

Dynavista

CAA V5 based

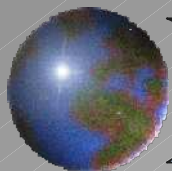
V10.2 / V11.2

Dieface Design

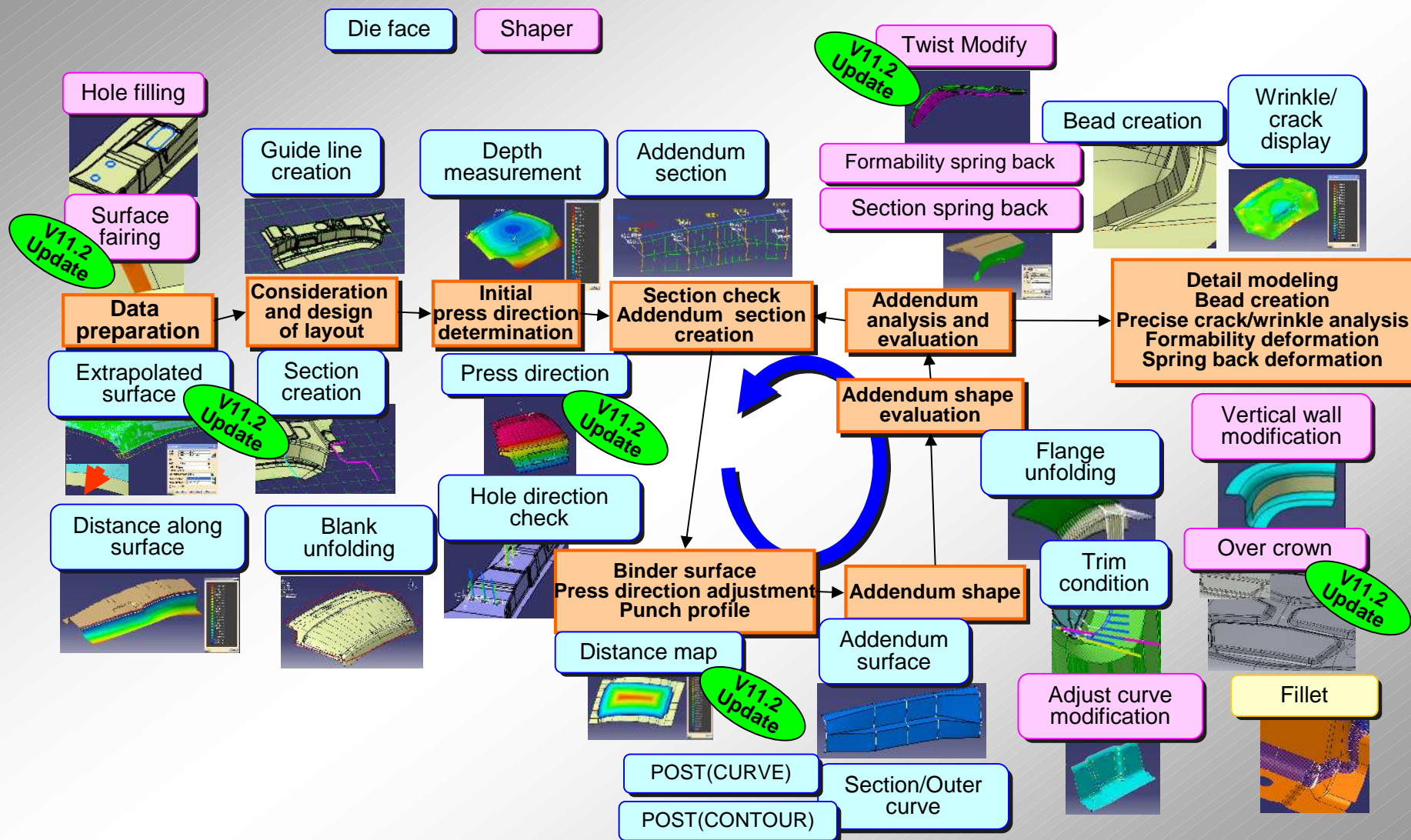
October, 2011

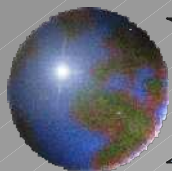
UNIADEX, Ltd.

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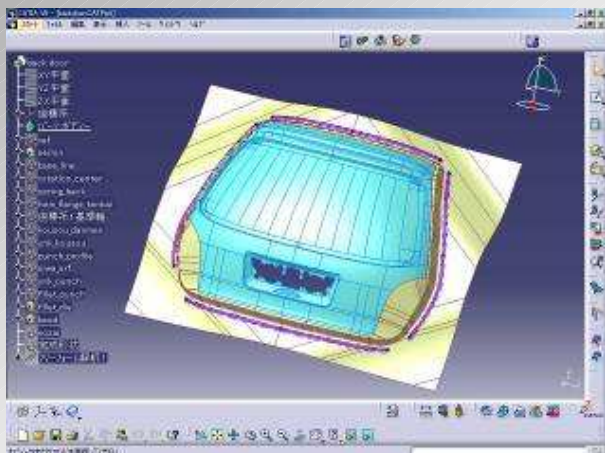
Dynavista functions for die layout design





Dieface Design

Improvement of efficiency and quality in die face design.



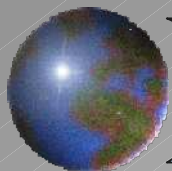
V5 prerequisites: HD2

- Drastic reduction of design man-hour by specific functions for die face shape - such as shape creation, edition and evaluation.
- Design quality can be automatically build-in by evaluation before production with the effective use of design know-how.

 V5R18

 V5R19

 V5R20



Dieface Design command

Die Axis System

Depth Measurement

Trim condition

Hole press
angle check

**Formability
evaluation**

Bead

R attribute

Flange expansion
(curve/surface)

Extrapolated
surface

Correct flange expansion

Addendum shape

Die face specific shape creation

POST (CURVE)

POST (CONTOUR)

Objective

Section/Outer curve

CAE Association

Dynavista / Dieface Design

3D guide line
creation

3D section
creation

2D section
creation

2D guide
line creation

Auxiliary guide
line creation

Layout support

Surface normal offset

Offset along surface

Variable translation

Automatic selection
of concave edges

Distance
along support

Dynavista Law

Complementary functions

Thickness

Distance map

Area projection

Distance along
surface

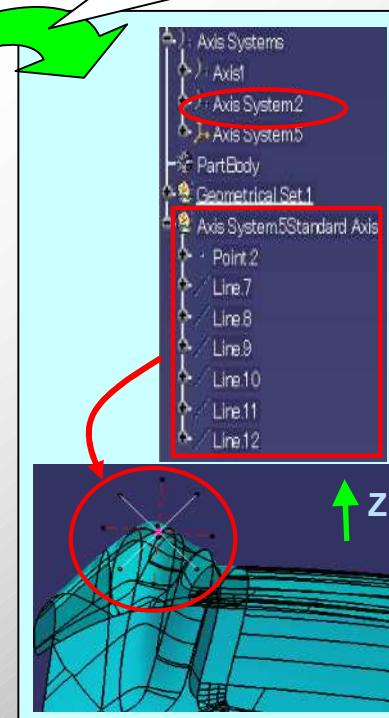
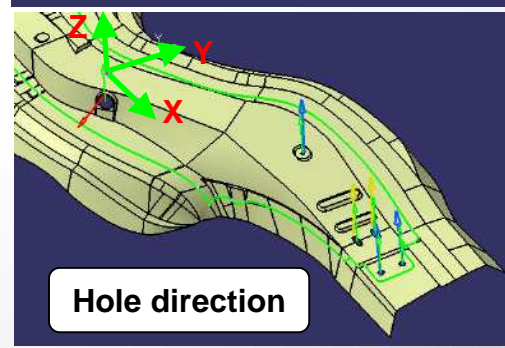
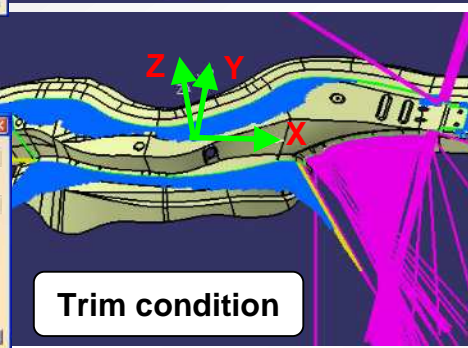
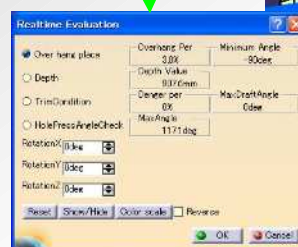
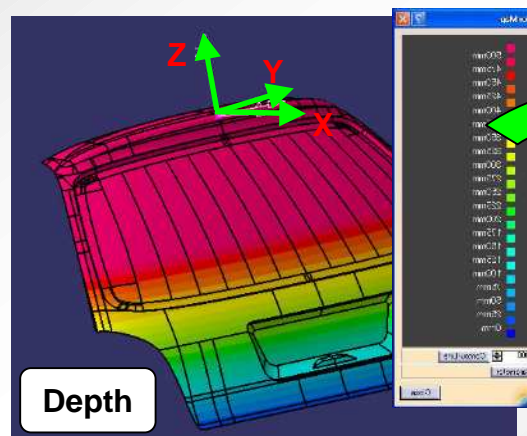
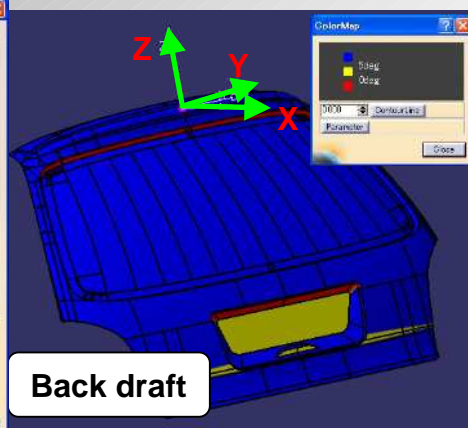
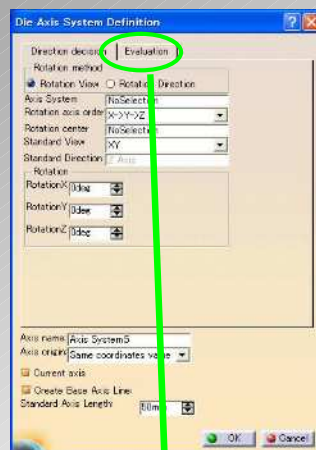
Analysis



Die standard Axis system

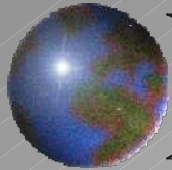
- Definition of a coordinate system by varying a press direction in real time manner.
- Evaluations (back draft, depth, trim condition and hole direction) are also available.

An optimum press direction is determined after the evaluation (A new coordinate system will be created.)



- Evaluation will be performed for the press direction.
- Each of the evaluation items will be checked by the real-time rotation of the axis.

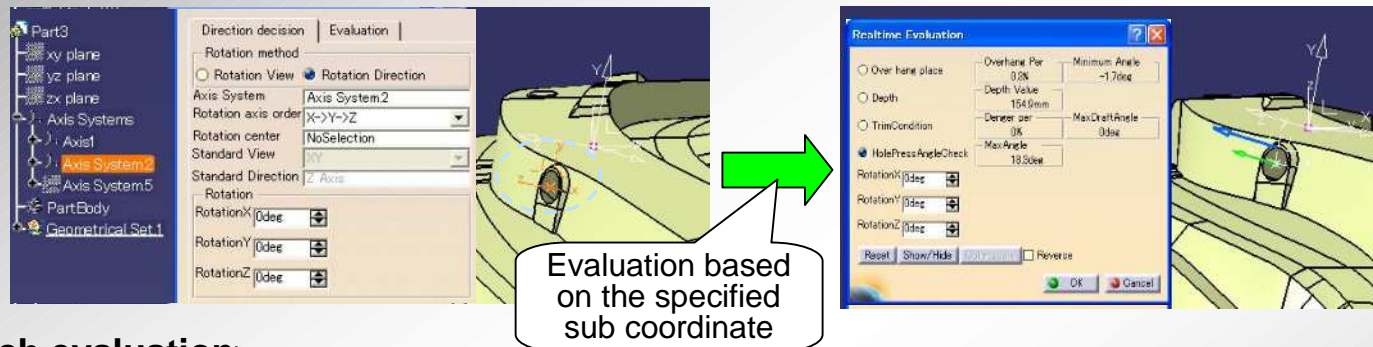




Die standard Axis system (sub-coordinate based, batch evaluation)

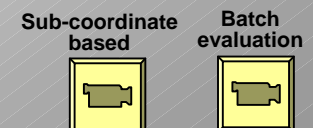
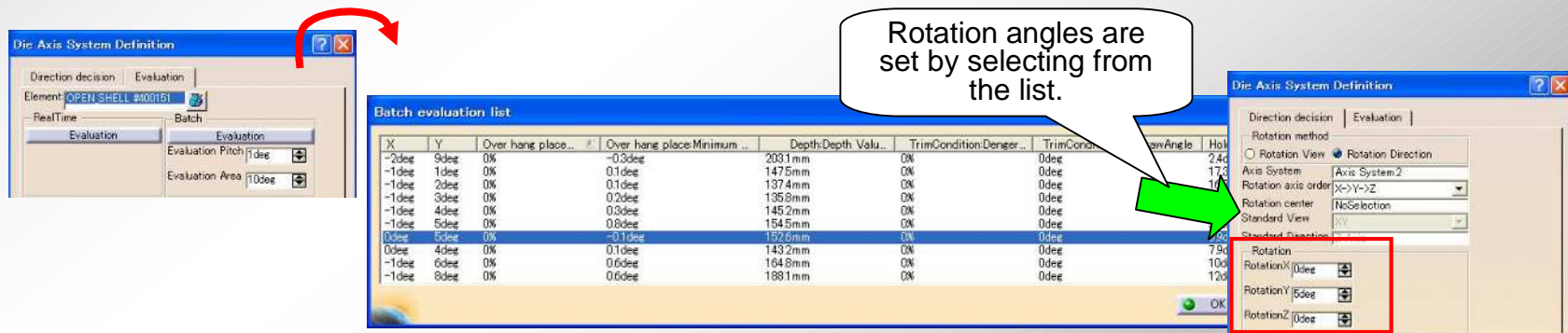
<A coordinate system creation based on another sub-coordinate system>

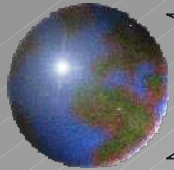
A sub-coordinate system can be created based on not only the main coordinate system but also a specified sub-coordinate system after evaluating various conditions.



