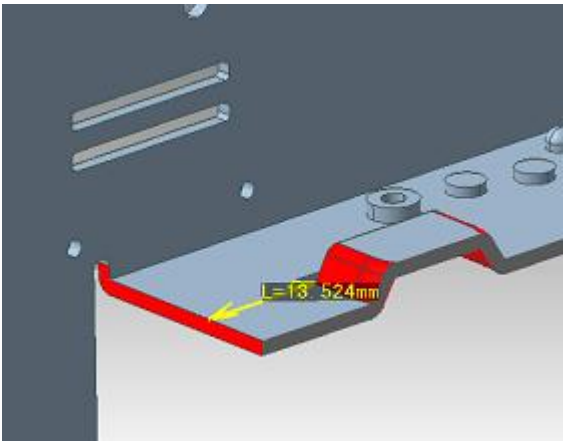
Highlight Checked Area - Fail



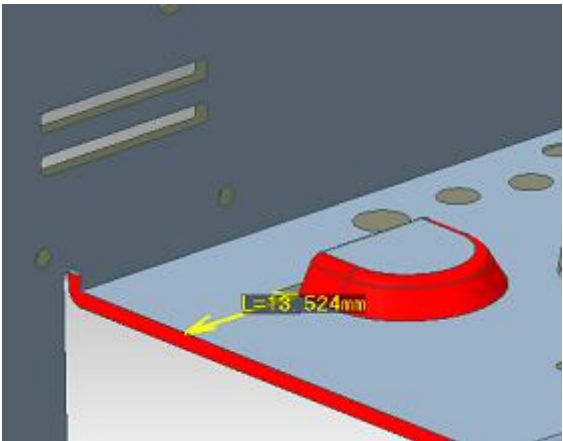
Highlight Checked Area - Pass



Section View



Hide Surroundings





8.3 Drawing: Distance to burring

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and a burring is shorter than the specified tolerance.

<Possible Molding Defects>

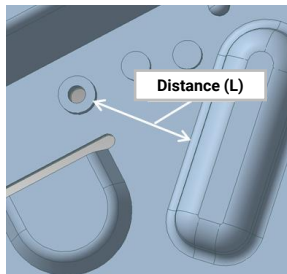


It may cause restrictions for machining methods, and/or a rise in the cost to make a drawing too close to a burring. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and a burring is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Burring](#)" and "[Drawing](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

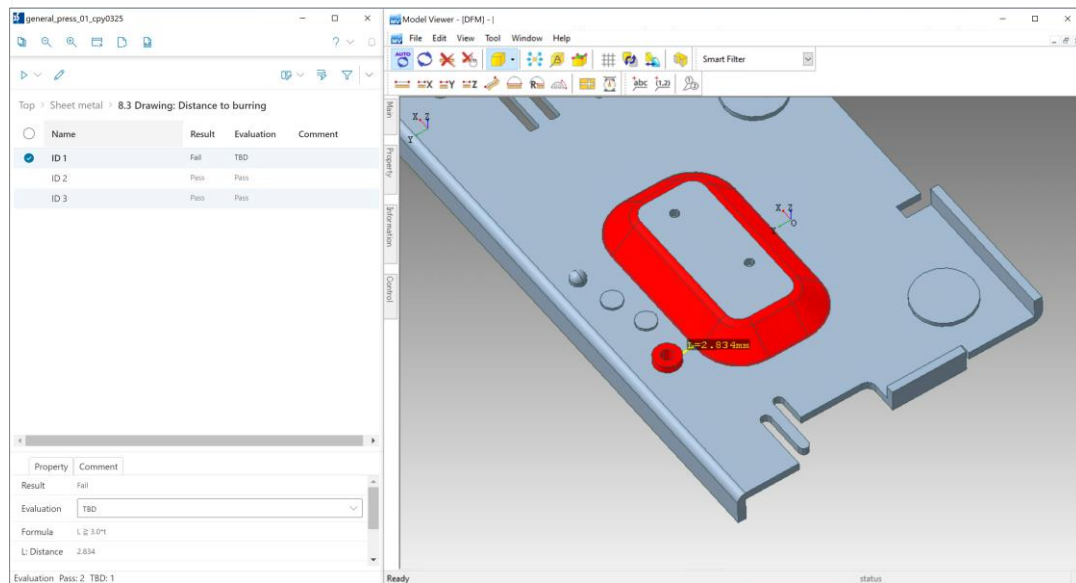
[A] When specifying by a fixed value with "Drawing: Distance to burring | Min. distance tol by fixed value" parameter: DrawingAndBurringDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to burring | Min. distance tol by ratio to sheet metal thickness" parameter: DrawingAndBurringDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing and burring pair which the distance in between is greater than the tolerance Red: [Fail] Drawing and burring pair which the distance in between is shorter than the tolerance



8.3 Drawing: Distance to burring

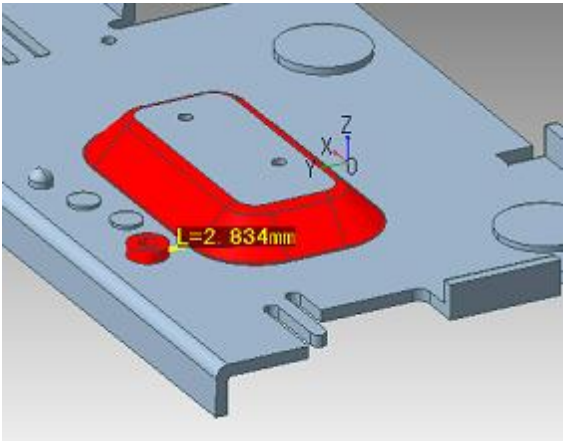


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

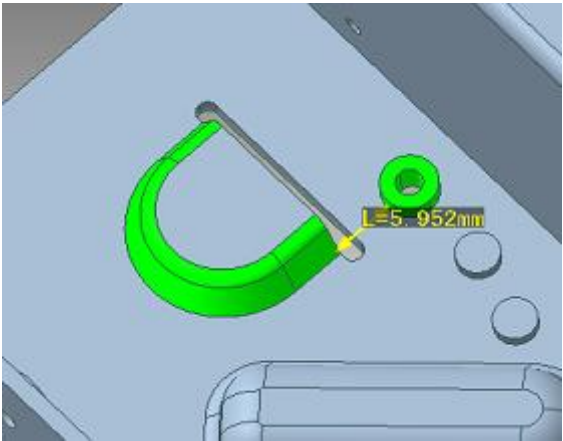
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and burring pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and burring pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

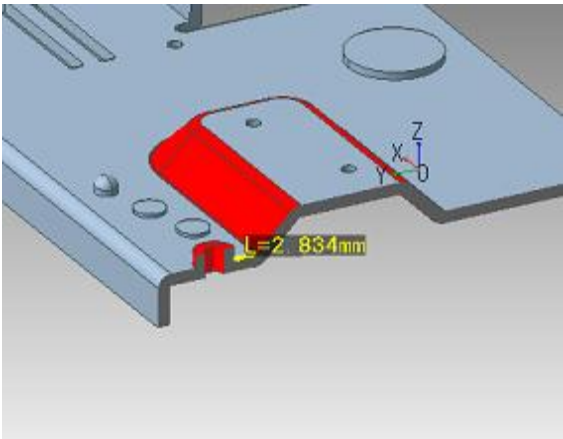
Highlight Checked Area - Fail



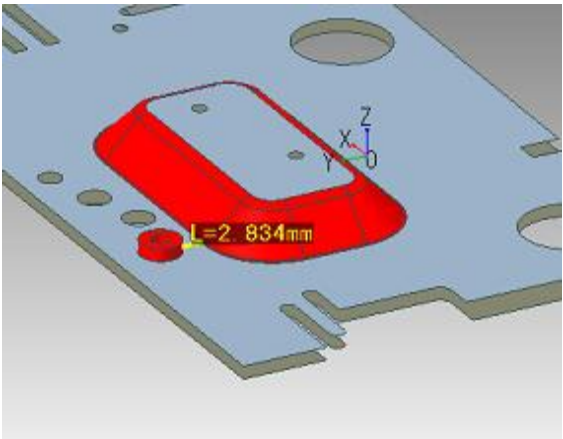
Highlight Checked Area - Pass



Section View



Hide Surroundings





8.4 Drawing: Distance to bend

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and an edge or a fillet on an inward/outward bend is shorter than the specified tolerance.

<Possible Molding Defects>

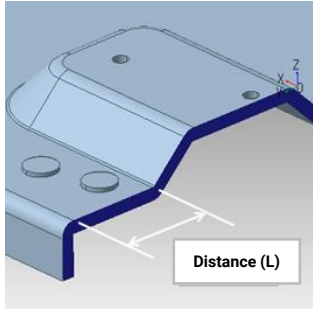


It may deform the drawing to bend after drawing the sheet metal.
It is recommended to make bends keeping an adequate distance from drawing.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and an edge or a fillet on an inward/outward bend is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Drawing](#)", "[Simple bend](#)", and "[Hem bend](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