<Batch evaluation>

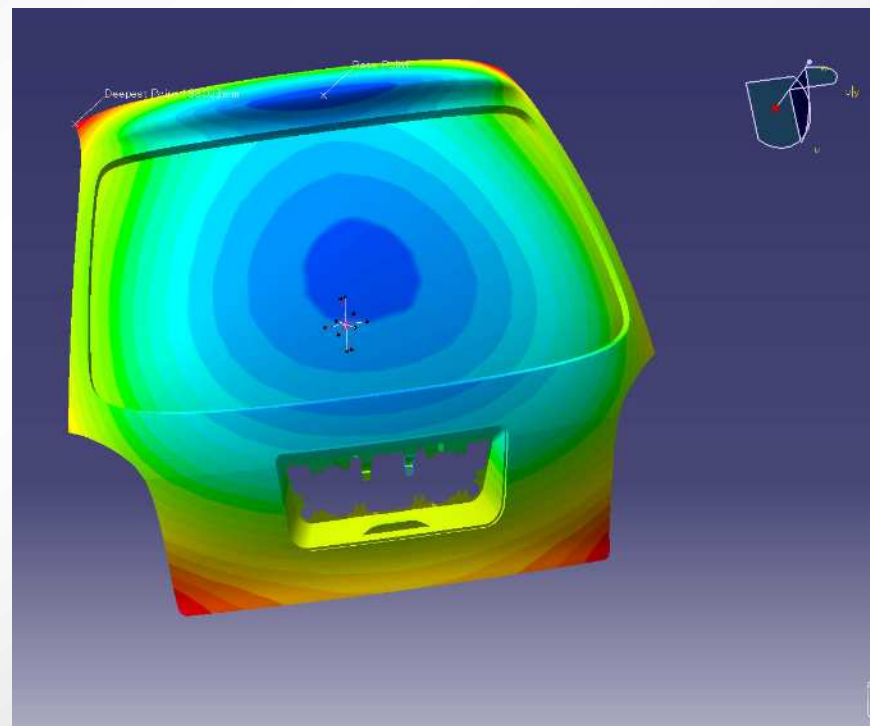
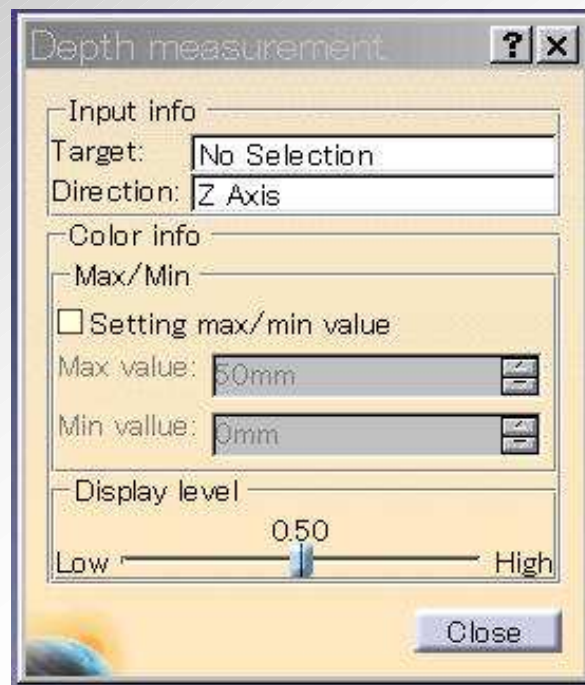
An optimum direction (rotation angle can be selected among an evaluation result list by running the Batch evaluation.





Depth Measurement

- Depth will be calculated from the highest point to the target surface along with specified coordinate axis.
- More precise color map will be displayed.



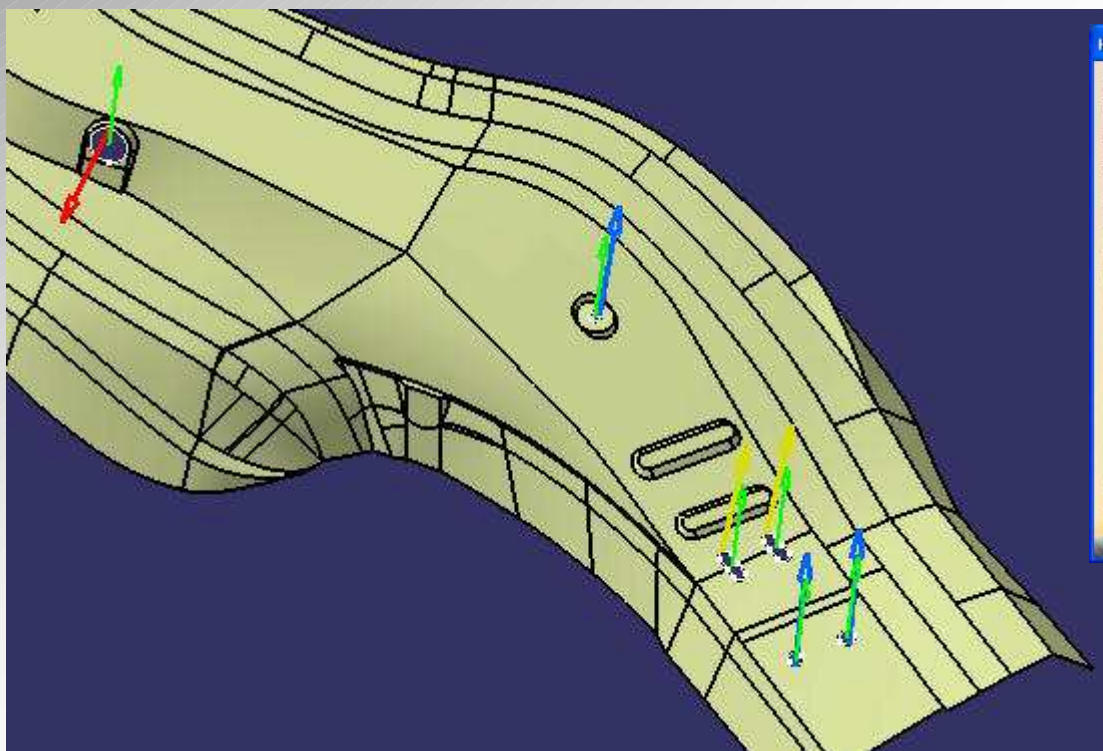
Color map





Hole press angle check

Check if piercing is possible along with a specified direction, and display the check result on the screen and a list.



HolePressDirectionCheck

ProductSurface: SurfaceA [Reverse]

Thickness: 1mm

Dangerous Angle: 5deg

Impossible Angle: 10deg

DirectionGroup: Axis System 4(HoleDirec) [List]

Evaluation: [Current DirectionGroup] [Each DirectionGroup] [Auto Group] [Auto Grouping]

HoleName	DirGroup	Result	Angle	Size	PierceDiameter
PunchHole.1	None	Possible	0deg	16mm	16mm
PunchHole.2	None	Possible	0deg	16mm	16mm
PunchHole.3	None	Dangerous	8.2deg	20.01mm	20.02mm
PunchHole.4	None	Dangerous	8.3deg	20.01mm	20.02mm
PunchHole.5	None	Impossible	13.8deg	12mm	12.04mm
PunchHole.6	None	Possible	2deg	71mm	Off the Subject

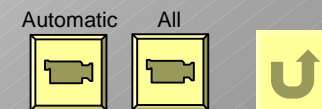
Add to current DirectionGroup

OK Cancel

Holes are automatically retrieved by specifying a product shape, and an arrow is displayed as follows.

- yellow for dangerous angle
- red for impossible angle
- blue for safe angle

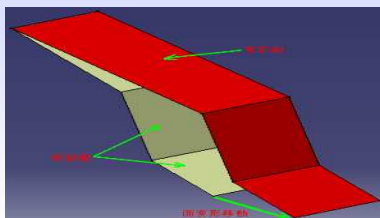
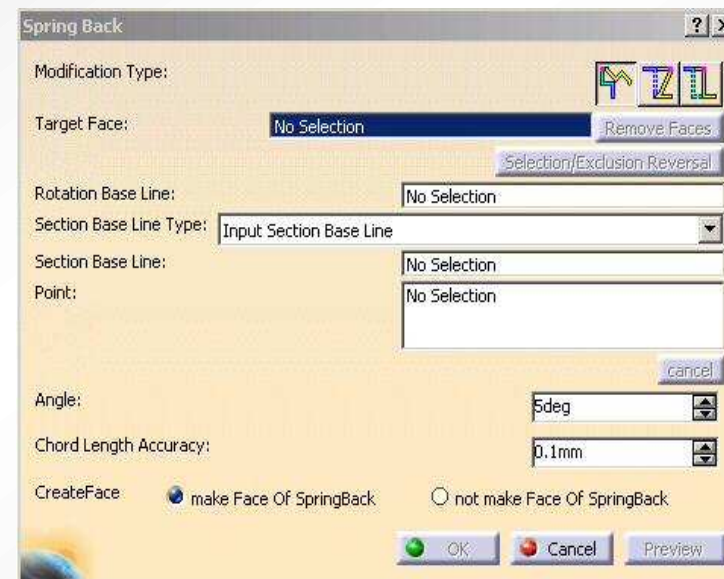
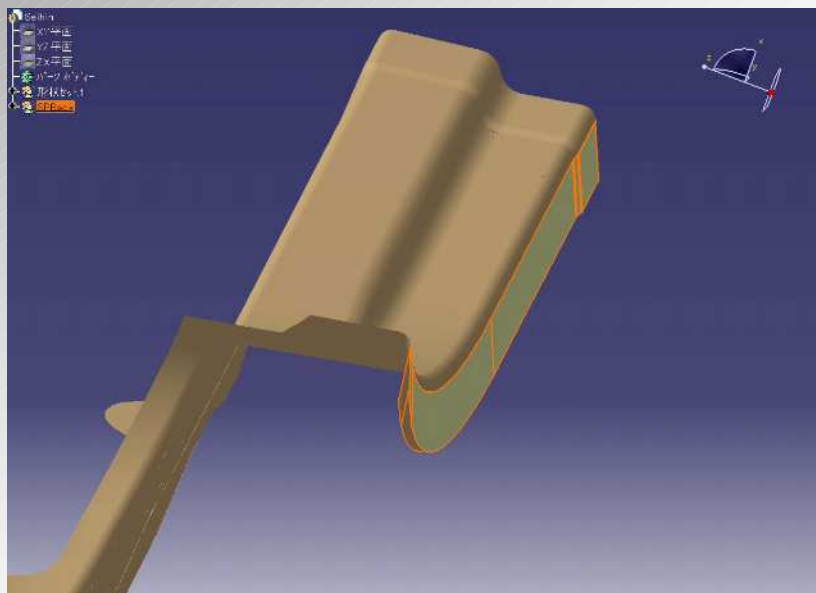
- Two or more directions can be defined for a cam mechanism.
- By "Automatic grouping", holes are automatically collected as a group if they can be pierced along the specified angle. For other holes, new piercing angles will be automatically created for the grouping.



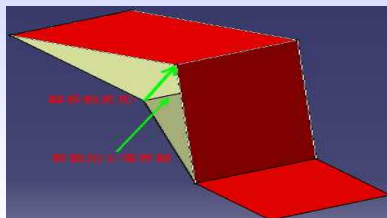


Spring back

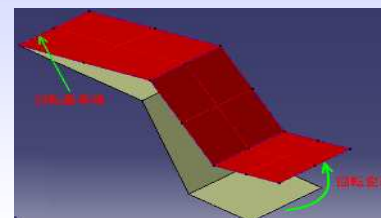
- Shape creation considering elastic behavior at the forming



Surface movement

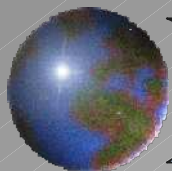


Line movement



Rotation

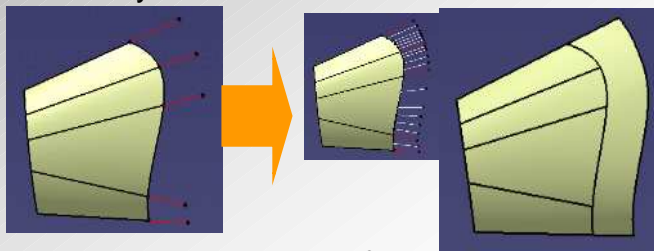




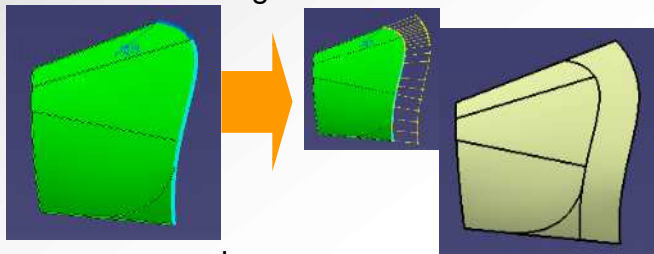
Extrapolated Surface

- Create a surface by extending outer boundary curves of a composite surface.
- Tangent continuity is secured between a specified surface and an extrapolated surface created .

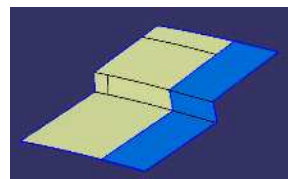
(1) Patch boundary is created between cross boundaries.



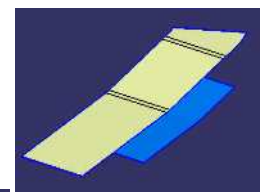
(2) Tangent direction is neglected for cross boundaries.



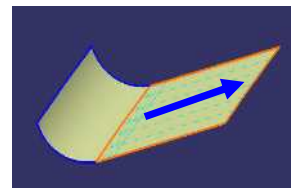
(3) Gouges are removed.



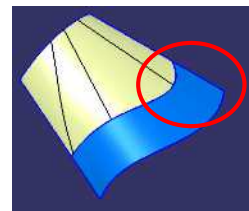
Extend after separating
bend portion



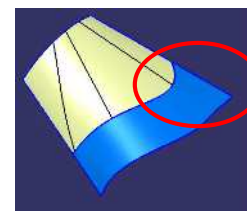
Range of extension
is specified.

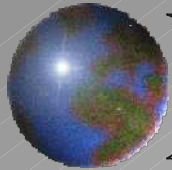


Linear extension with G2 continuity at end portion.

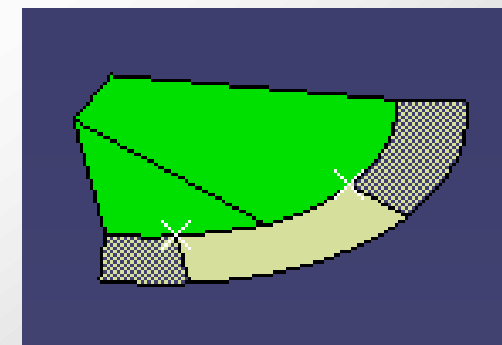
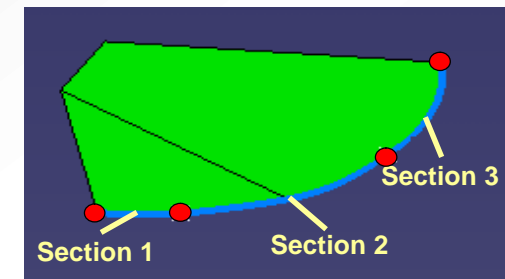
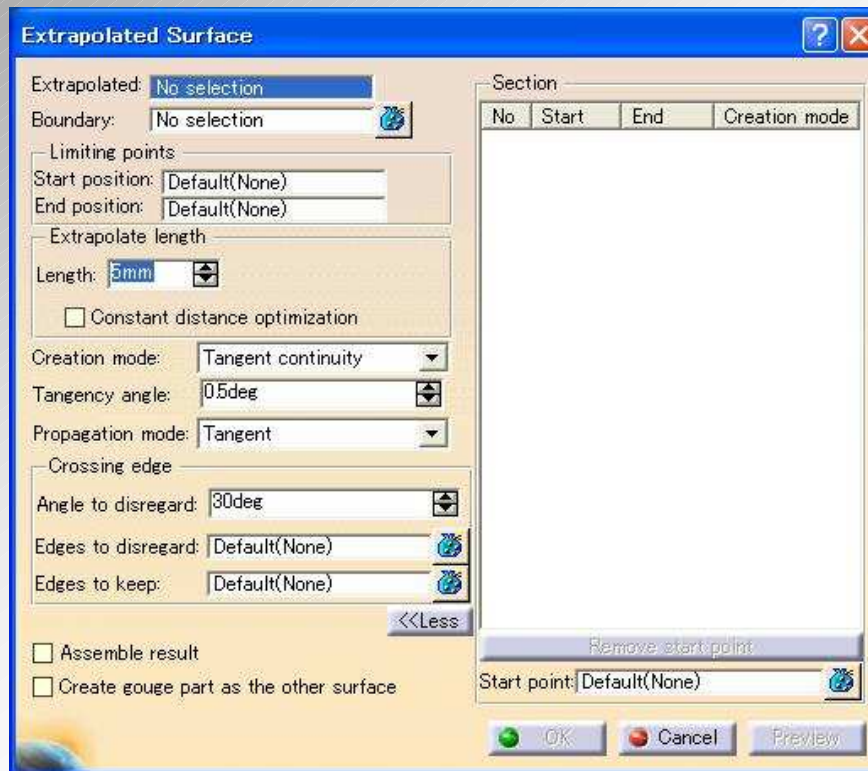


Constant surface width is possible.





Extrapolated Surface



An extrapolated surface will be created based on sections divided by a user from a range of extrapolation.
Element surfaces will be separated at a gouge portion.

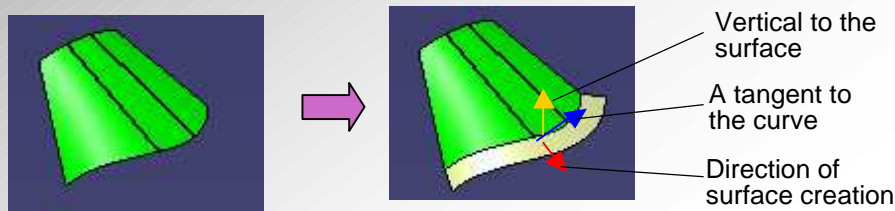


Extrapolated Surface

-Two types of surface creation, Curve normal and Surface normal.

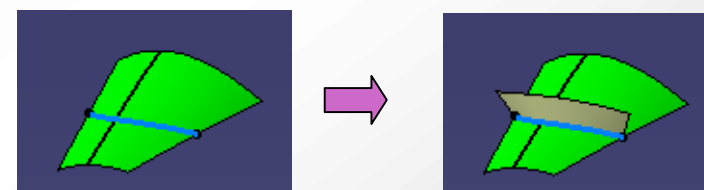
Curve normal

A surface is created so that a patch boundary is perpendicular to both a tangent to the curve and vertical to the surface.



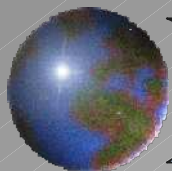
Surface normal

A surface is created so that a patch boundary is vertical to the surface



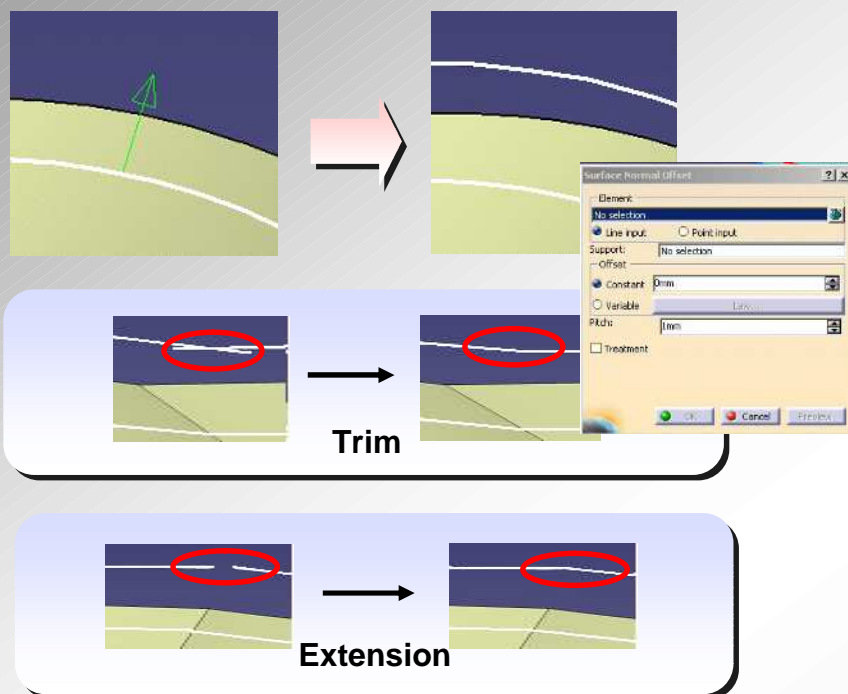
Handling at a bending corner

		former	Corner handling
Convex			
Concave			



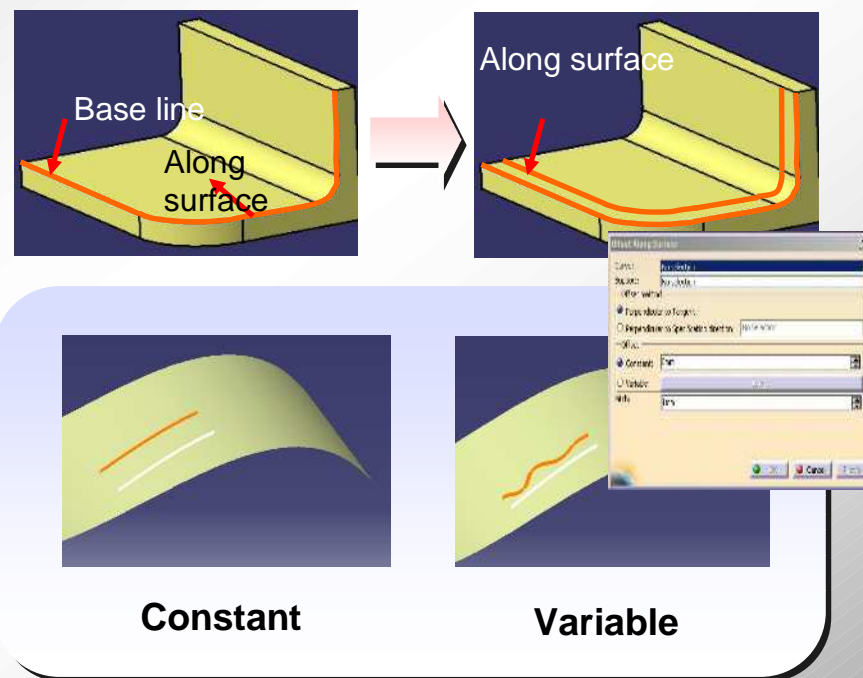
Surface normal offset / Offset along surface

- Curve offset perpendicular to a surface



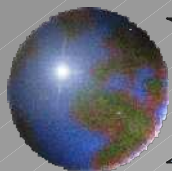
- Two or more curves are allowed to input.
- Variable offset value is possible.

- Curve offset along a surface



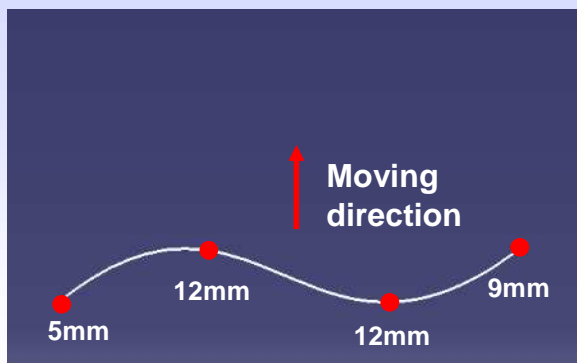
- Two or more curves are allowed to input.
- Variable offset value is possible.



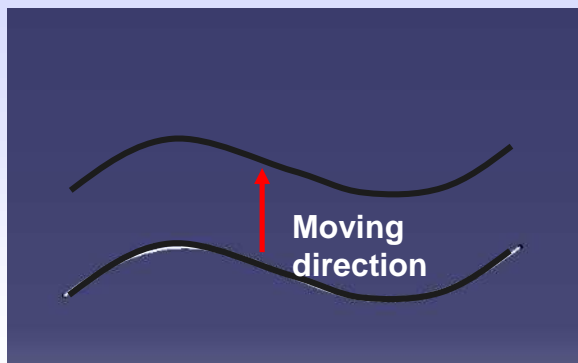
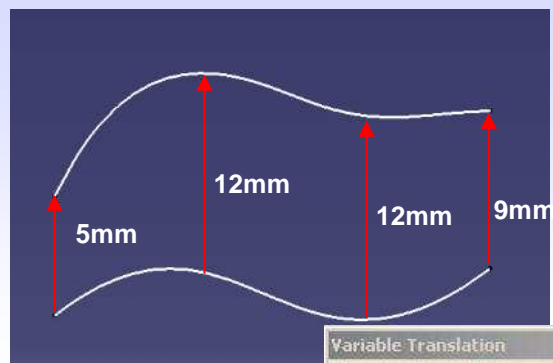


Variable translation

- Line movement along specified direction



Variable



Constant



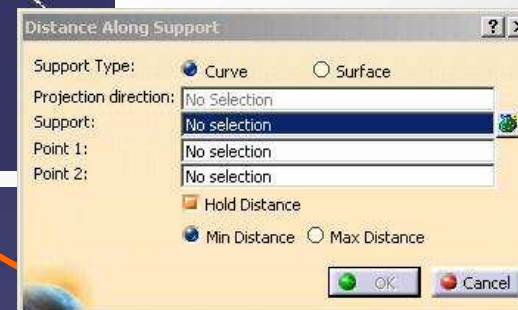
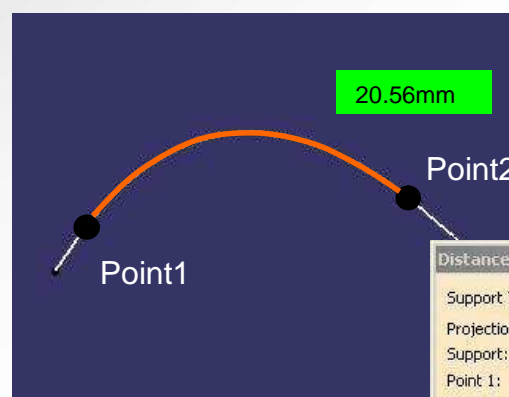
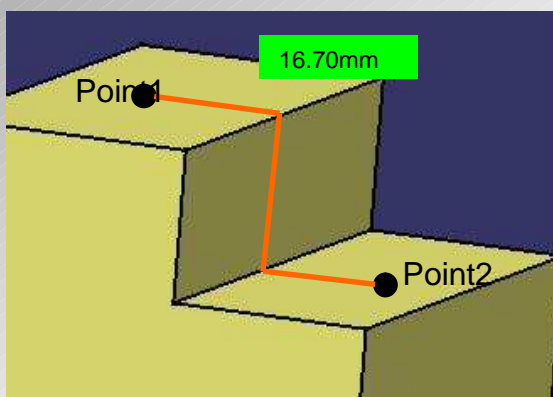
- Two or more curves are allowed to input.
- Variable offset value is possible.



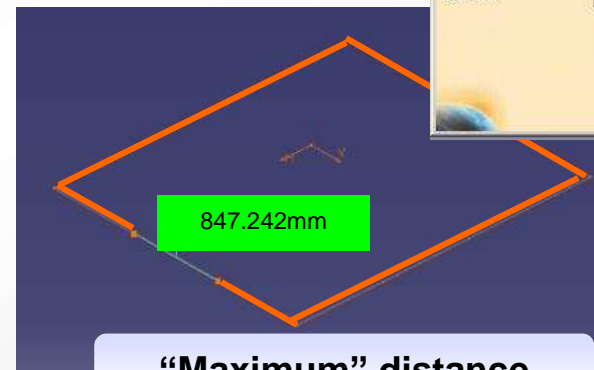


Distance along support

- Measurement of distance between two points along two or more curves or surfaces



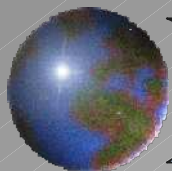
“Minimum” distance



“Maximum” distance

- The result can be saved as a feature
- Collaboration with the knowledge function

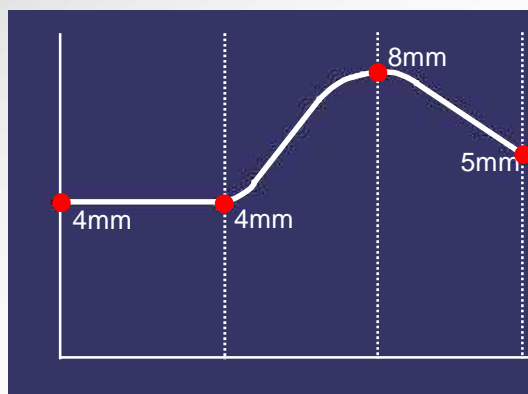




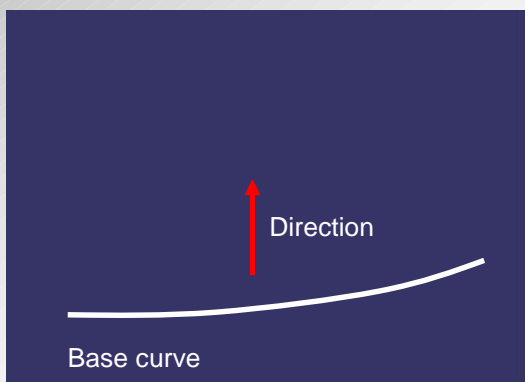
Dynavista Law

- Definition of a variable rule. Different from CATIA law, variable value can be specified at each point in Dynavista law.

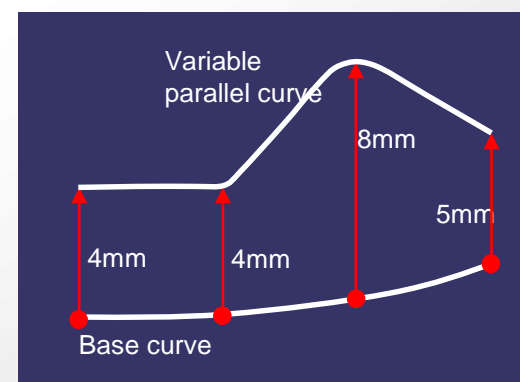
Defined law



Sample of the law (variable parallel curve)

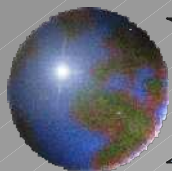


Variables defined by the law



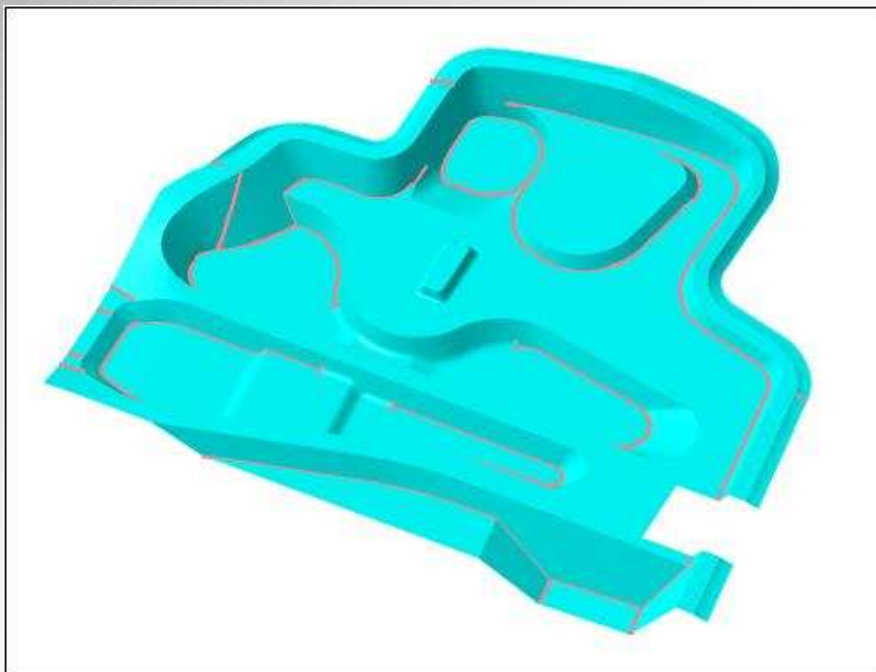
- This law can be used in the commands such as “Vertical offset”, “Fillet” and “Spring back”.
- Also can be used in existing CATIA command.





Automatic selection of concave edges

- Extract concave edges by viewing from front side from composite surface boundaries.
- Threshold angle of the bend can be changed.