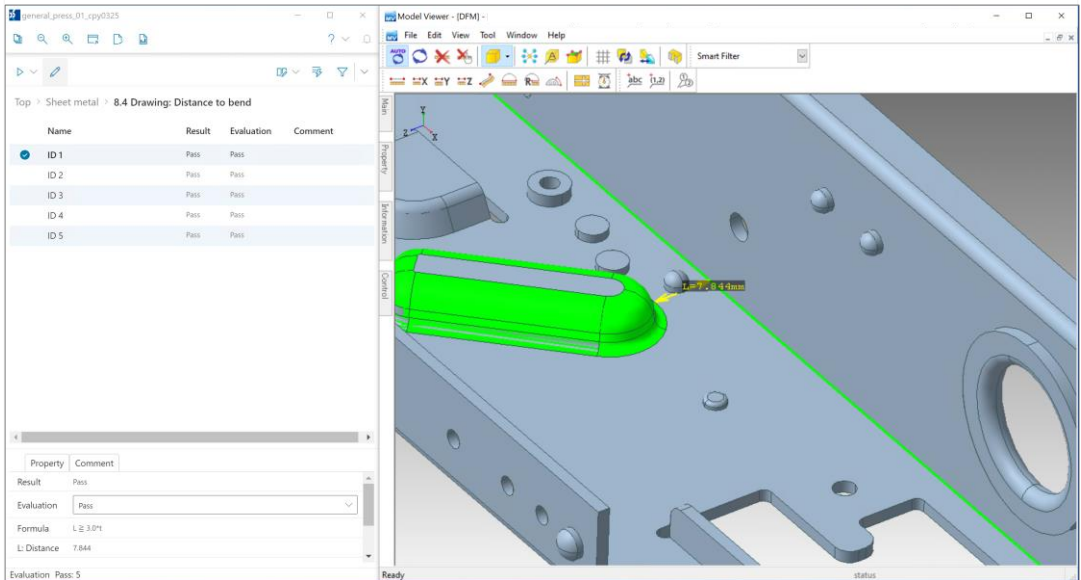
[A] When specifying by a fixed value with "Drawing: Distance to bend | Min. distance tol by fixed value" parameter: BendingAndDrawingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to bend | Min. distance tol by ratio to sheet metal thickness" parameter: BendingAndDrawingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing and bend pair which the distance in between is greater than the tolerance Red: [Fail] Drawing and bend pair which the distance in between is shorter than the tolerance



8.4 Drawing: Distance to bend

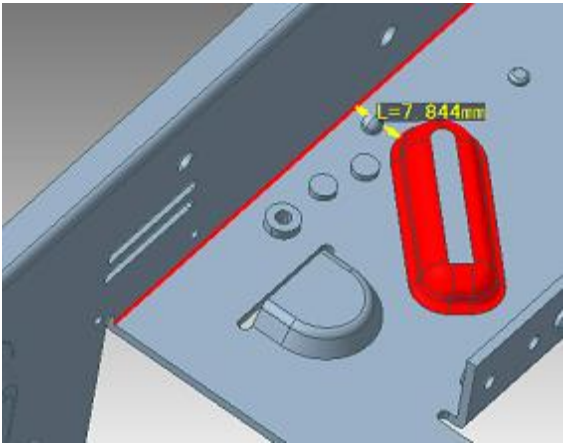


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

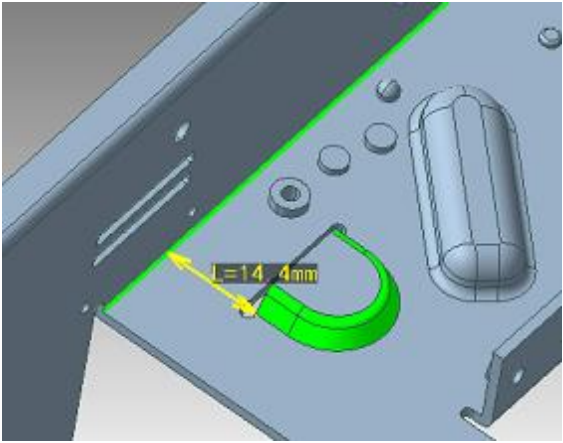
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and bend pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and bend pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

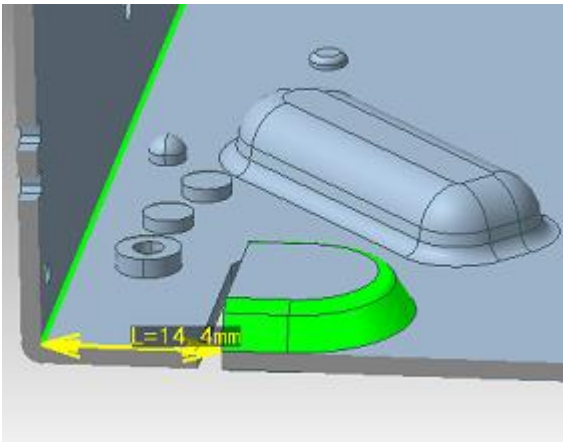
Highlight Checked Area - Fail



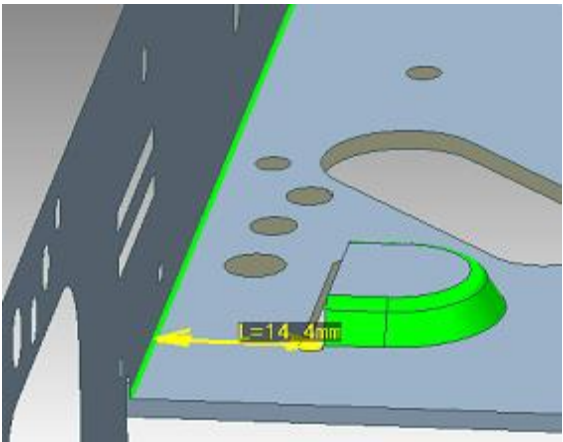
Highlight Checked Area - Pass



Section View



Hide Surroundings





8.5 Drawing: Distance to dowel

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and a dowel is shorter than the specified tolerance.

<Possible Molding Defects>

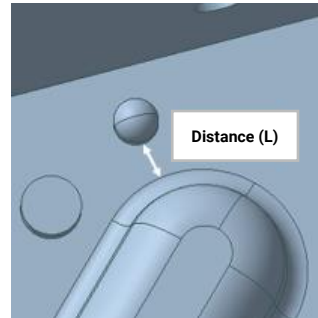


It may cause restrictions for machining methods, and/or a rise in the cost to make a drawing too close to a dowel. It may also cause deformation in machining.

1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and a dowel is greater than the tolerance.



<Prerequisites>

Geometry recognition is completed on "[Dowel](#)" and "[Drawing](#)".

2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.

A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

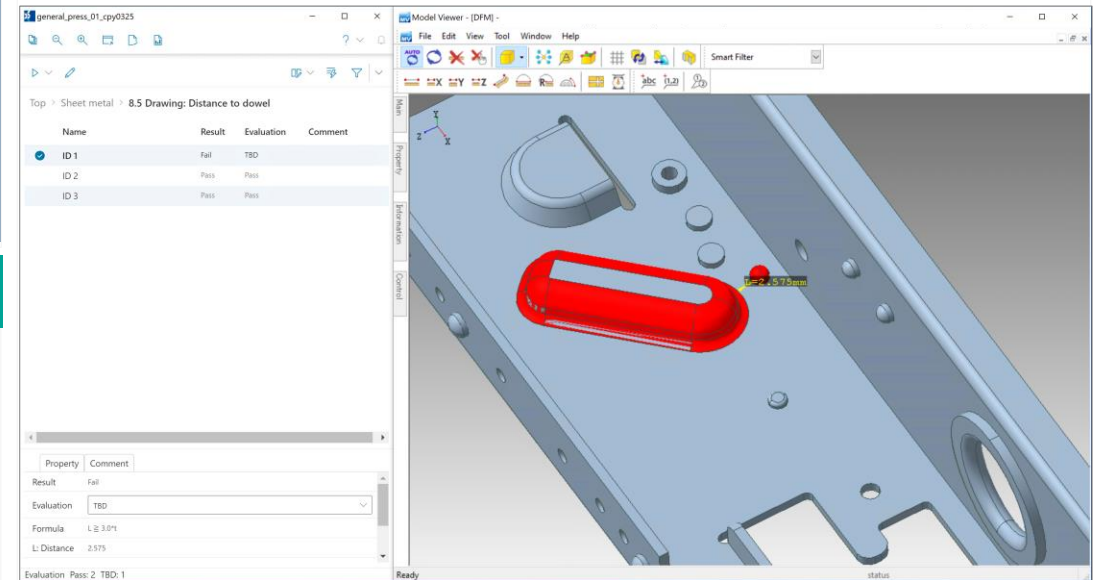
[A] When specifying by a fixed value with "Drawing: Distance to dowel | Min. distance tol by fixed value" parameter: DrawingAndDowelDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to dowel | Min. distance tol by ratio to sheet metal thickness" parameter: DrawingAndDowelDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and dowel pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and dowel pair which the distance in between is shorter than the tolerance



8.5 Drawing: Distance to dowel

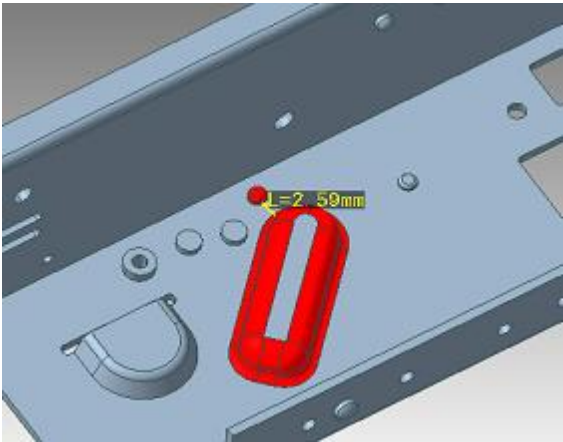


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

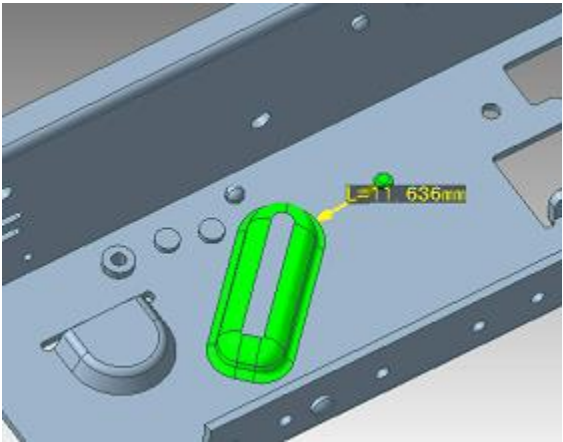
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and dowel pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and dowel pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

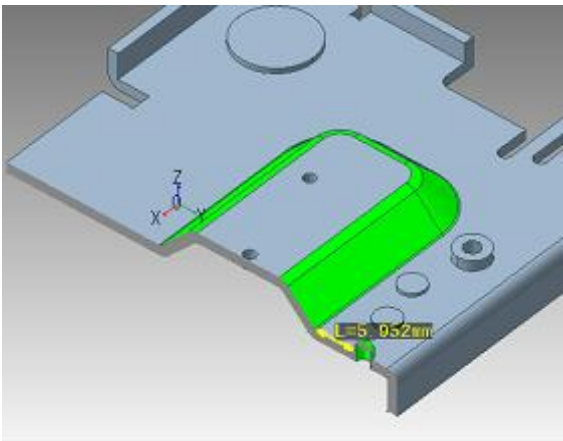
Highlight Checked Area - Fail



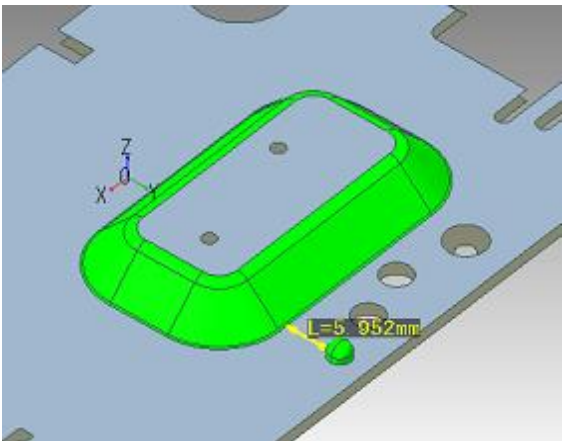
Highlight Checked Area - Pass



Section View



Hide Surroundings






8.6 Drawing: Distance to half-punching

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and a dowel is shorter than the specified tolerance.

<Possible Molding Defects>

 It may cause restrictions for machining methods, and/or a rise in the cost to make a drawing too close to a half-punching. It may also cause deformation in machining.

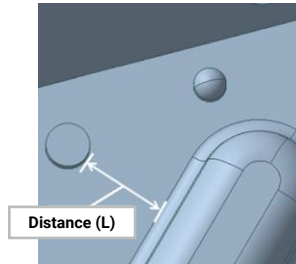
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and a half-punching is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Half-punching](#)" and "[Drawing](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

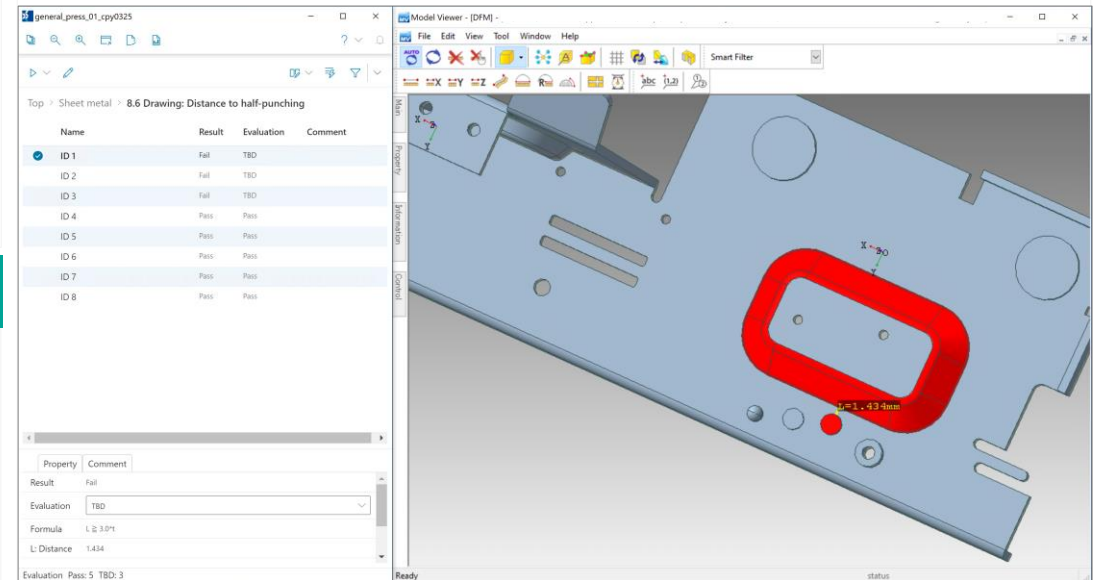
[A] When specifying by a fixed value with "Drawing: Distance to half-punching | Min. distance tol by fixed value" parameter: DrawingAndHalfPunchDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to half-punching | Min. distance tol by ratio to sheet metal thickness" parameter: DrawingAndHalfPunchDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing and half-punching pair which the distance in between is greater than the tolerance Red: [Fail] Drawing and half-punching pair which the distance in between is shorter than the tolerance





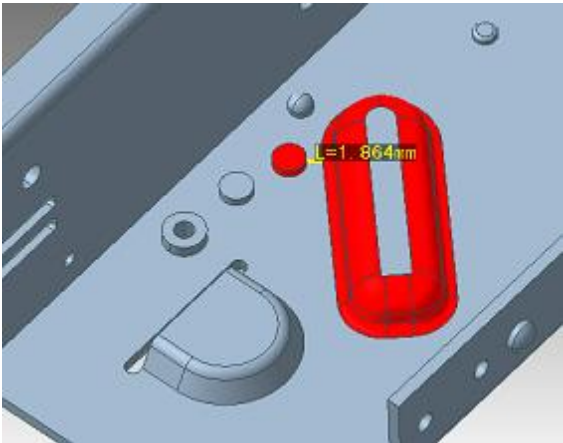
8.6 Drawing: Distance to half-punching

4. How to Analyze Check Result in Inspector / Model Viewer (Details)

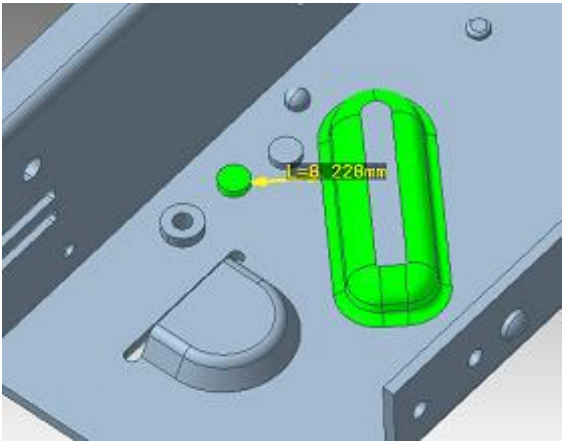
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and half-punching pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and half-punching pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

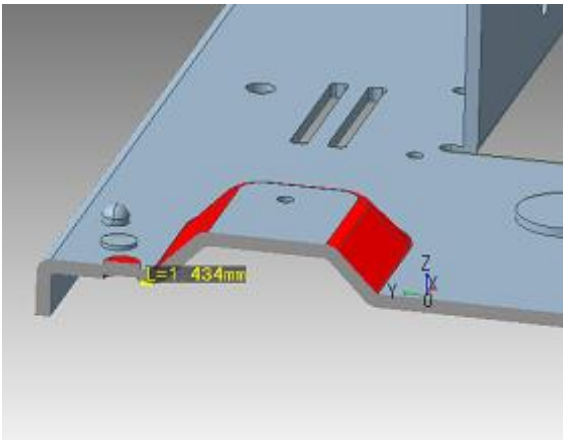
Highlight Checked Area - Fail



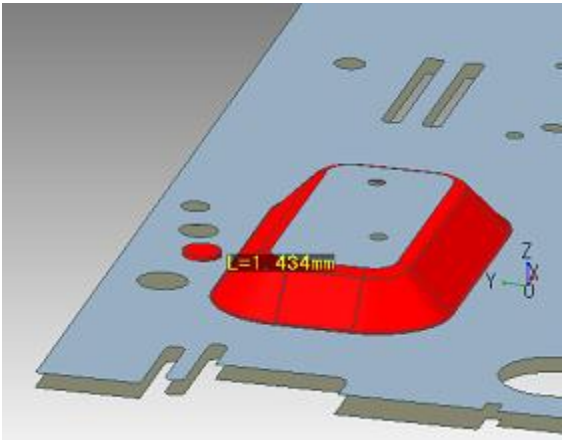
Highlight Checked Area - Pass



Section View



Hide Surroundings





8.7 Drawing: Distance to drawing

Description of Check Criterion & Possible Molding Defects

<Description>

Detects areas where the distance between a drawing and another drawing is shorter than the specified tolerance.

<Possible Molding Defects>



It may cause restrictions for machining methods, and/or a rise in the cost to make a drawing too close to another drawing. It may also cause deformation in machining.

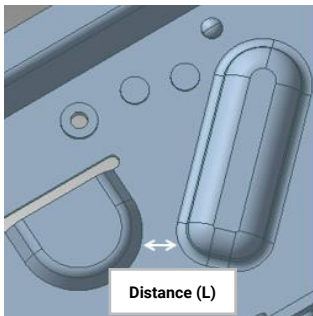
1. How to Check & Prerequisites

<How to Check>

Checks whether the distance (L) between a drawing and another drawing is greater than the tolerance.

<Prerequisites>

Geometry recognition is completed on "[Drawing](#)".



2. How to Judge & Default Value

<How to Judge>

Marks the area as Fail when the distance (L) is shorter than the tolerance.
A greater value will be used as the tolerance when both [A] (a fixed value) and [B] (standard thickness multiplied by the ratio) parameters are set. Set [A] to "-1" to always use the tolerance specified with [B].