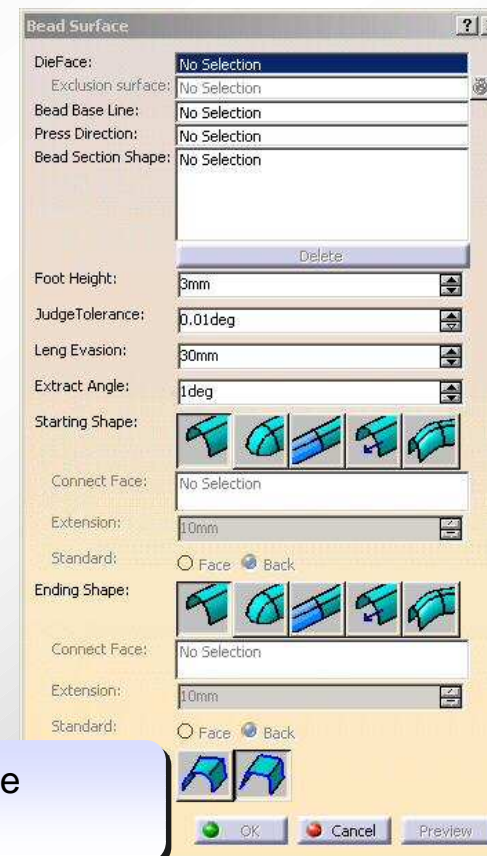
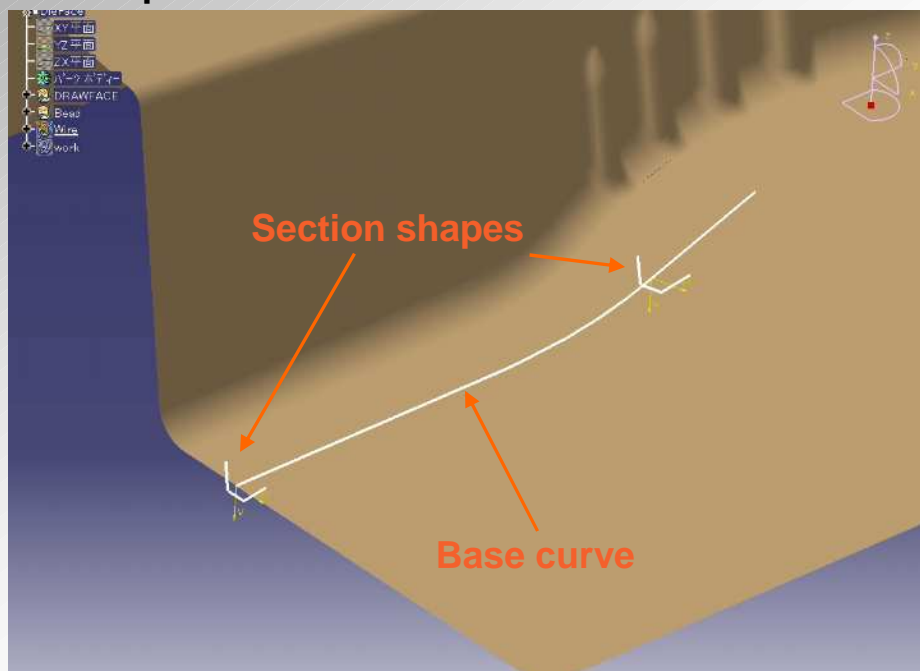
- Sub-elements of specified feature are the target.
- Direction can be specified to determine side of the surface.





Bead

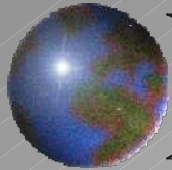
- Creation of a swept surface along a bead base curve connecting two or more bead section shapes defined as a sketch.



A bead shape is created using a specified draft angle where the angle makes a back draft to the press direction.

The bead shape is smoothly transformed at portions adjacent to the back draft portion.

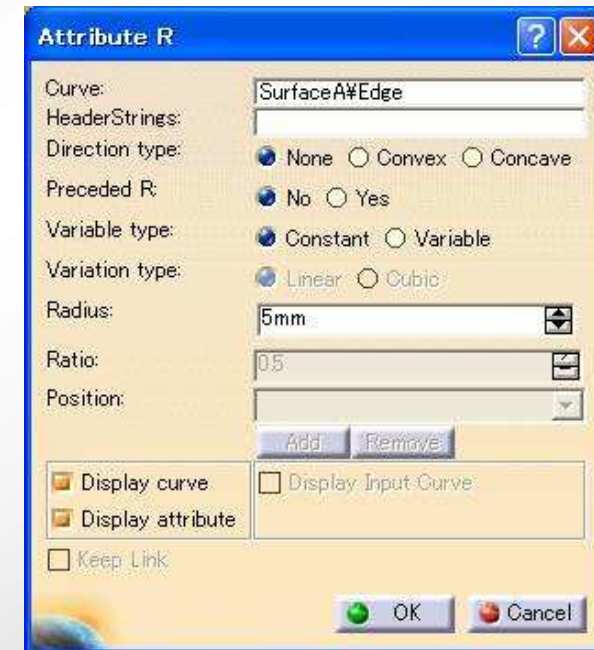
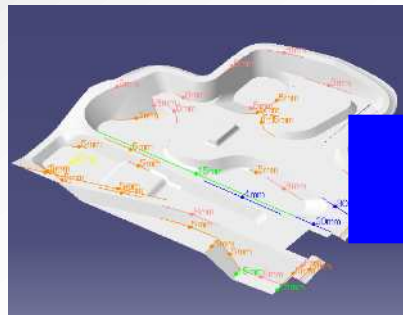
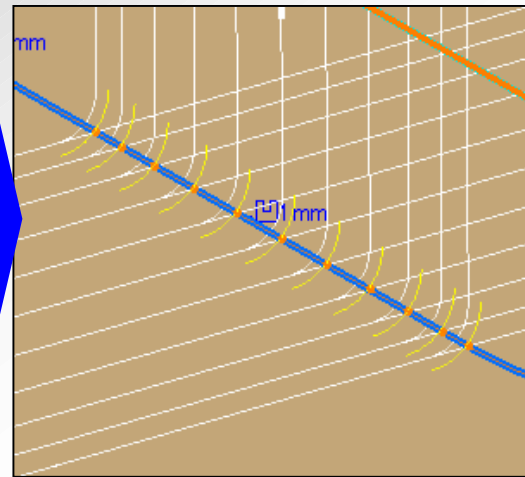
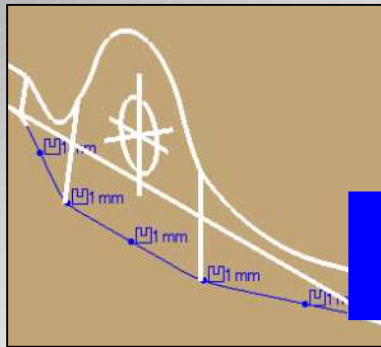


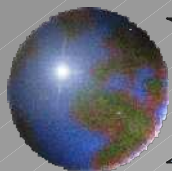


R attribute

-Actual R values are defined as attributes of a curve or an edge.

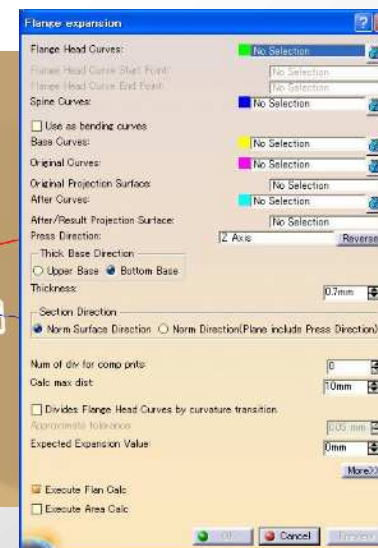
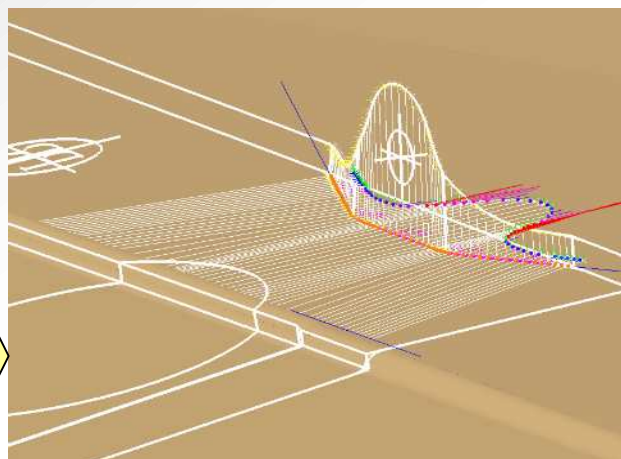
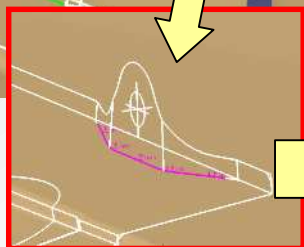
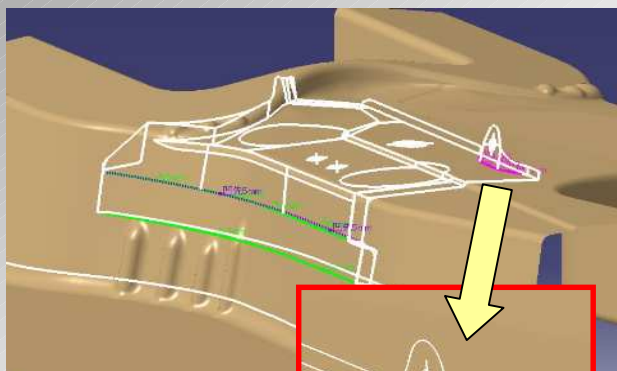
Defined R attributes are used for Flange expansion of Die face or R value of Formability Fillet.



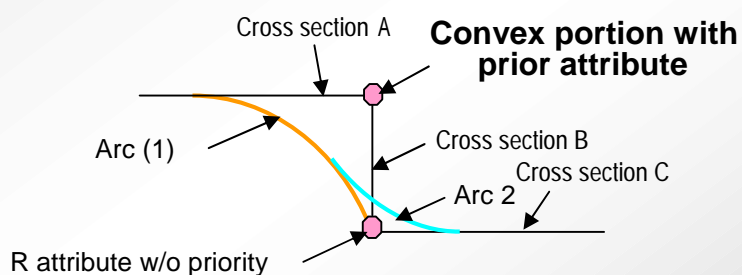


Flange expansion / R attribute

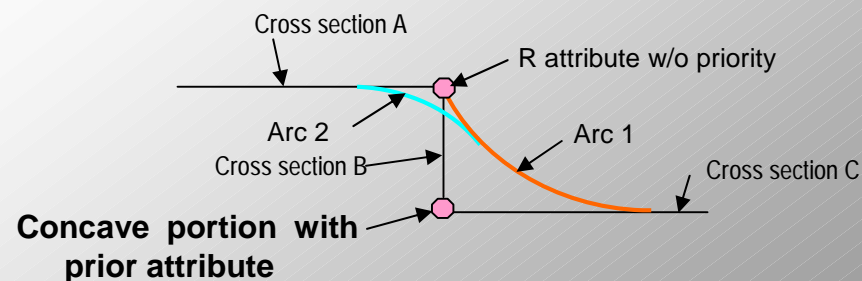
- Mapping of a trim line onto addendum shapes and die face surface in die process design. Flange expansion is possible even if bending curves are complex.



- Prior R attribute



Convex first



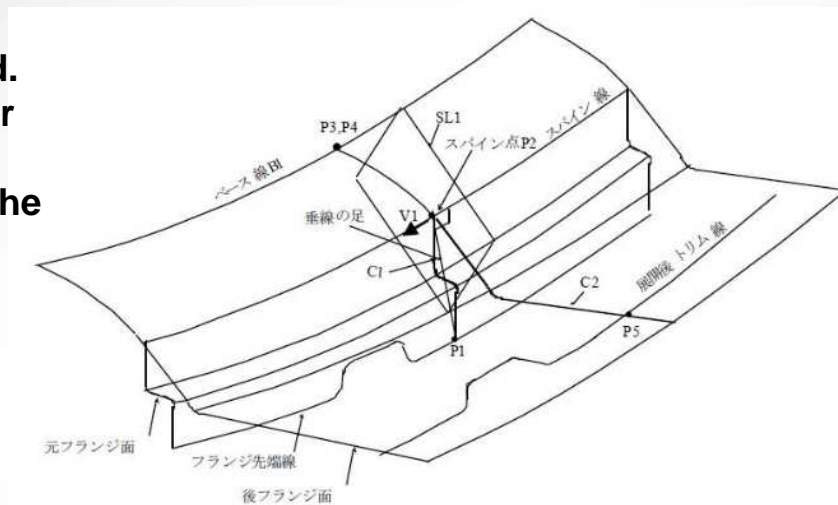
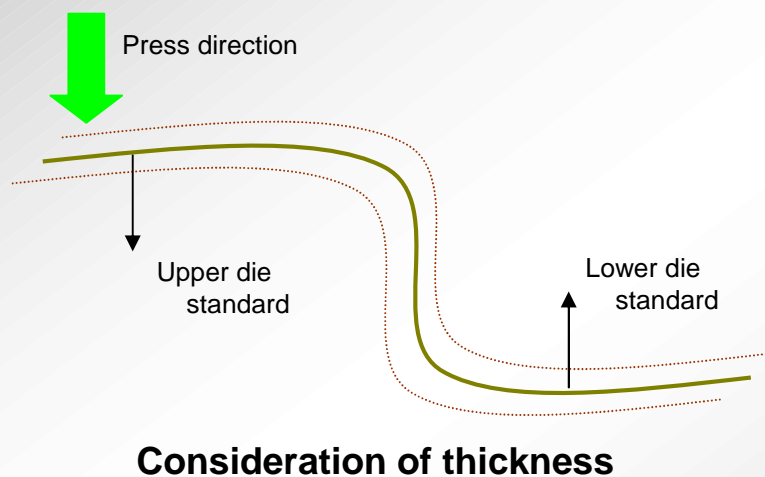
Concave first





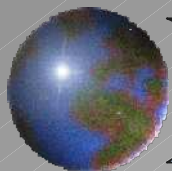
Flange expansion / Surface

- Flange expansion for a surface.
- A shape which has fillets can be processed.
- Two or more target surfaces are allowed for expansion.
- Area difference is also available (same as the curve version).



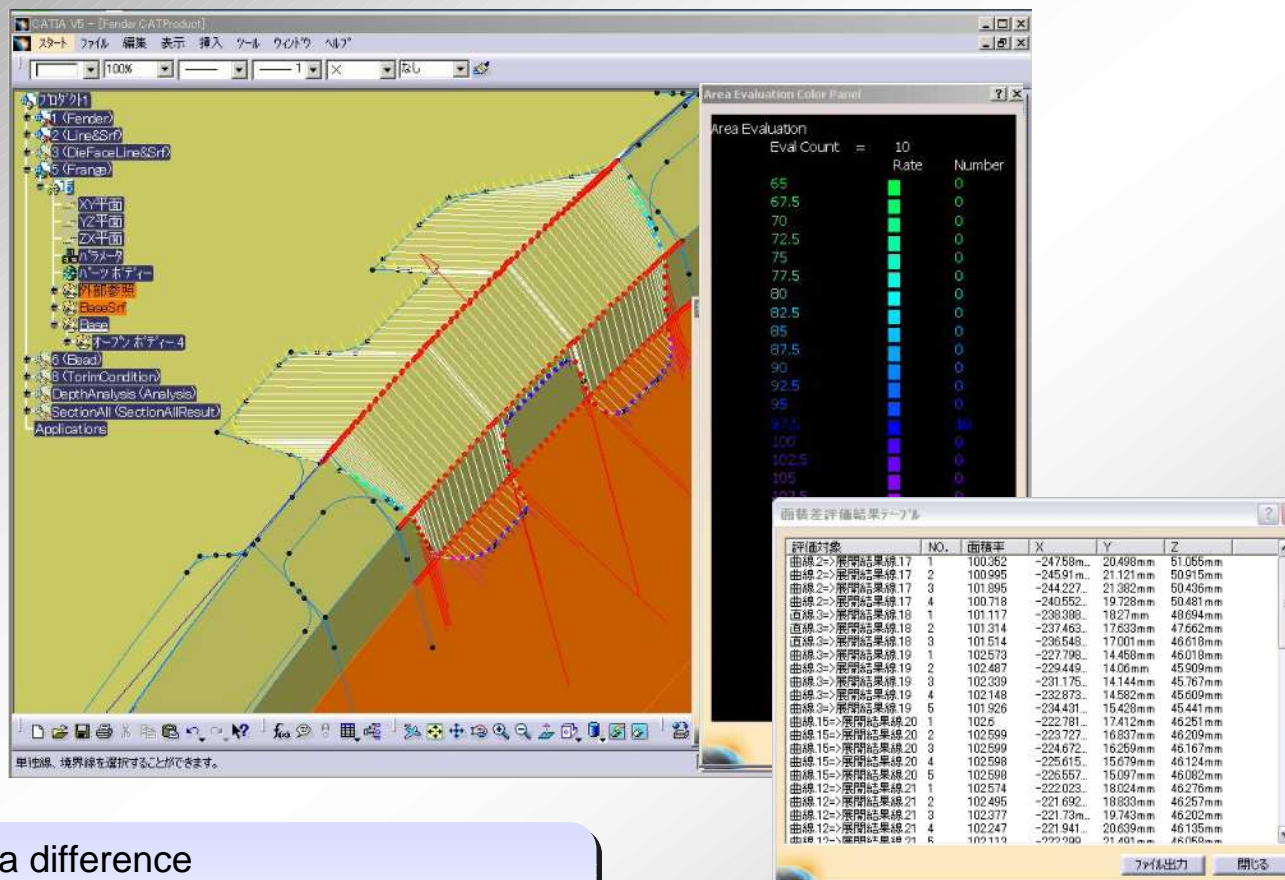
- Selection of the standard, upper die or lower die.
- Creation of a neutral line and a neutral axis.
- Input of product thickness (from 0 to 50 mm)
It will be used as an offset value (thickness x 0.5) at the calculation of actual length of the neutral axis.