<Default Value>

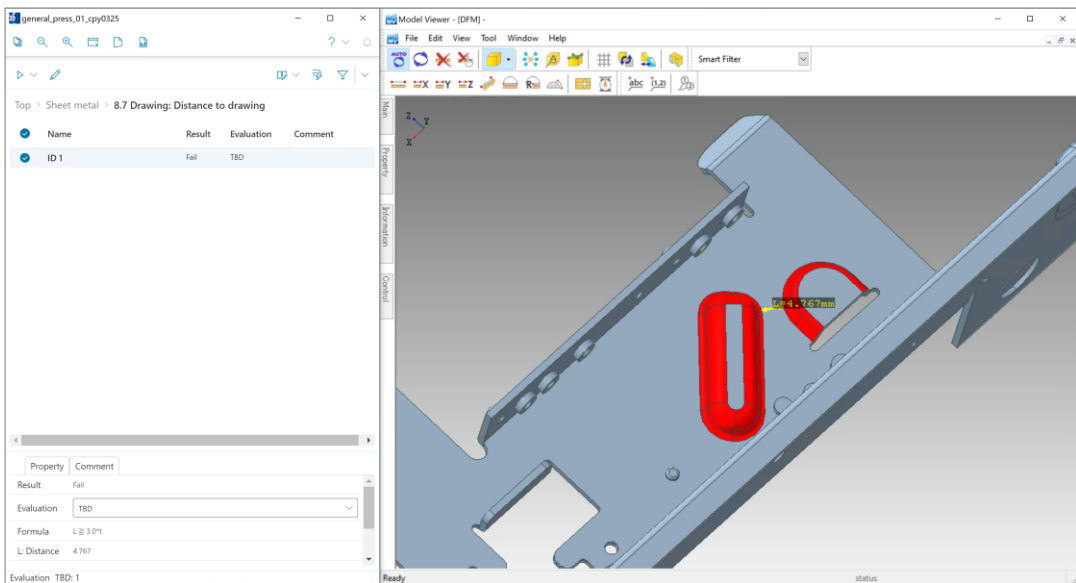
[A] When specifying by a fixed value with "Drawing: Distance to drawing | Min. distance tol by fixed value" parameter: DrawingAndDrawingDistance.Min=3.0 (Unit: mm)

[B] When specifying by a ratio to sheet metal thickness with "Drawing: Distance to drawing | Min. distance tol by ratio to sheet metal thickness" parameter: DrawingAndDrawingDistance.Min.Ratio=3.0

3. How to Analyze Check Result in Inspector / Model Viewer (Overview)

The check result will be listed in Item List of Inspector. Click a checked area to highlight it in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none"> Green: [Pass] Drawing and another drawing pair which the distance in between is greater than the tolerance Red: [Fail] Drawing and another drawing pair which the distance in between is shorter than the tolerance



8.7 Drawing: Distance to drawing

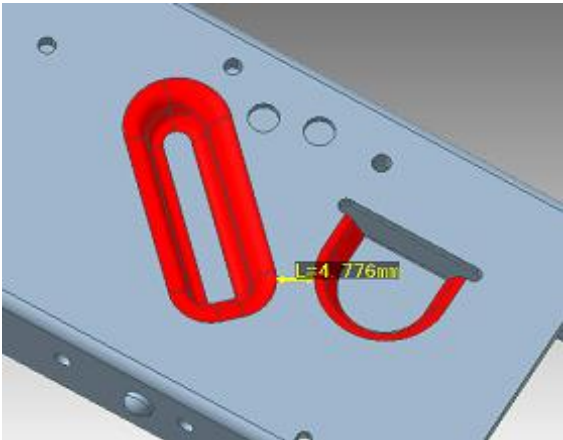


4. How to Analyze Check Result in Inspector / Model Viewer (Details)

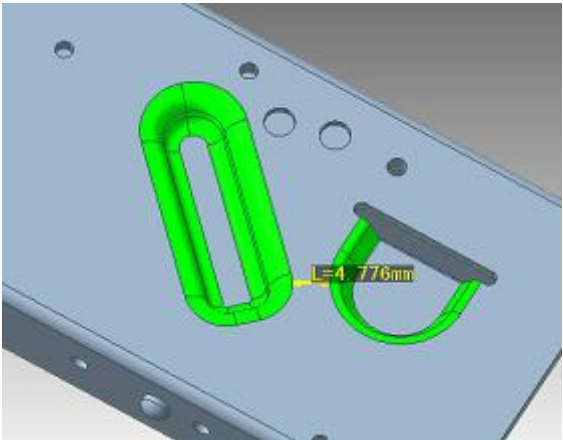
Click a checked area in Item List of Inspector to show the check result in Property List.
Also, right-click a checked area in Item List, and select following view commands from the context menu to analyze in Model Viewer.

View operation command	Description
Highlight Checked Area	<ul style="list-style-type: none">Green: [Pass] Drawing and another drawing pair which the distance in between is greater than the toleranceRed: [Fail] Drawing and another drawing pair which the distance in between is shorter than the tolerance
Highlight Checked Area (Without Dimension)	Highlights the checked areas as described above, but hides dimensions.
Fit to Checked Area	Fits to the selected checked area.
Section View	Shows the selected checked area in section view.
Hide Surroundings	Hides the faces for surroundings to show the selected checked area only.
Section View + Hide Surroundings	Hides the faces for surroundings in section view.
Simple View	Shows in simple view, canceling the applied highlighting and/or section view.
Show Selected Only	Filters checked areas in Item List to the selected ones only.
Edit Property	Opens a window to update Evaluation property, add a comment, etc.

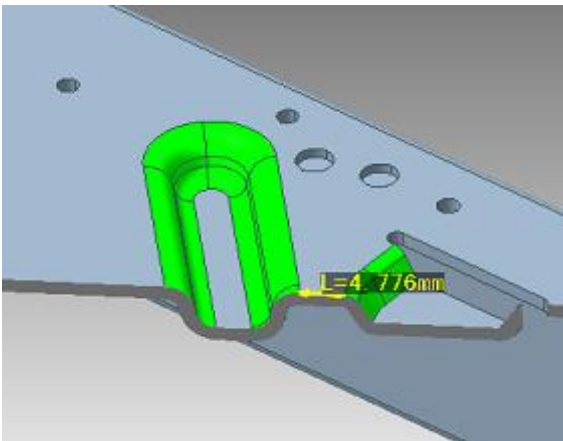
Highlight Checked Area - Fail



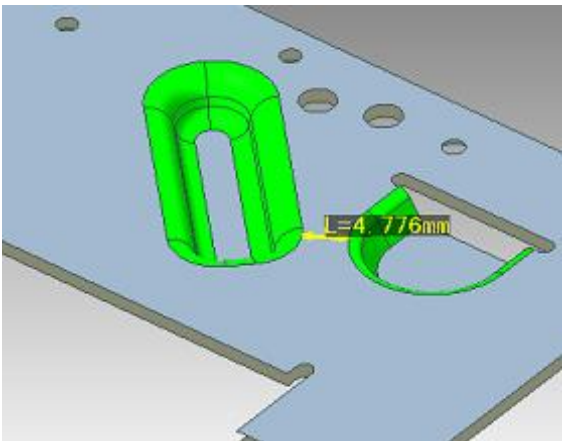
Highlight Checked Area - Pass



Section View

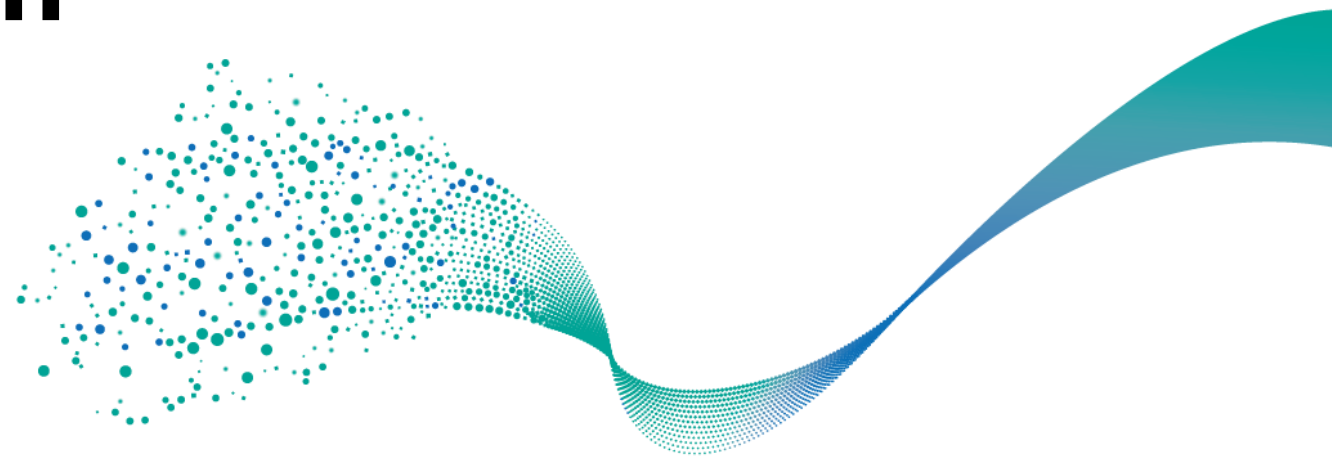


Hide Surroundings



Feature Recognition

Parameters and the Default Value





Index: Feature Recognition

- Sheet metal thickness & Standard thickness
- Front face / Back face / Side face
- Burring
- Punching hole
- Half-punching
- Dowel
- Drawing
- Simple bend
- Hem bend



Feature recognition: Sheet metal thickness & Standard thickness

Description of Geometry Check

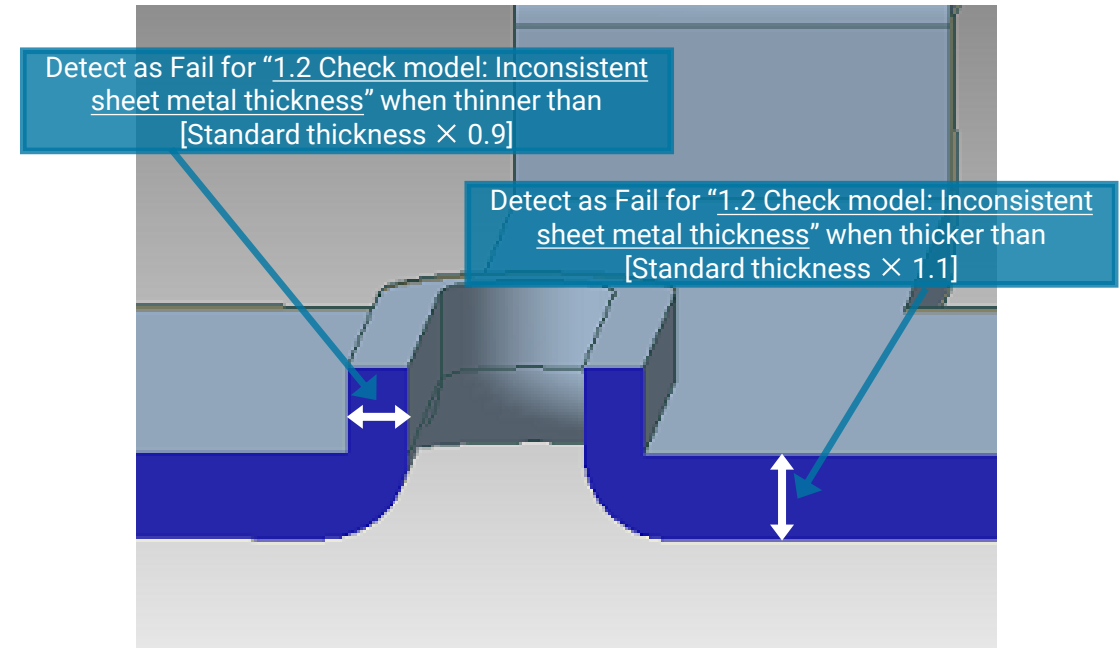
<Description>

Sheet metal thickness

- Recognizes the diameter of an inscribed sphere that passes a sampling point on a face and another face which makes an angle of 179 degrees or greater with that sampling point as the sheet metal thickness.

Standard thickness

- Uses the mode value of the sheet metal thickness measured at all sampling points as the standard thickness.
- Defines the maximum sheet metal thickness by the standard thickness multiplied by the ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" parameter below.
- Defines the minimum sheet metal thickness by the standard thickness multiplied by the ratio tolerance specified "Min. result to mode value ratio tol for sheet metal thickness" parameter below.



1. Prerequisites

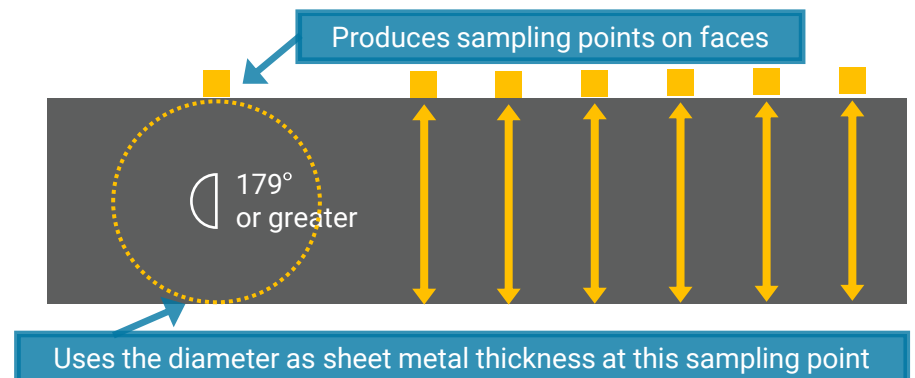
<Prerequisites>

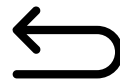
None

2. Parameters

Parameter	Default value	Description
Max. result to mode value ratio tol for sheet metal thickness StandardThicknessMax.Ratio	1.1	Specify the maximum sheet metal thickness by the ratio to the standard thickness.
Min. result to mode value ratio tol for sheet metal thickness StandardThicknessMin.Ratio	0.9	Specify the minimum sheet metal thickness by the ratio to the standard thickness.

How to Calculate Standard Thickness





Feature recognition: Front face / Back face / Side face

Description of Geometry Check

<Description>

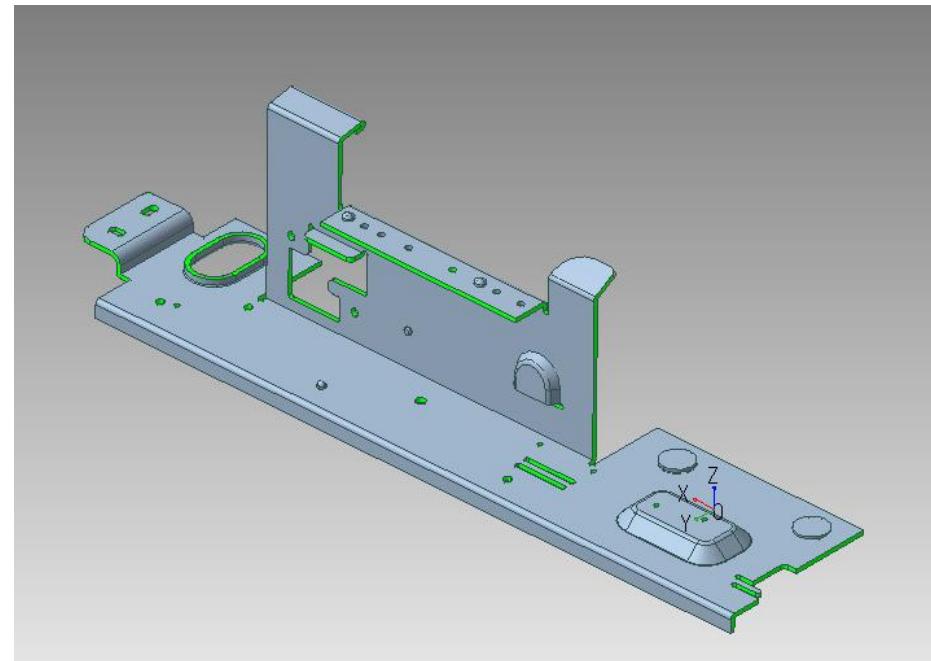
1. Recognizes as Front face / Back face when two faces satisfy following condition(s):

- Back face is located inside Front face within the distance of maximum sheet metal thickness specified with "PressThickness.Max" parameter below.
- Back face is located inside Front face within the distance of maximum sheet metal thickness specified with "Max. result to mode value ratio tol for sheet metal thickness" parameter.
- Front face and Back face are not connected to each other.
- Front face and Back face are facing each other.

2. Recognizes as Side face when a face satisfies following condition(s):

- Side face is connected to both Front face and its paired Back face.

Exception) Recognizes the top face of burring (co-axial through hole boss) as Side face.



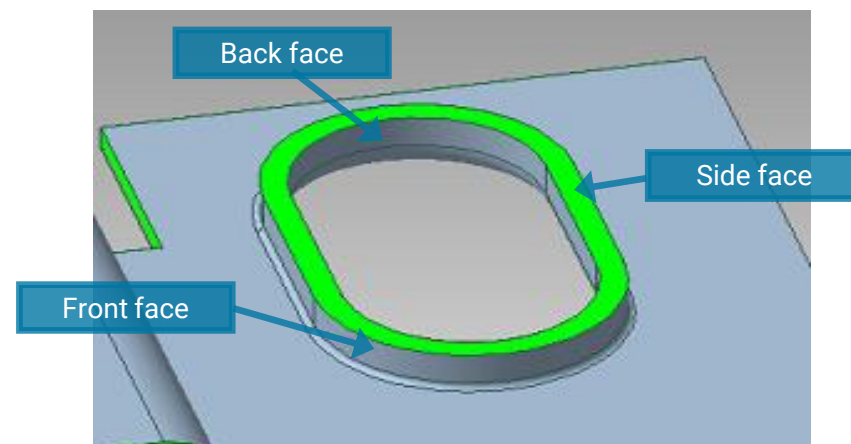
1. Prerequisites

<Prerequisites>

Geometry recognition is completed on ["Sheet metal thickness & Standard thickness"](#).

2. Parameters

Parameter	Default value	Description
Max. sheet metal thickness tol PressThickness.Max	5.1 mm	Specify the maximum thickness to recognize as Sheet metal thickness.





Feature recognition: Burring

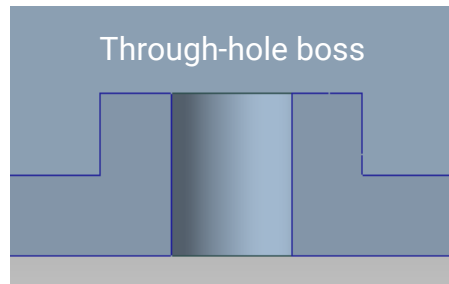
Description of Geometry Check

<Description>

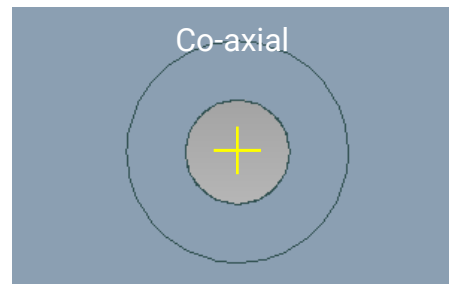
Recognizes the boss as Burring when it satisfies following condition(s):

- Boss that has a co-axial through hole.
- The inner diameter is 100 mm or smaller.
- The height is 5 mm or smaller.
- The sheet metal thickness is within the maximum sheet metal thickness (Standard thickness multiplied by the "Max. result to mode value ratio tol for sheet metal thickness").