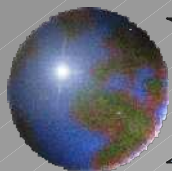
Flange expansion - Area difference

- Calculate extension rate by area difference.



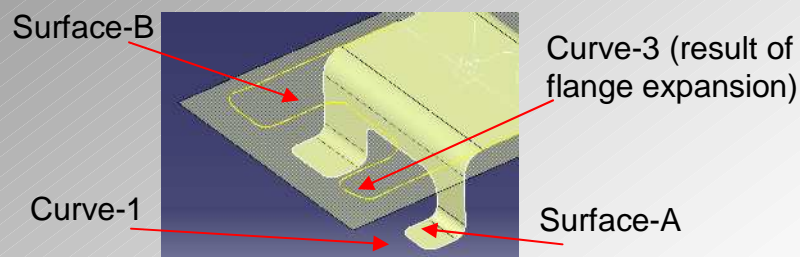
- Area difference
Calculation of area difference between before and after the flange expansion





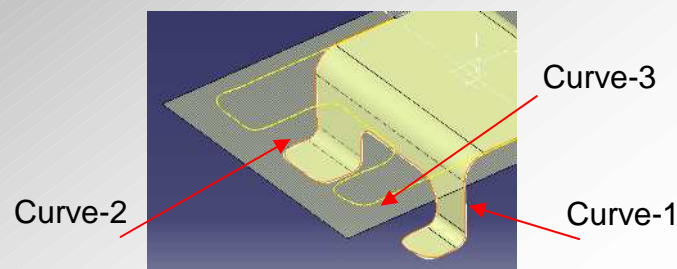
Flange expansion – Correct flange expansion

- **Correct flange expansion curve reflecting the measured data or CAE results.**

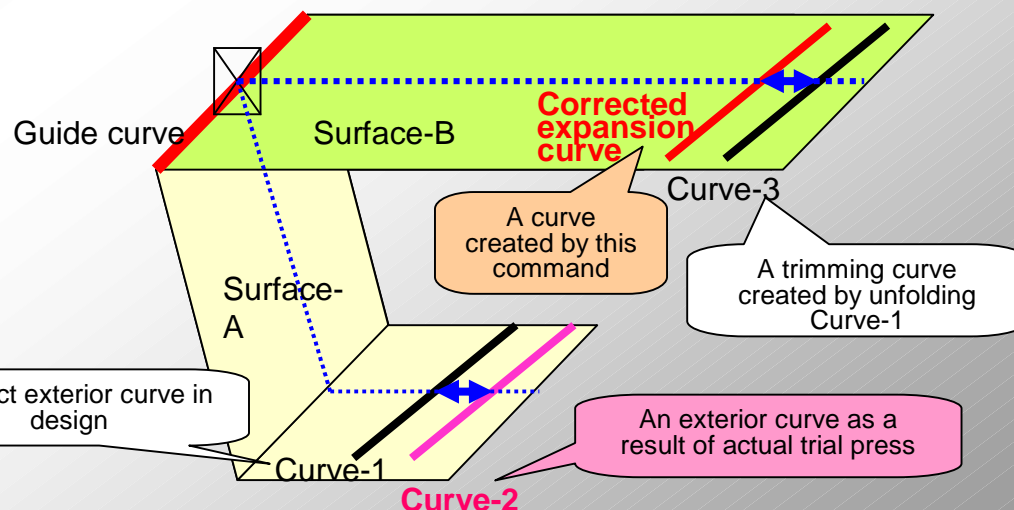
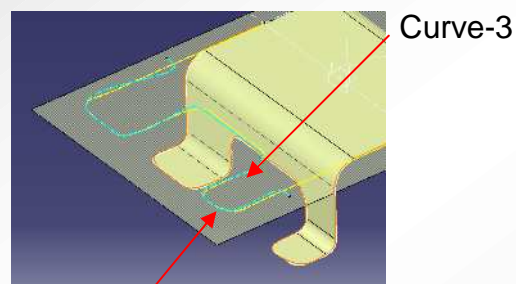


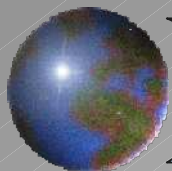
1. Create Curve-3 by unfolding Curve-1 in Dynavista Flange expansion command.

2. Calculate Curve-2 and import as a curve.



3. By Correct flange expansion command, the difference between Curve-1 and Curve-2 is reflected to Curve-3 and Corrected expansion curve is created.





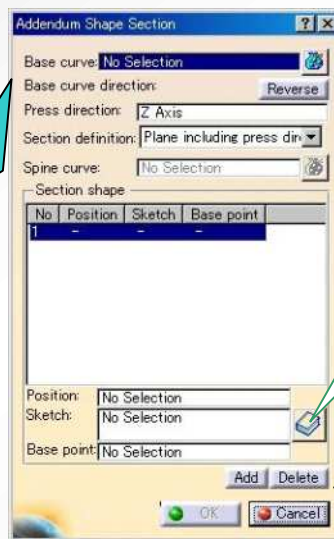
Addendum section

- An addendum section shape (sketch) is copied to a specified location on a surface boundary of a product shape. A curve feature is also accepted. It is possible to copy different addendum section shapes to two or more locations.

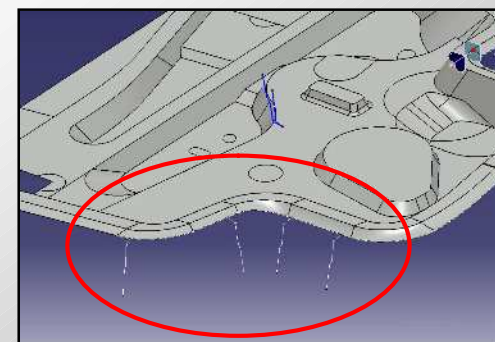


Section shapes saved in a catalog after making them a know-how.

Not place but copy



Sketch can be selected from the catalog

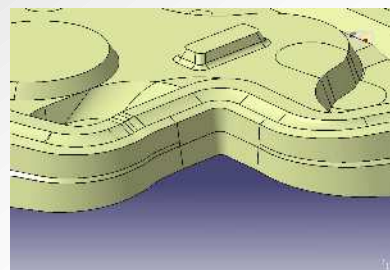
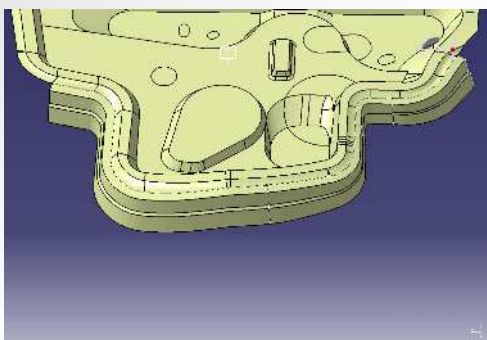
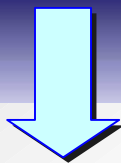
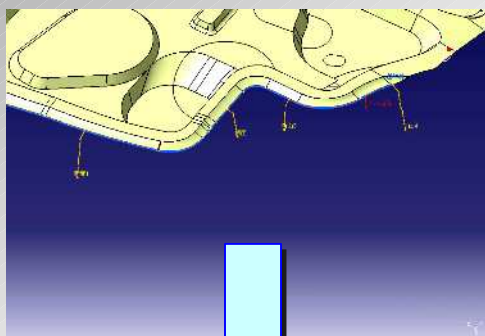


Two or more sketches are located.

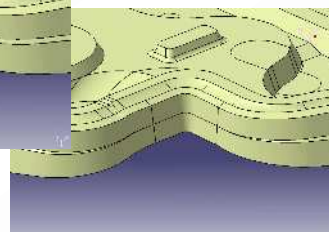


Addendum surface

- A swept shape can be created along a base curve after connecting two or more addendum sections. Smoothly varied surface between two sections is created if adjacent addendum section shapes are different.
- The surface is created automatically removing gouges if it contains self intersections.



Variable surface can be changed by specifying "combination".



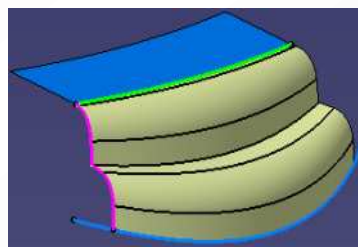
Cross section
definition surface



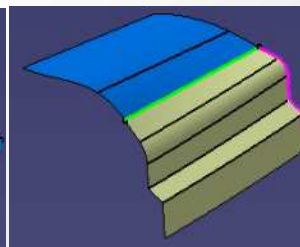
Connection



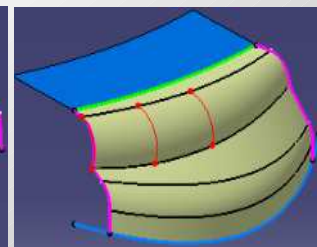
Tangent
continuity



Arcs and free
curves can be
specified as a
cross section
curve.

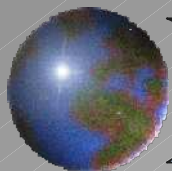


Tangential
connection to
a product
surface is
possible.



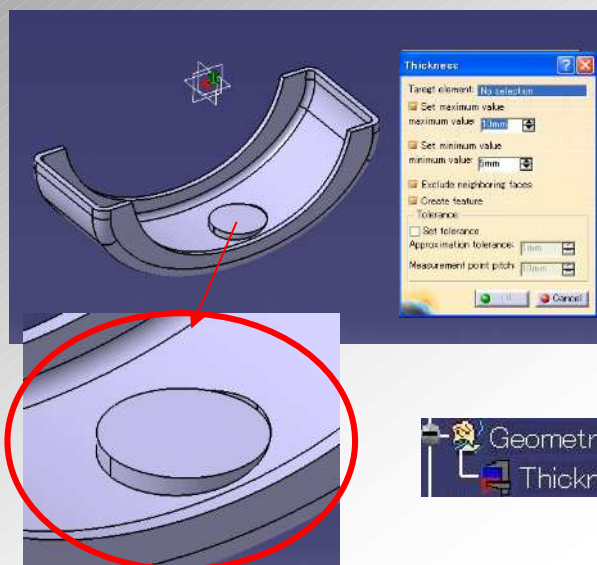
Temporary
display of
back drafts is
possible





Thickness

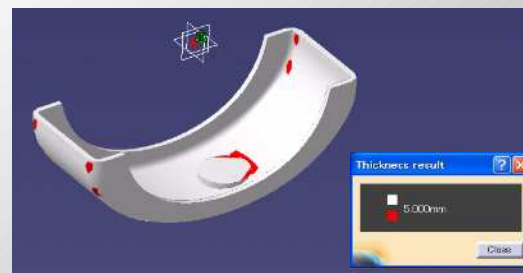
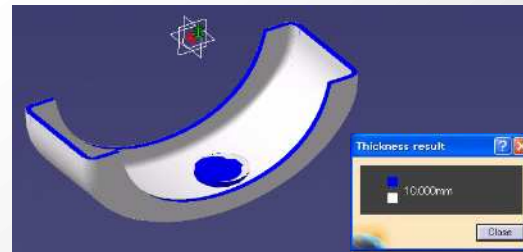
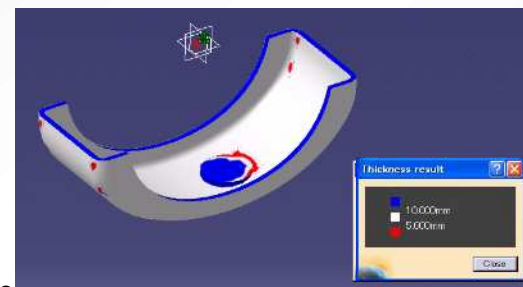
Thickness of a solid is shown with a color map.



Specification of maximum and minimum thickness value

Only maximum value is specified.

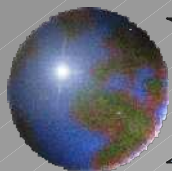
Only minimum value is specified.



- Maximum and minimum thickness can be specified.
- Check result is saved as a feature and can be seen afterward.
- Detail condition (approximation tolerance, measurement pitch) can be specified.

Thick of thin portion of a product or a die (effective for a forging die) can be detected. A range can be specified by maximum and minimum values.