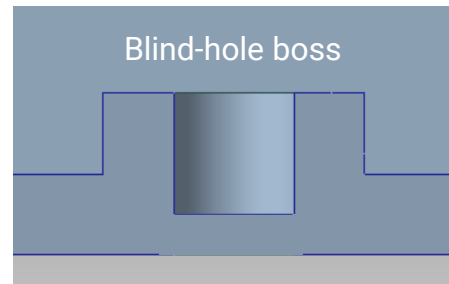
Recognizes as Burring



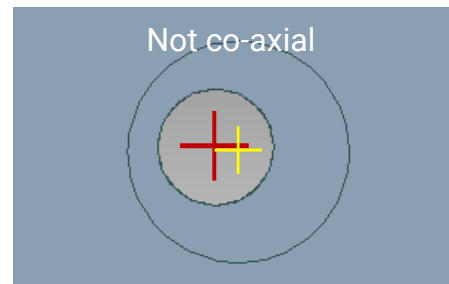
AND



Don't recognize as Burring



OR



1. Prerequisites

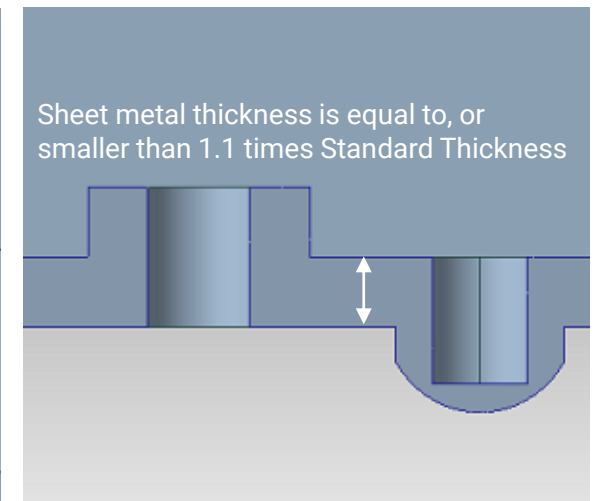
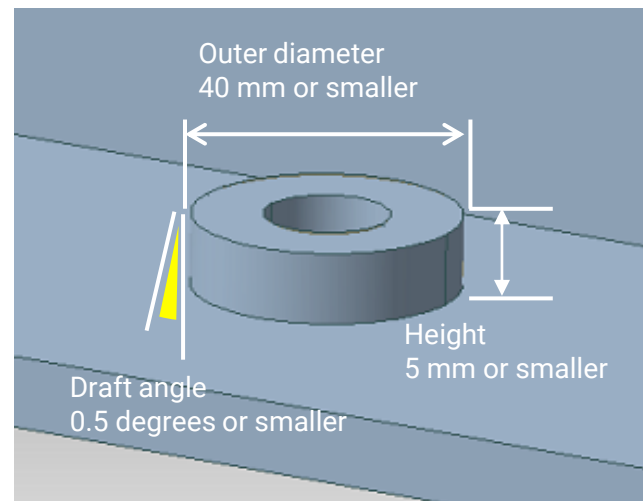
<Prerequisites>

Geometry recognition is completed on ["Sheet metal thickness & Standard thickness"](#).

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. burring outer diameter (*)	40 mm	Specify the maximum outer diameter to recognize as Burring.
Max. burring height tol RecognizedBurringHeight.Max	5 mm	Specify the maximum height to recognize as Burring.
Max. taper angle of Side face (*)	0.5 degrees	Specify the maximum taper angle of Side face to recognize as Burring.





Feature recognition: Punching hole

Description of Geometry Check

<Description>

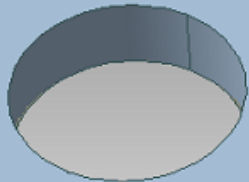
Recognizes the hole as Punching hole when it satisfies following condition(s):

- Hole whose Side face does not belong to the outline of the part.
- The inner diameter is 100 mm or smaller.

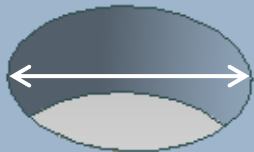
Exception) Does not recognize Burring holes as Punching hole.

Recognizes as Punching hole

Side face does not belong to
part outline

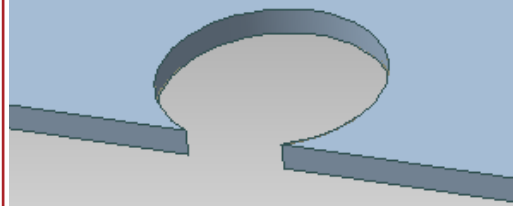


Hole diameter
is 100 mm or smaller

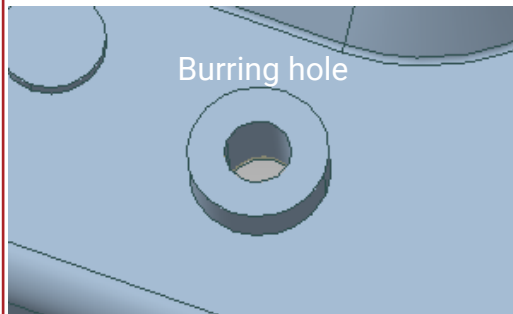


Don't recognize as Punching hole

Side face belongs to part outline



Burring hole



1. Prerequisites

<Prerequisites>

Geometry recognition is completed on "Recognize Round Hole".

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. hole diameter (*)	100 mm	Specify the maximum hole diameter to recognize as Punching hole.

Feature recognition: Half-punching

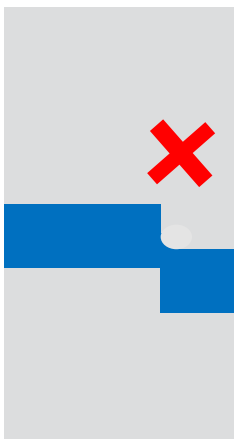
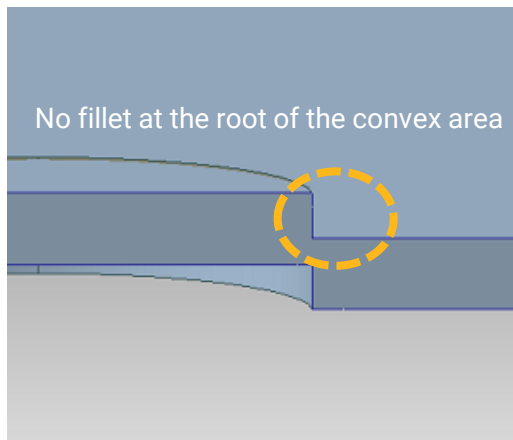


Description of Geometry Check

<Description>

Recognizes as Half-punching when the area satisfies following condition(s):

- Consists of faces that are recognized as Front face and Back face.
- Convex shape whose height is smaller than Standard thickness.
- Half-punching has a blind punching hole of the same height as the convex height.
- Top face of convex shape (Front face) and back face of punching hole (Back face) are not facing each other, and are parallel to each other (makes an angle of 179 degrees or greater).
- Half-punching does not have a fillet face at the root of the convex shape.
- The difference between the size of Front face outline and that of Back face outline is within Standard thickness multiplied by "0.2".



1. Prerequisites

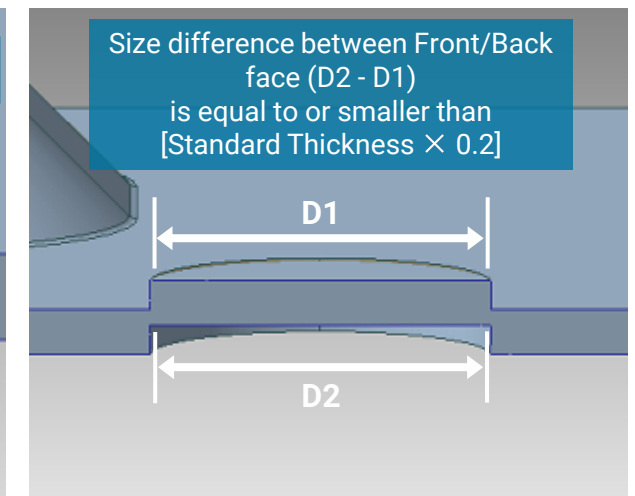
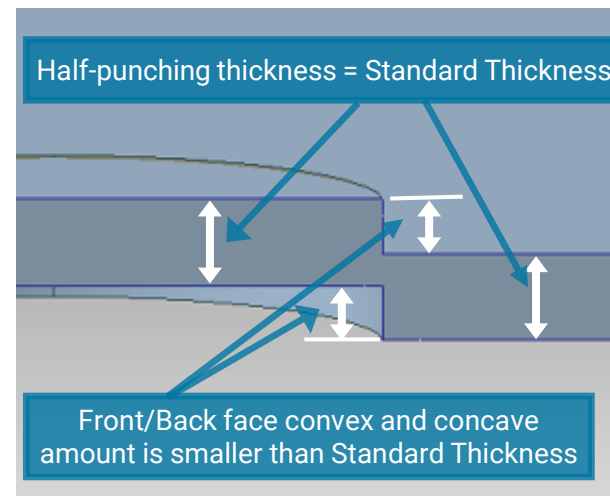
<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)".

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. difference tol between Front face outline size and Back face outline size (*)	Standard thickness \times 0.2	Specify the maximum difference tolerance between the size of Front face outline and that of Back face outline.
Max. convex / concave height (*)	Standard thickness	Specify the maximum convex / concave height.
Sheet metal thickness (Distance between Front face and Back face)) (*)	Standard thickness	



Feature recognition: Dowel

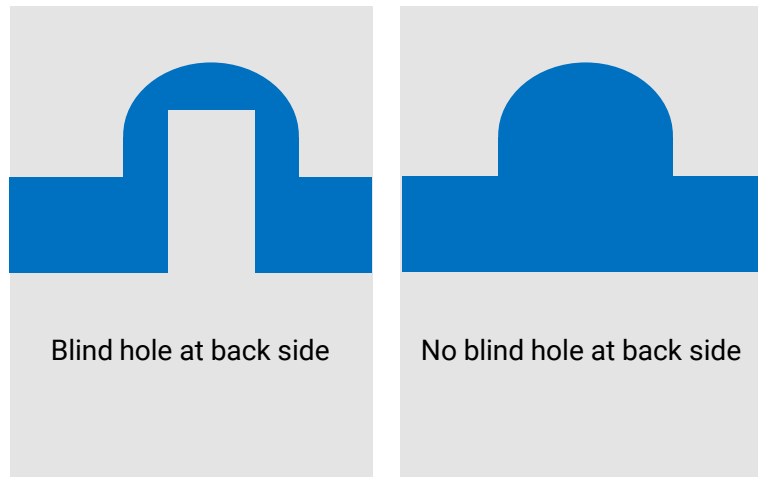
Description of Geometry Check

<Description>

Recognizes Boss as Dowel when it satisfies following condition(s):

- Boss that has either a co-axial blind hole or no hole at the back side.
- The difference between the outer diameter and the inner diameter of the co-axial blind hole is within Standard thickness multiplied by "0.2".
- The outer diameter is 40 mm or smaller.

Recognizes as Dowel



1. Prerequisites

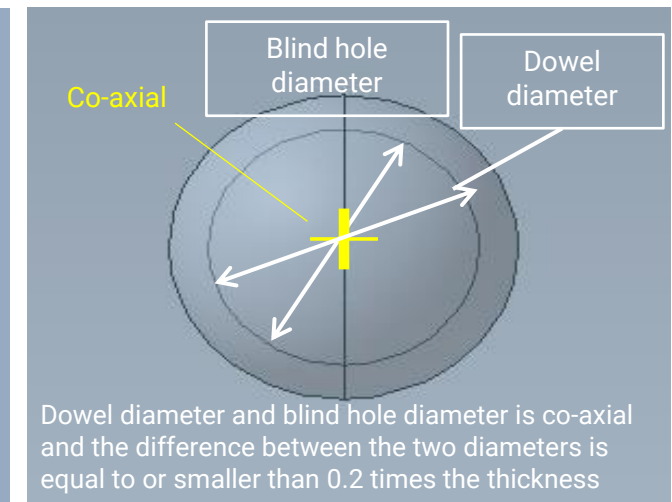
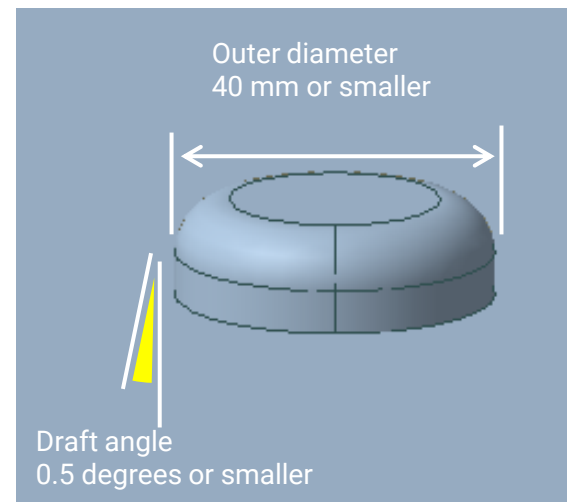
<Prerequisites>

Geometry recognition is completed on ["Sheet metal thickness & Standard thickness"](#).

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. boss outer diameter (*)	40 mm	Specify the maximum outer diameter to recognize as Boss.
Max. taper angle of Side face (*)	0.5 degrees	Specify the maximum taper angle of Side face to recognize as Dowel.



Feature recognition: Drawing



Description of Geometry Check

<Description>

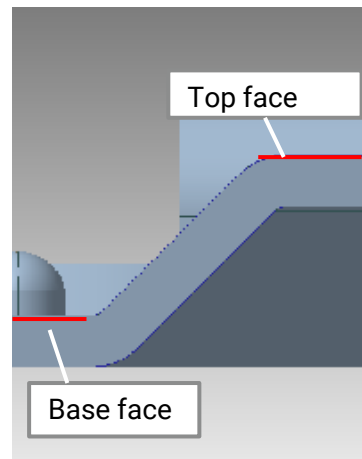
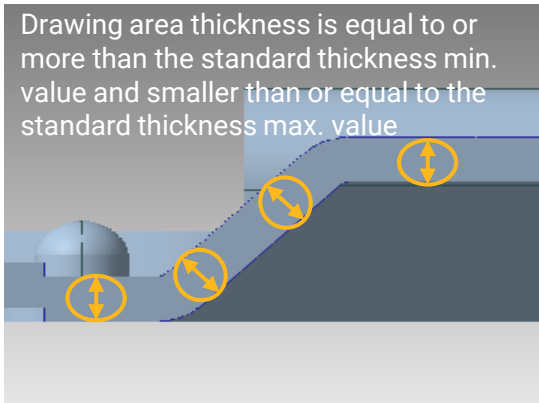
Recognizes as Drawing when an area satisfies following condition(s):

- Consists of faces that are recognized as Front face and Back face.
- Drawing shape whose height is greater than Standard thickness.

Exception) Does not recognize Half-punching as Drawing.

- Top face of Drawing (Front face) and back face of Drawing (Back face) are not facing each other, and are parallel to each other (makes an angle of 179 degrees or greater).
- The sheet metal thickness is within the thickness range (thicker than Standard Thickness multiplied by ratio tolerance specified with "Min. result to mode value ratio tol for sheet metal thickness" parameter, and thinner than Standard Thickness multiplied by ratio tolerance specified with "Max. result to mode value ratio tol for sheet metal thickness" parameter).
- Drawing has fillet face(s) that are not cylindrical faces at connecting points with Top face and Base face.
- The drawing height is 15 mm or smaller.

Drawing area thickness is equal to or more than the standard thickness min. value and smaller than or equal to the standard thickness max. value



1. Prerequisites

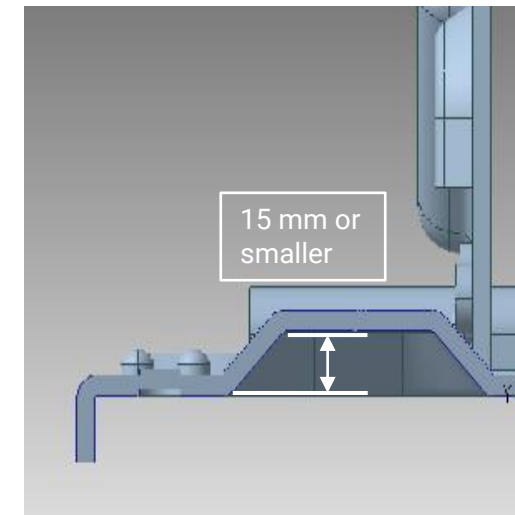
<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)" and "[Half-punching](#)".

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
StandardThicknessMax.Ratio (*)	1.1	Specify the maximum sheet metal thickness by the ratio to the standard thickness.
StandardThicknessMin.Ratio (*)	0.9	Specify the minimum sheet metal thickness by the ratio to the standard thickness.
Max. drawing height tol RecognizedDrawingCompoundHeight.Max	15 mm	Specify the maximum drawing height to recognize as Drawing.





Feature recognition: Simple bend

Description of Geometry Check

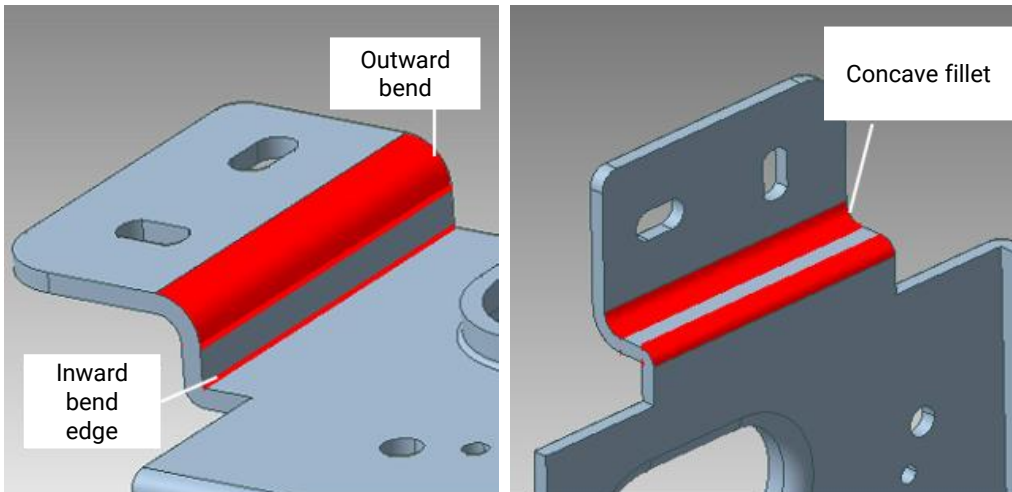
<Description>

Recognizes as Fillet on bend when a face satisfies following condition(s):

- Is a face that is recognized as Front face or Back face.
- Is a fillet, and a cylindrical face.
- Is connected to planar faces (Front face / Back face).

Recognizes as Edge on bend when an edge satisfies following condition(s):

- Is an edge that belongs to faces recognized as Front face or Back face.
- Is an edge connecting two planar faces (Front face / Back face) making a concave shape.