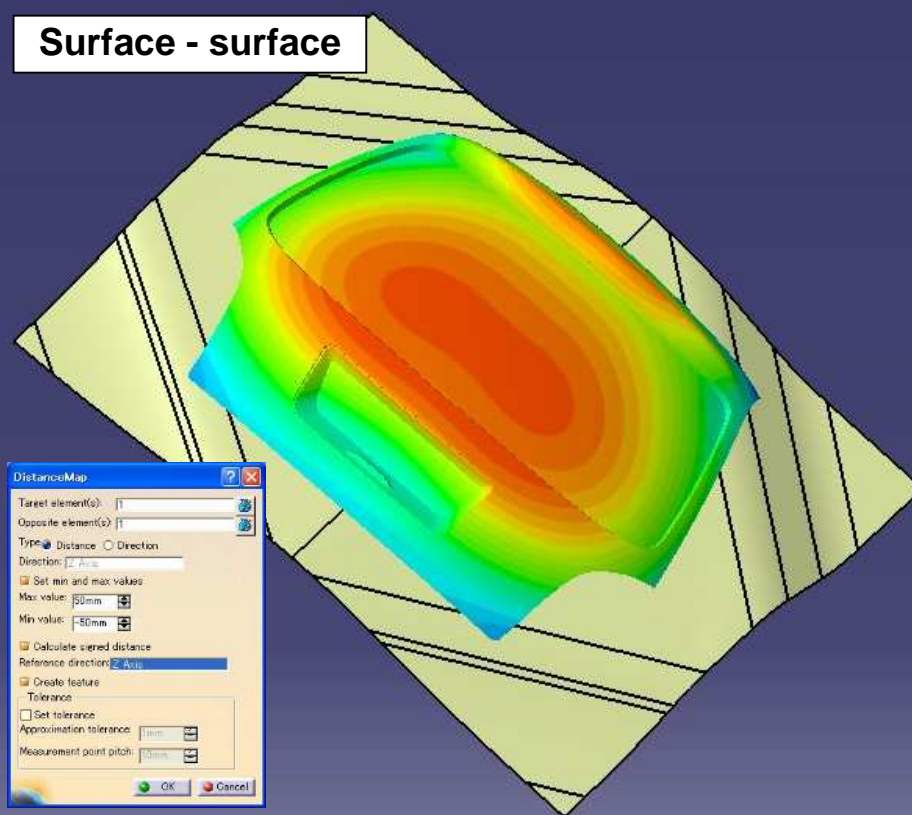




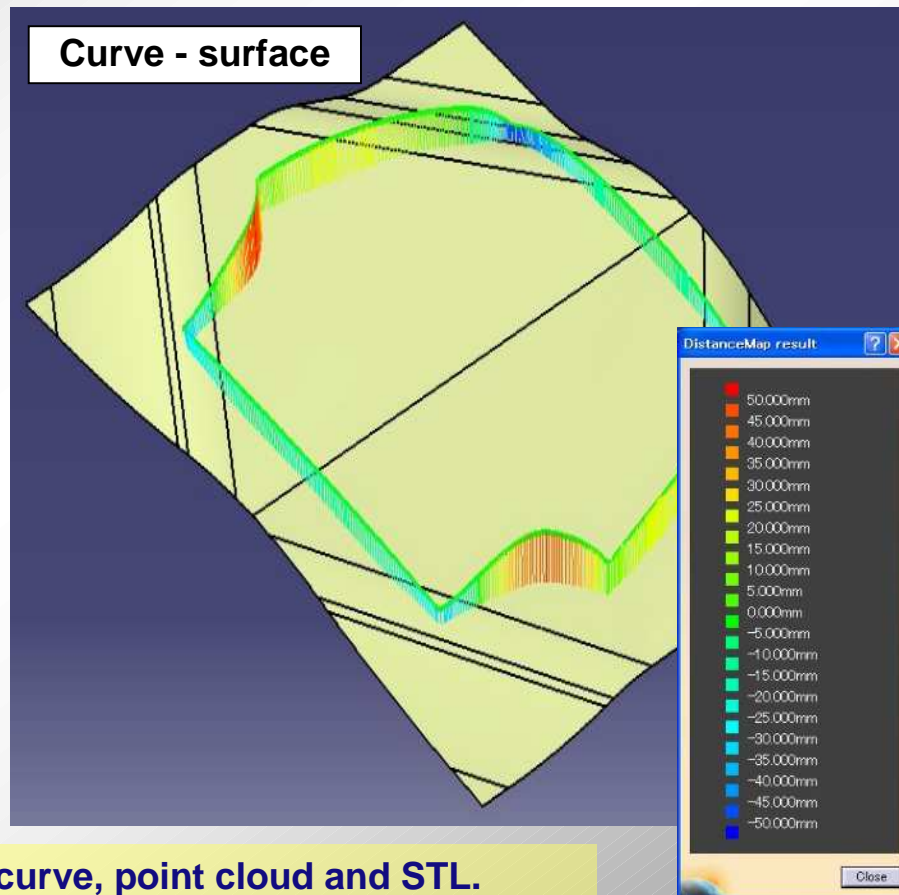
Distance Map

Distance from a target object to another is measured and its result is shown on the object with gradation.

Surface - surface



Curve - surface



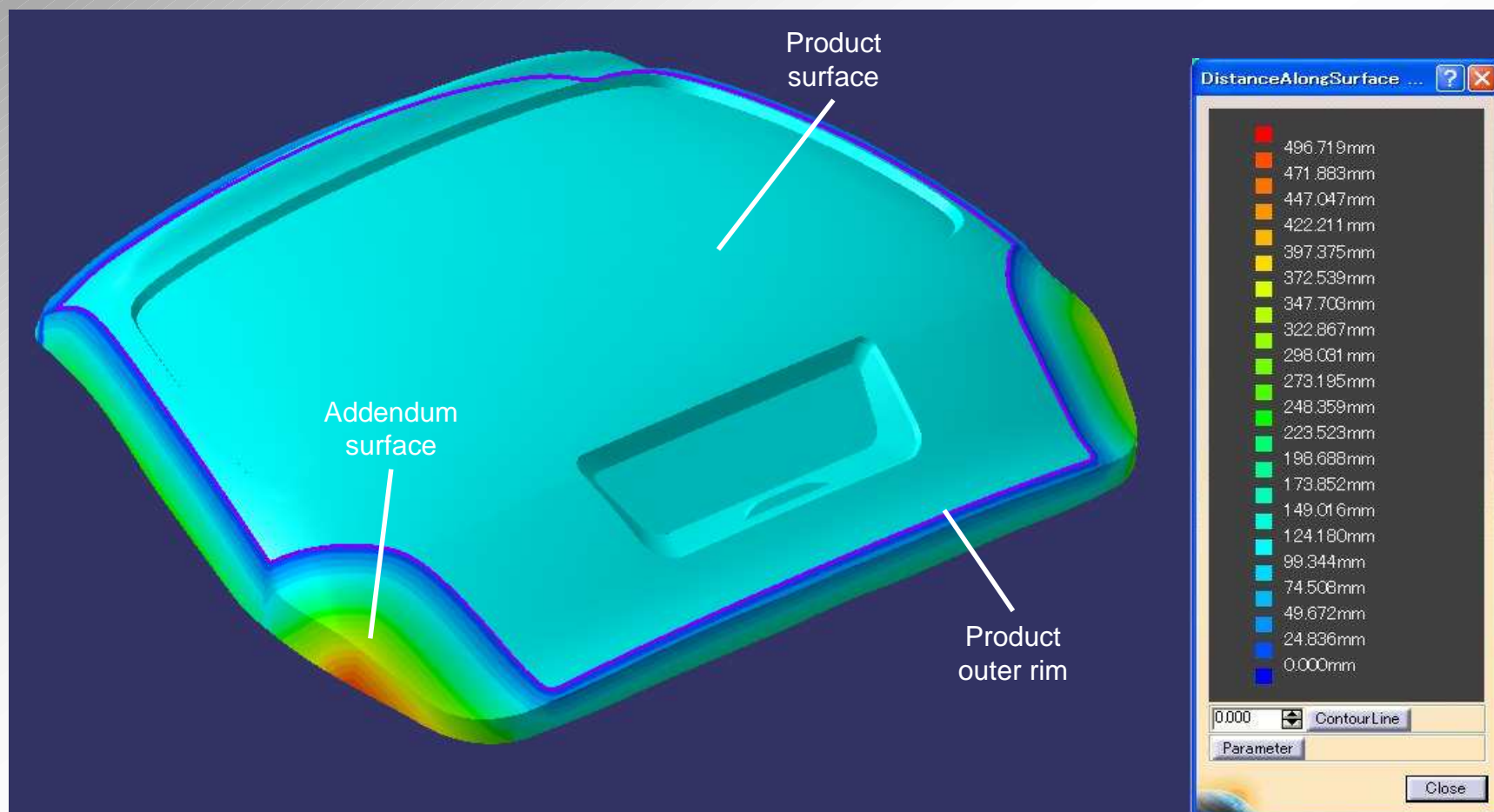
- Check can be made for solid, surface, curve, point cloud and STL.
- Positive distance is shown if direction from the target object to another is same as reference direction, otherwise negative distance is shown.





Distance along surface

- A color map is displayed varying its color depending on the distance between a test surface and a curve.



- Distance along a surface from a base curve will be displayed on a product surface or a machining surface.

Color map

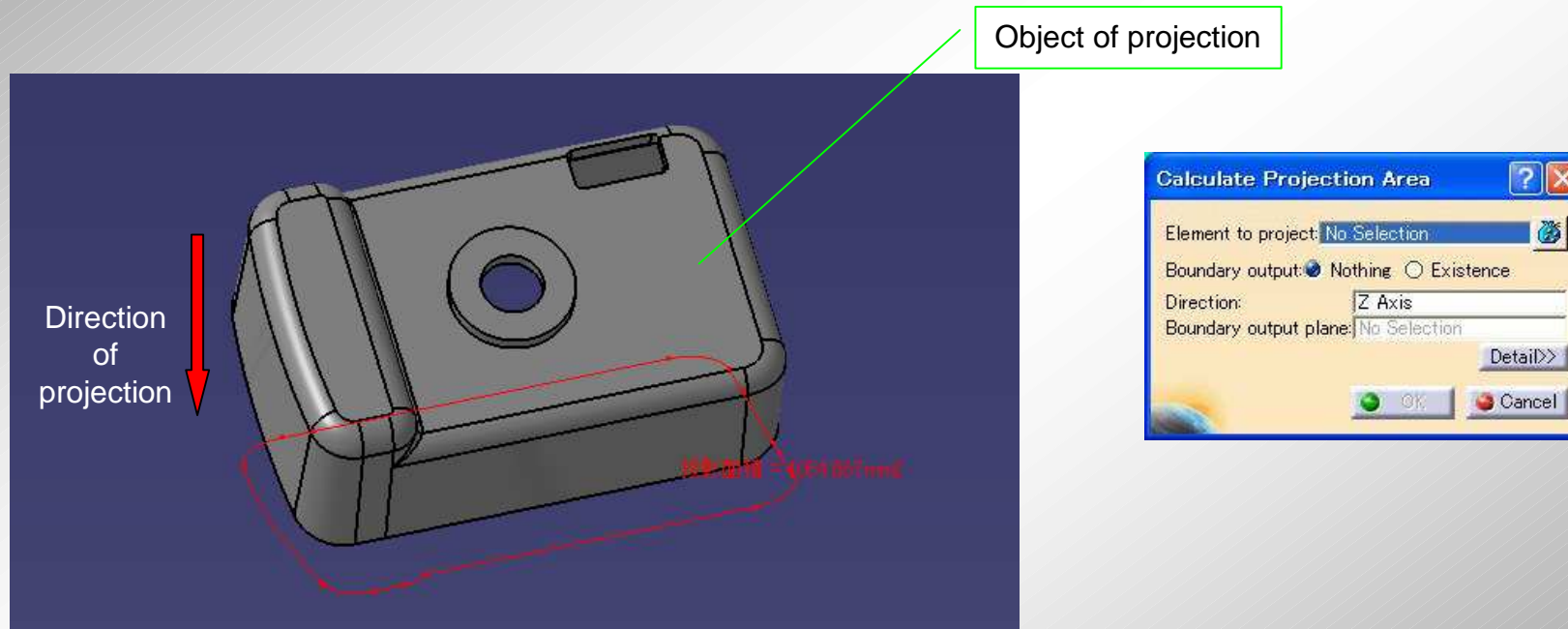




Calculate Projection Area

Projection area of a product is calculated by specifying a solid (or a composite surface) and projection direction.

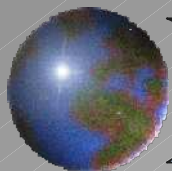
An area is calculated surrounded by outermost boundaries or outermost edges along with specified projection direction. Projected outermost curves and area will be shown as a temporary figure.



- High robustness (The solution is secured.)

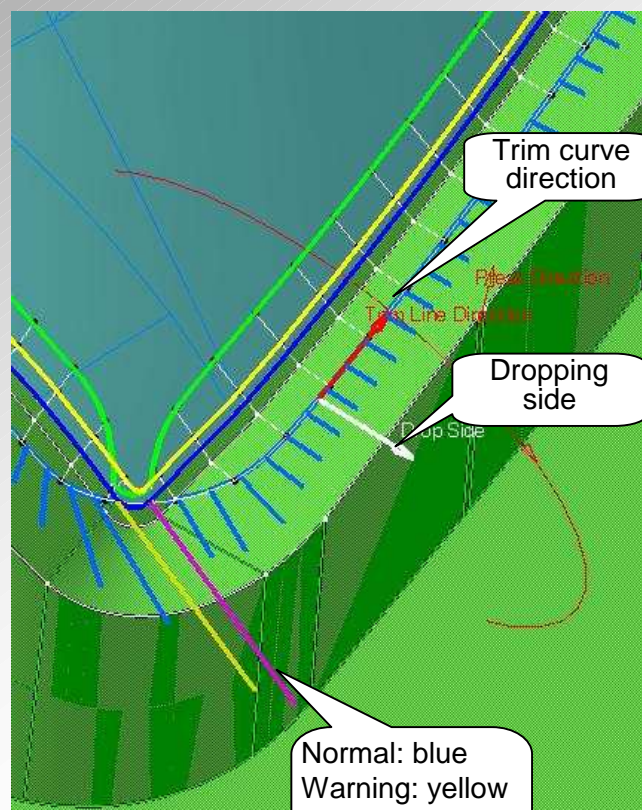
(Dynavista uses polygons for projection. The result is securely obtained since inside or outside is judged by lattice points.)





Trim condition

- Condition of a trim curve and its surrounded portion is evaluated by giving the trim curve, surfaces and a press direction.
- A list of evaluation result (advancing angle which is an angle of the trim curve along the curve, trimming angle which is an orthogonal angle and judgment result) is also displayed.



Normal: blue
Warning: yellow
Danger: purple
Vertical: red

TrimCondition Evaluate Command			
No.	Direction...	Extracts...	Result
120	-34.03deg	10.475deg	OK
121	-35.187d...	10.004deg	OK
122	-36.423d...	9.451deg	OK
123	-37.633d...	8.788deg	OK
124	-38.893d...	7.883deg	OK
125	-39.952d...	7.577deg	OK
126	-40.648d...	8.559deg	OK
127	-39.901d...	13.17deg	OK
128	-35.482d...	22.201deg	OK
129	-30.666d...	28.931deg	OK
130	-16.311d...	43.59deg	Warning
131	-15.821d...	46.161deg	Danger
132	11.182deg	13.092deg	OK
133	12.003deg	13.555deg	OK
134	16.047deg	11.089deg	OK
135	20.17deg	8.084deg	OK
136	23.104deg	6.716deg	OK
137	24.092deg	6.306deg	OK
138	23.803deg	5.986deg	OK
139	23.396deg	5.793deg	OK

TrimCondition

Target Face: Surface58

Trim Line: FlanCrvCurve.5

Drop Side: Reverse

Trim Line Direction: Reverse

Section Type: Norm Plane Type

Press Direction: Axis System.4#Z Axis

Pitch: 5mm

Length Parm: 1mm

EvaluateAngle

Warning Angle: 30deg

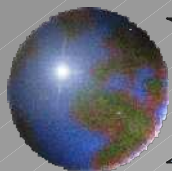
Danger Angle: 45deg

Maximum Projection Distance: 1mm

OK Cancel Preview

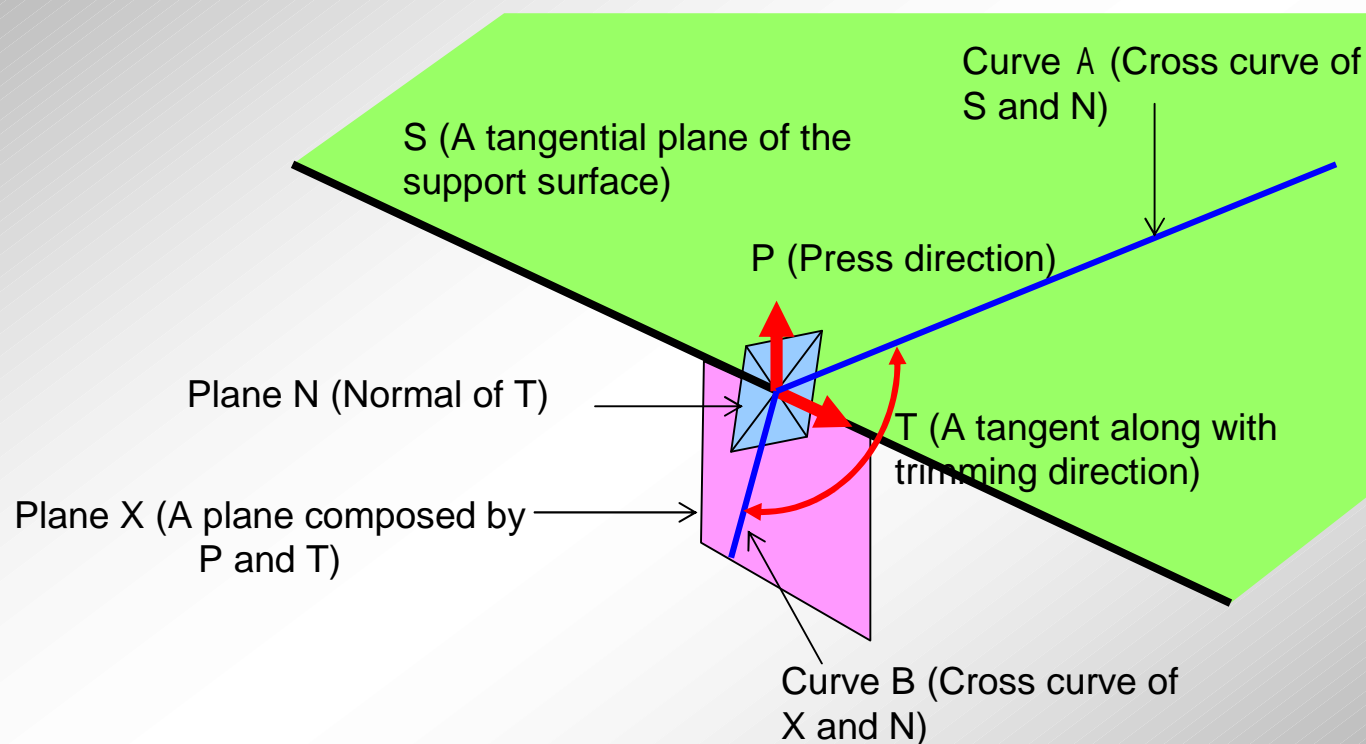
Safe: OK
Warning: Warning
Danger: Danger
Vertical: Norm