1. Prerequisites

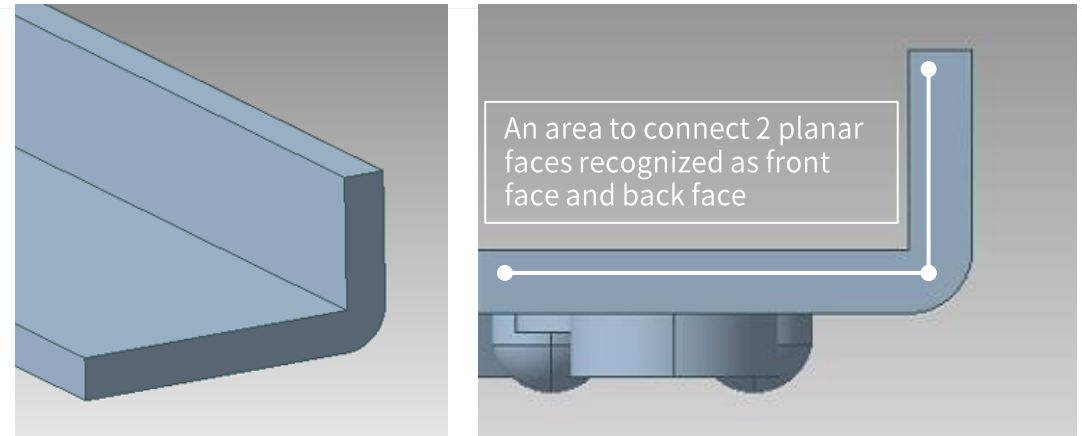
<Prerequisites>

Geometry recognition is completed on "[Front face](#) / [Back face](#) / [Side face](#)".

2. Parameters

(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. central angle for fillet on outward bend (*)	100 degrees	Specify the maximum central angle for fillets on outward bend to recognize as Simple bend.



Feature recognition: Hem bend

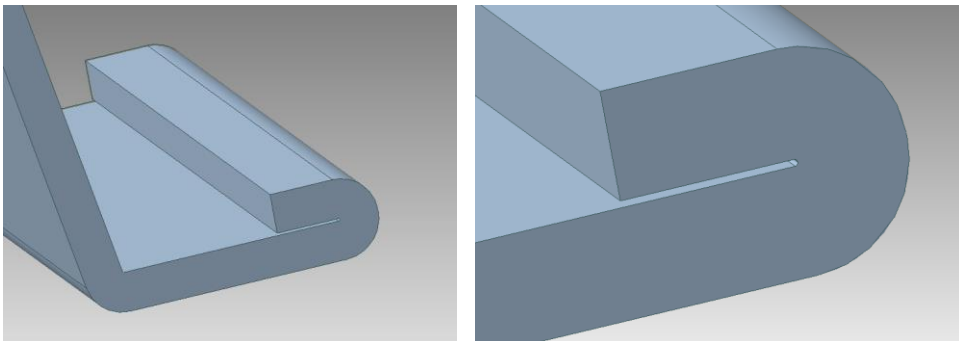


Description of Geometry Check

<Description>

1. Recognizes as Fillet on bend when a face satisfies following condition(s):
 - Is a face that is recognized as Front face or Back face.
 - Is a fillet and a cylindrical face.
 - Is connected to planar faces (Front face / Back face).
2. Recognizes as Edge on bend when an edge satisfies following condition(s):
 - Is an edge that belongs to faces recognized as Front face or Back face.
 - Is an edge connecting two planar faces (Front face / Back face) making a concave shape.
3. Recognizes as Hem bend when an area satisfies following condition(s):
 - The faces that are connected to Fillet on bend and Edge on bend are planar faces, parallel to each other, and facing each other.

Exception) This targets Open hems only, and does not recognize Tear-dropped hems as Hem bend.



1. Prerequisites

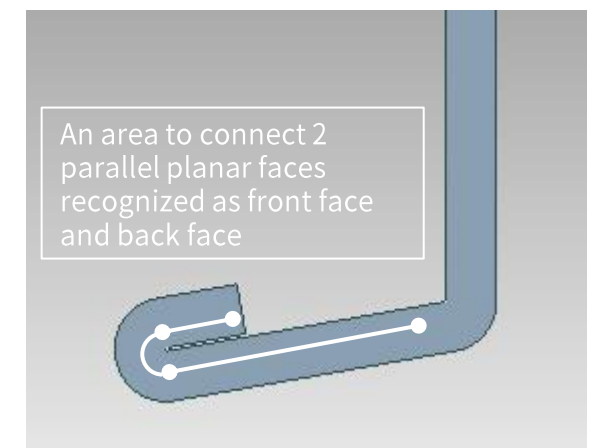
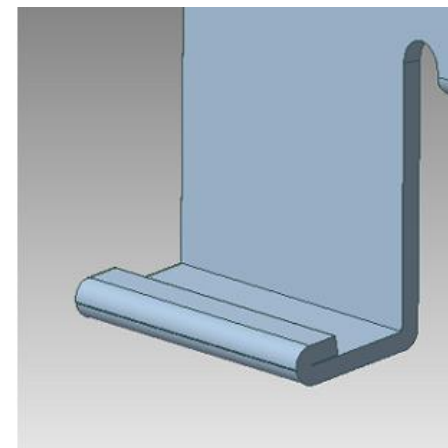
<Prerequisites>

Geometry recognition is completed on "[Front face / Back face / Side face](#)".

2. Parameters

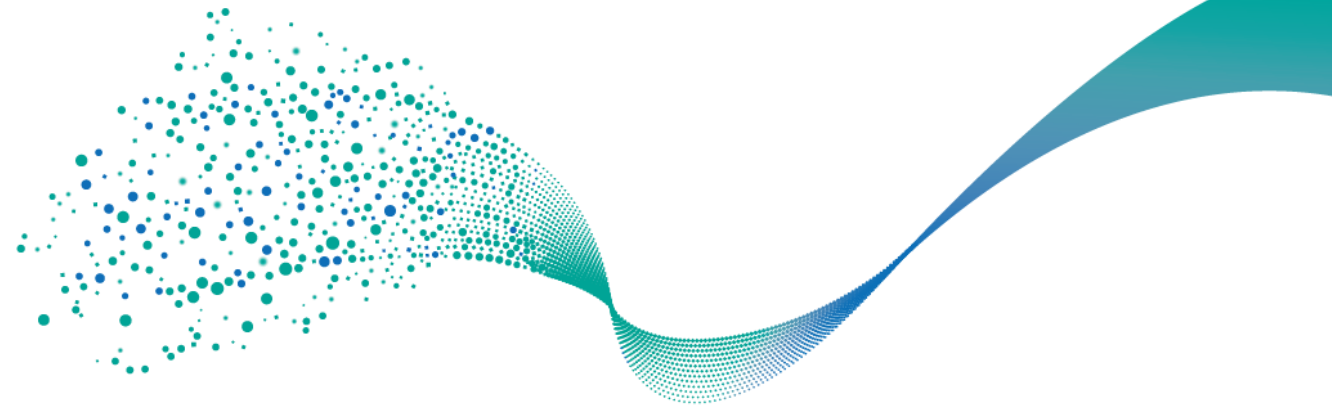
(*: Please note that parameters with "*" are internal parameters, and are uneditable.)

Parameter	Default value	Description
Max. central angle for fillet on outward bend (*)	100 degrees	Specify the maximum central angle for fillets on outward bend to recognize as Hem bend.
Max. hem inside diameter (*)	1.01 mm	Specify the maximum hem inside diameter (=distance between two parallel planar faces).
Min. hem inside diameter (*)	0 mm	Specify the minimum hem inside diameter (=distance between two parallel planar faces).



Appendix

How to Edit Parameter Settings



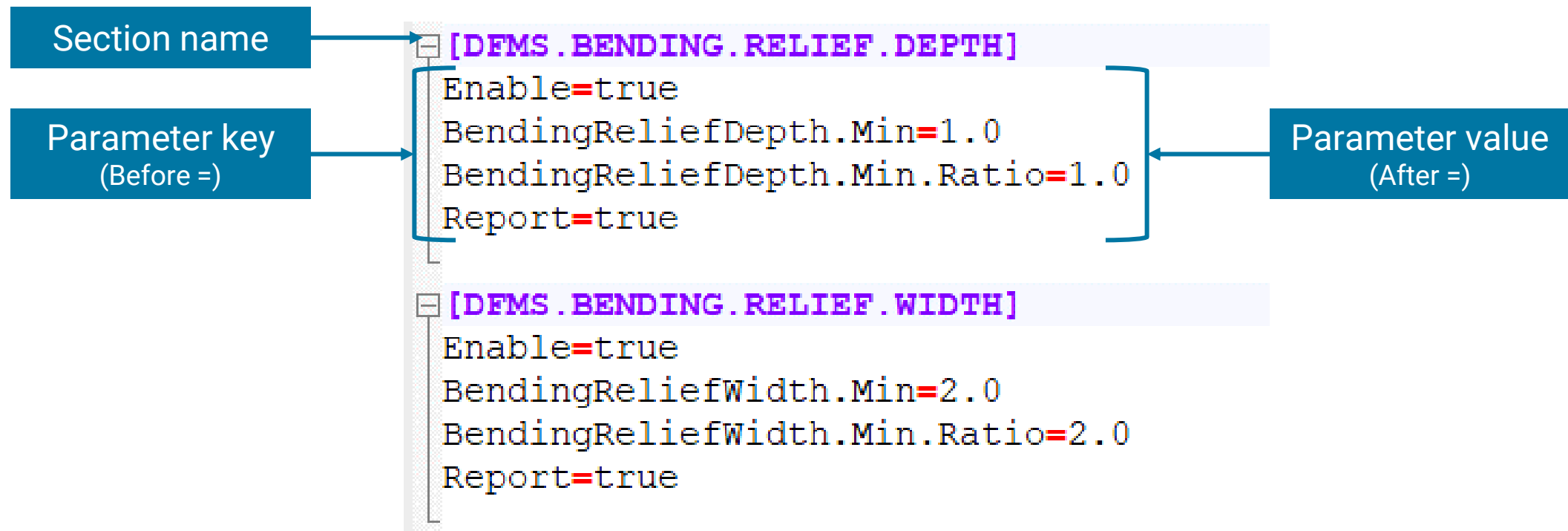


How to Edit Parameter Settings

Please use Parameter Settings Tool to edit parameter values.

In case you edit the parameter files (*.ini) directly, the format is as shown below.

Please confirm the correspondence between the check criteria name and the section key in Parameter Settings Tool.



All rights reserved by Elysium or the original author of this material.
The content may not be edited, reproduced, distributed, transmitted, displayed,
published, broadcast, sold or lent without the prior permission of the author.