Trim condition

Hem flange angle measurement



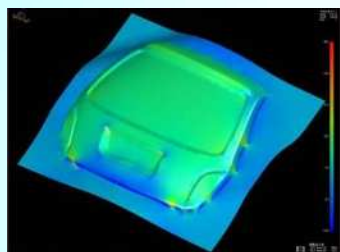


CAE Association / Objective

High quality formability compensation by the use of simulation

Early improvement of design quality

CAE



Panel formability simulation

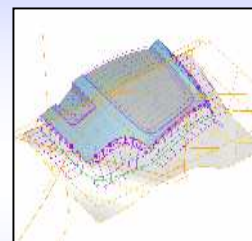
Designer CAE

Use of analysis result on CAD

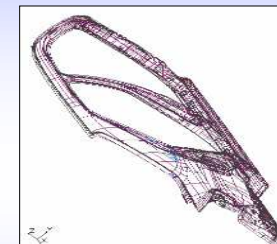
Association with formability deformation

Dynavista

Die Layout Design



Simple and quick checks assist designer's judgment

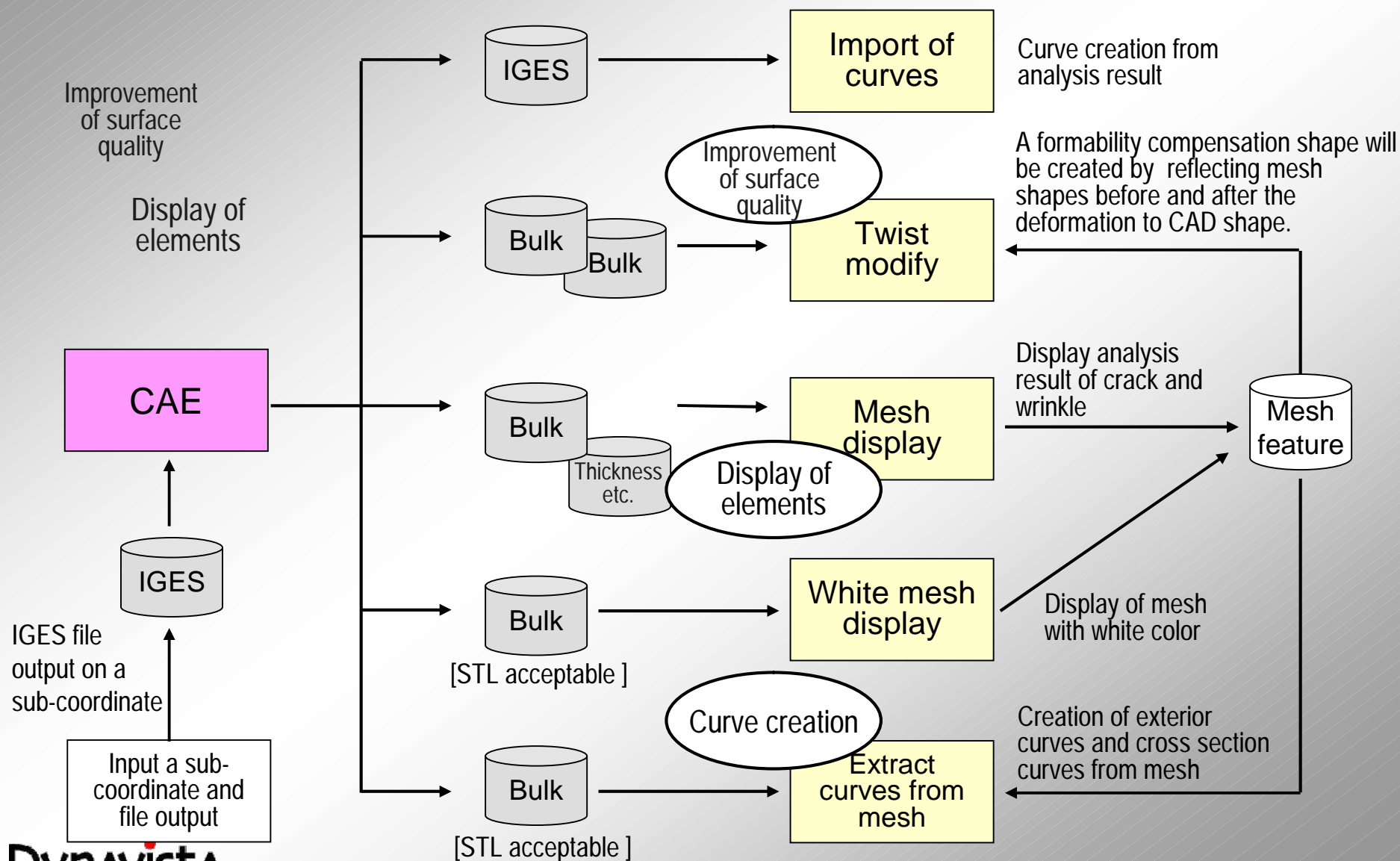


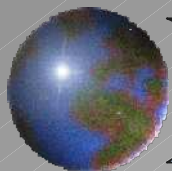
Advanced functions for die modeling (Formability compensation, improvement of productivity)





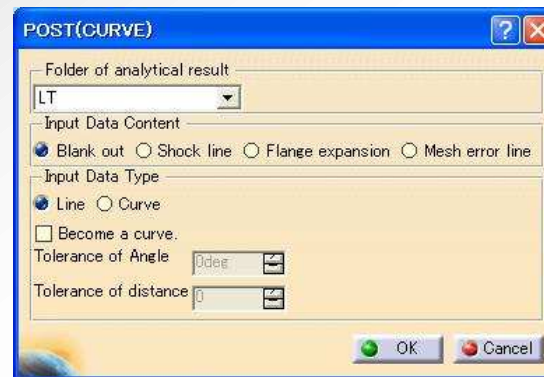
CAE Association



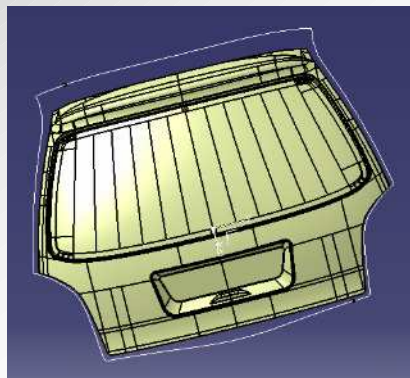


POST (Curve)

- Import curve segments or curves which are simulation result of CAE as CAD curves.
- As for blank unfolding and flange expansion, curve segments can be merged into curves at the import.

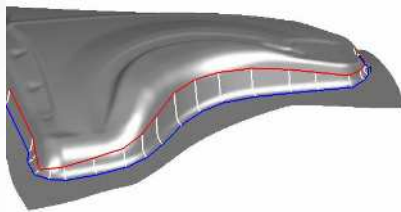


Blank unfolding



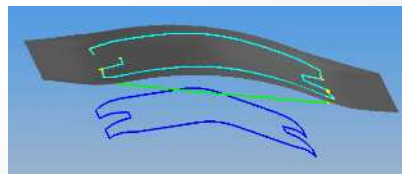
Blank unfolded curves are imported. Can be merged into curves.

Shock line



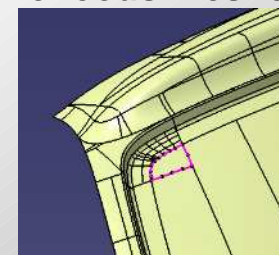
- Contact front edge
 - Trace
 - Contact evaluation curve
 - Contact rear edge
- are separately imported.

Flange expansion



Flange expansion curves are imported. Can be merged into curves.

Outer curve of erroneous meshes



Boundary curve is imported for the portion where mesh cannot be created successfully by CAE solver.

Blank unfolding

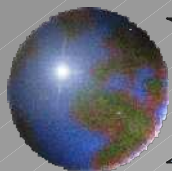


Shock line



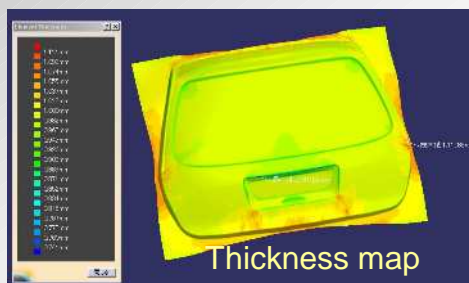
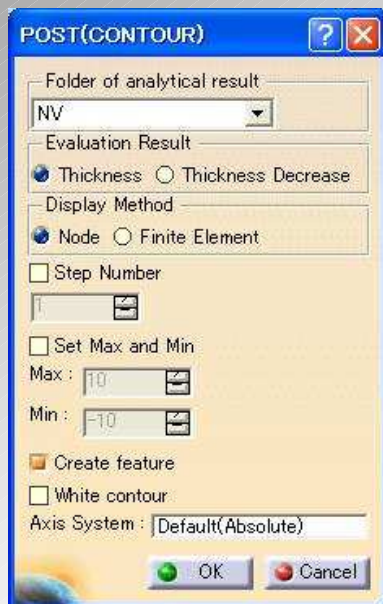
Flange expansion



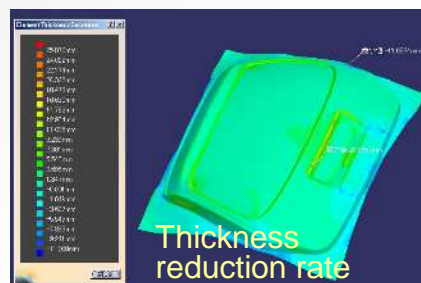


POST (Contour)

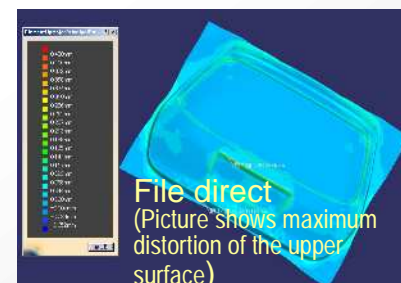
- Overlay display of the CAE simulation result for cracks and wrinkles on CAD model.



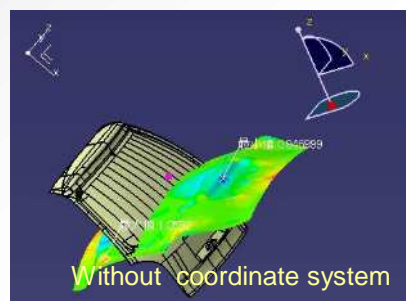
Thickness map



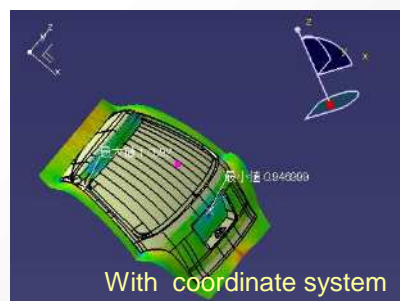
Thickness reduction rate



File direct
(Picture shows maximum distortion of the upper surface)



Without coordinate system



With coordinate system



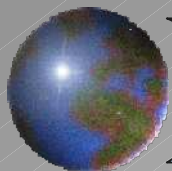
White contour display

- Contour display of various analysis results for judgment of cracks and wrinkles
- The analysis result will be preserved as a history.
- A coordinate system can be specified at the import.
- Step numbers can be specified.



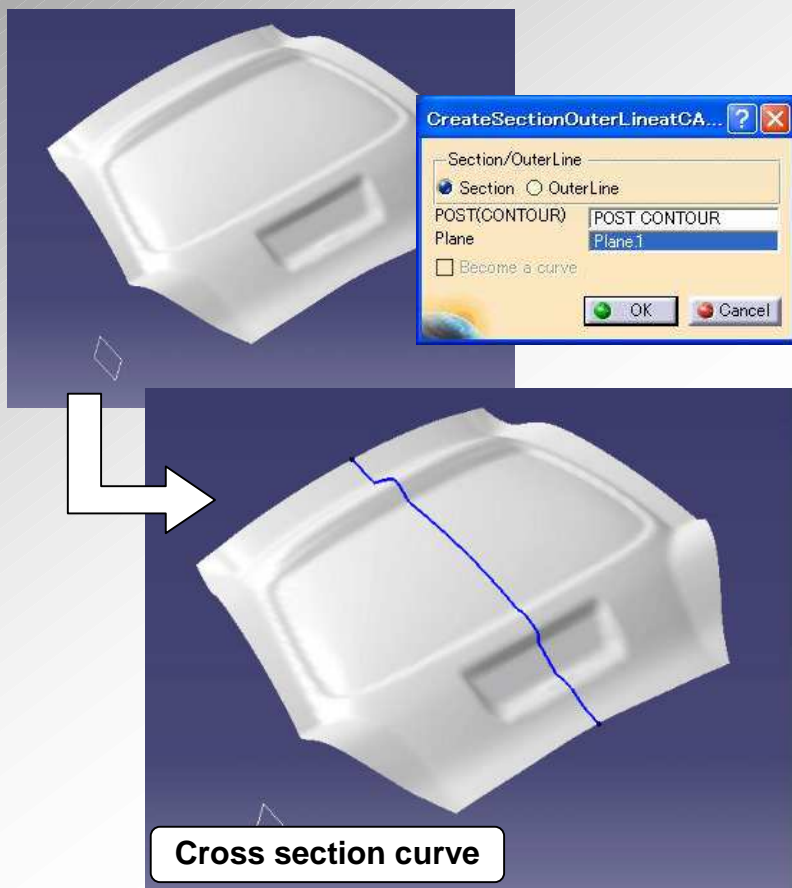
Color map





Association with CAE – Cross section / outer curve

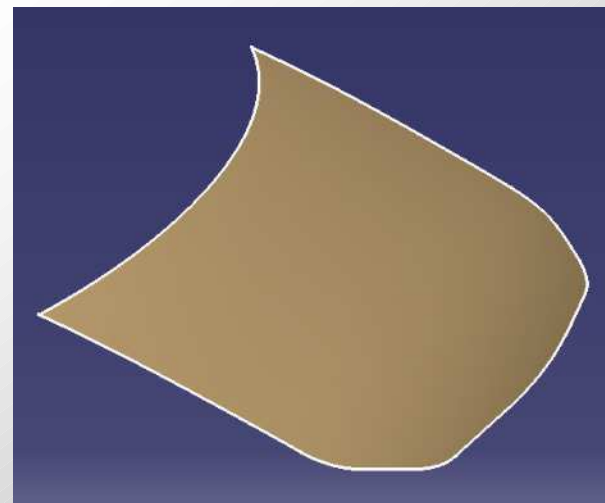
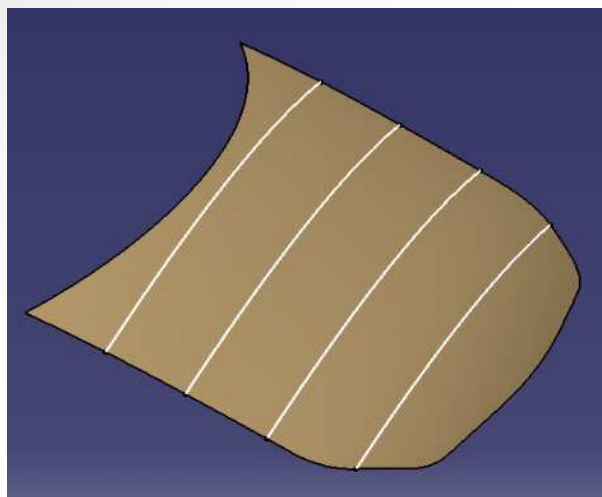
Cross section curves and outer curves are extracted based on a POST (CONTOUR) feature.

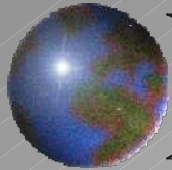




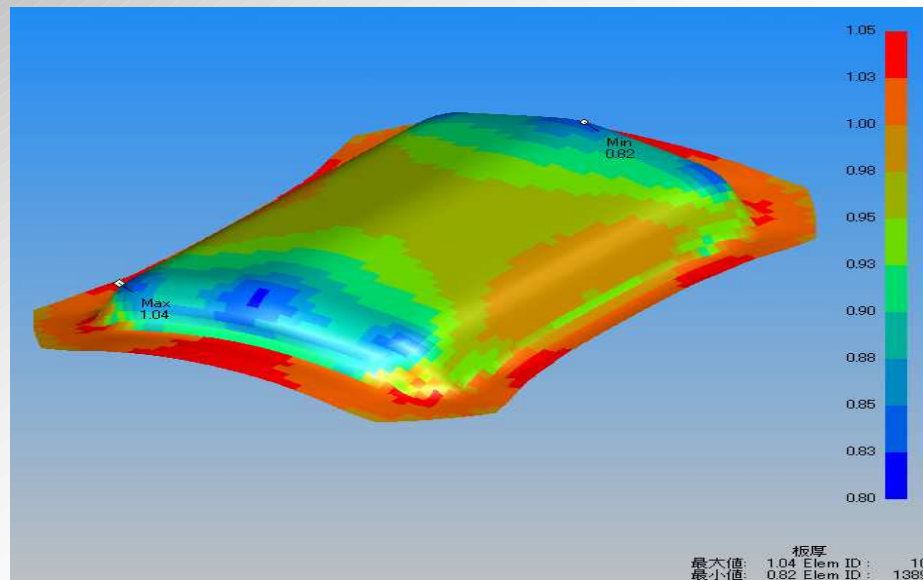
Extract curves from mesh

- Create a curve for a cross section curve of a mesh and a plane or an exterior curve of the mesh.



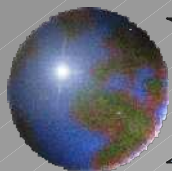


Color painting of each element



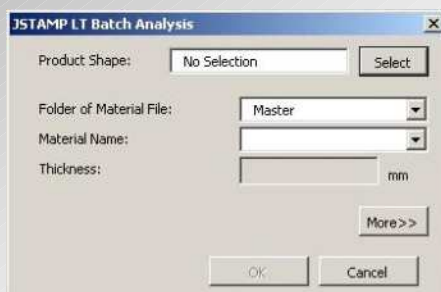
Color painting mode will be changed from node value to each element wise.

Currently even though element values are given for thickness value and so on, they are converted to node values for color map.

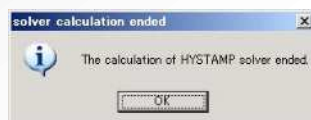


JSTAMP LT batch analysis

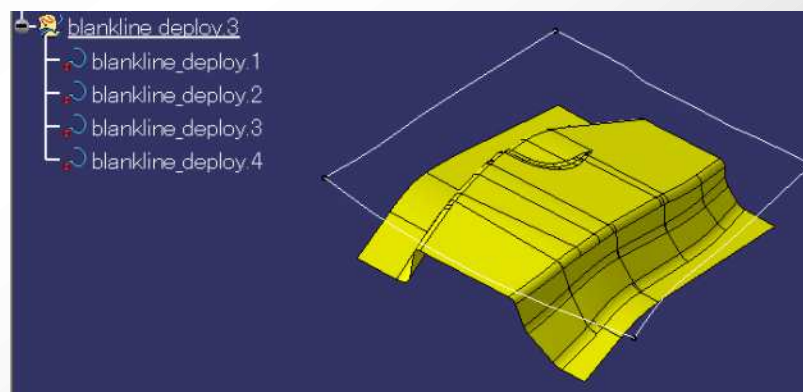
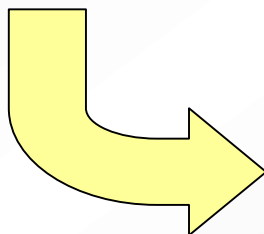
**When JSTAMP LT is used as a CAE solution, direct activation is possible for an analysis of exterior curve of blank expansion, thickness distribution and thickness reduction rate.
No CAE operation is necessary for a CAD user.**



A batch task is activated from Dynavista, then input material name and thickness.

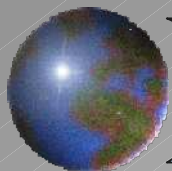


Calculation completed



The result can be imported by the used of POST(CURVE) and POST(CONTOUR) command.





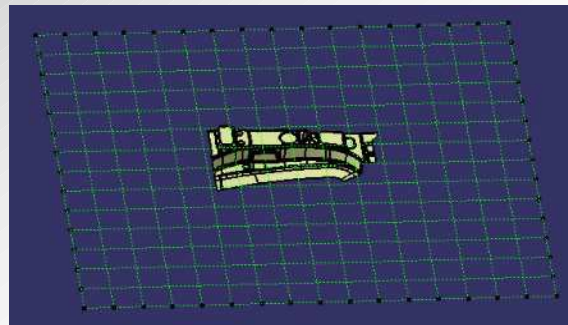
Layout support

Data preparation

Layout review/design

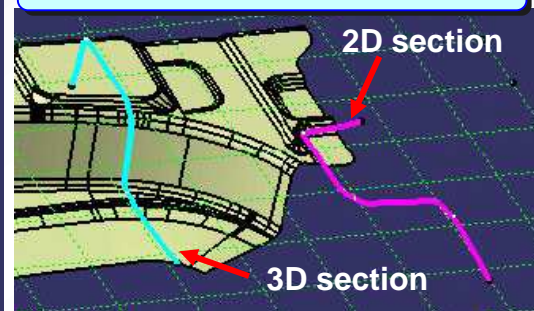
Primary press direction

Layout support/3D guide line



Layout support/3D section

Layout support/2D section

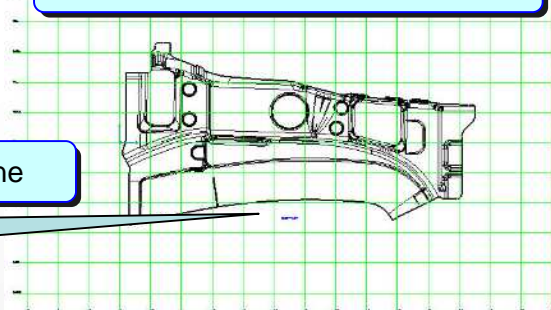


3D shape



Drafting

Layout support/2D guide line



Layout support/auxiliary guide line

W15R→W1R

Drawing



3D Guide line creation

Guide lines will be created on a 3D model.

Create 3D support line

Coordinate system: Absolute coordinate system

Support plane:

Standard plane: XY Plane

Height: Plane

Plane: xy plane#Face

Range of making:

Starting point (-500.000 mm , -500.000 mm)

Ending point (500.000 mm , 500.000 mm)

Pitch: 100 mm

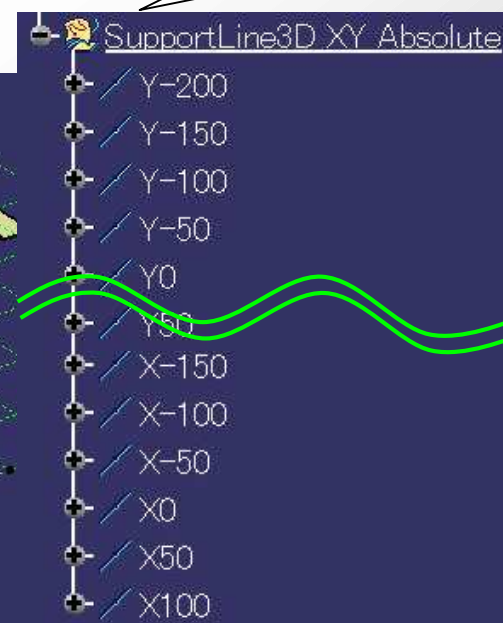
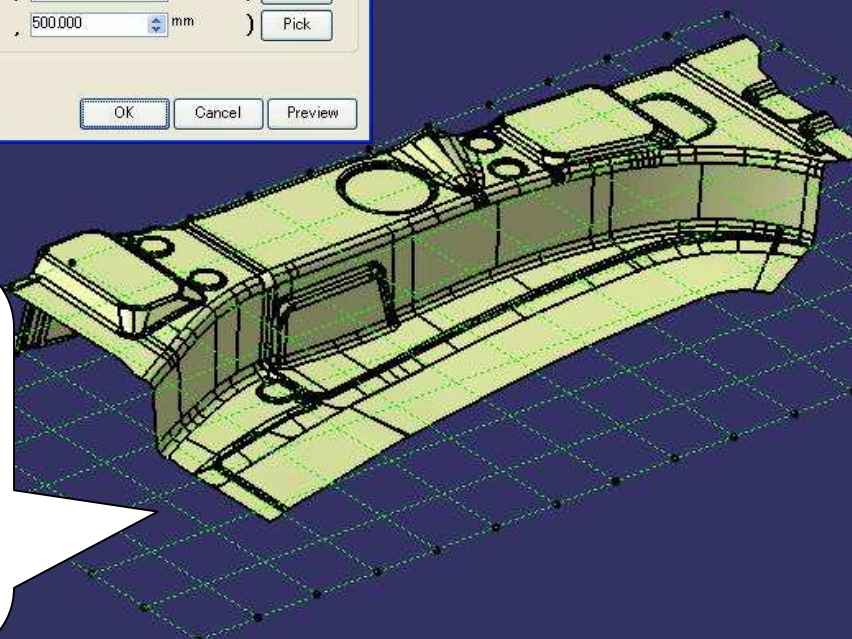
- Guide lines are created on basic planes (planes parallel to XY plane / YZ plane / ZX plane)
- Range and pitch can be specified.

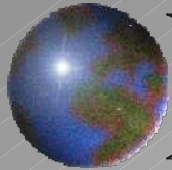
A new geometry set is created , then lines are created.

Guide lines are created in a

Selection Sets Selection

SupportLine3DSet_XY_Absolute

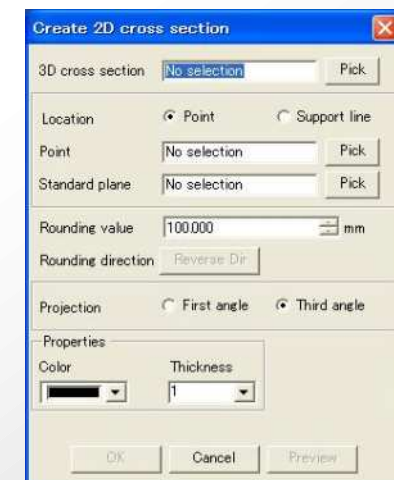
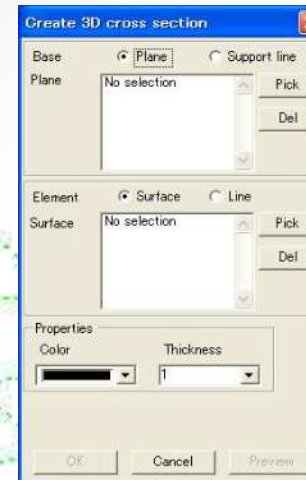
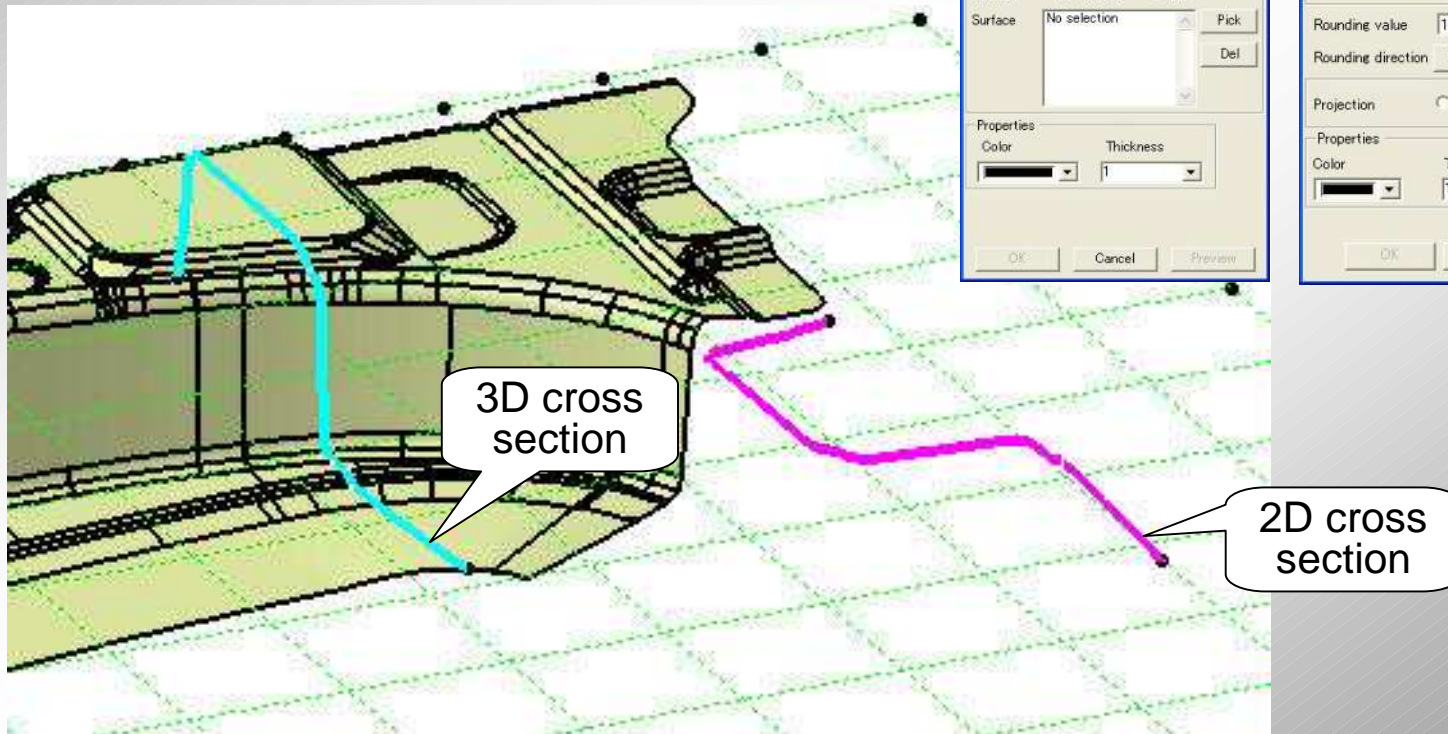




Creation of 3D cross section / 2D cross section

3D cross sections and 2D cross sections will be created in a 3D model.

- A cross section of a 3D model is output at a guide line or a plane. (3D cross section)
- The 3D cross section is placed on a plane parallel to the guide line (2D cross section)

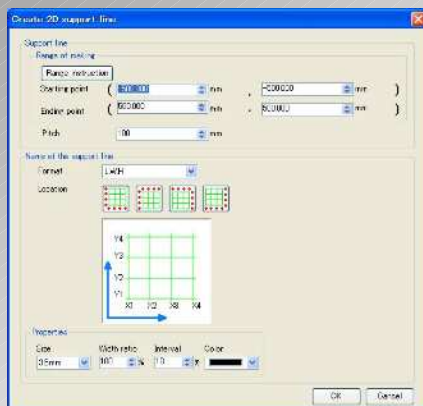




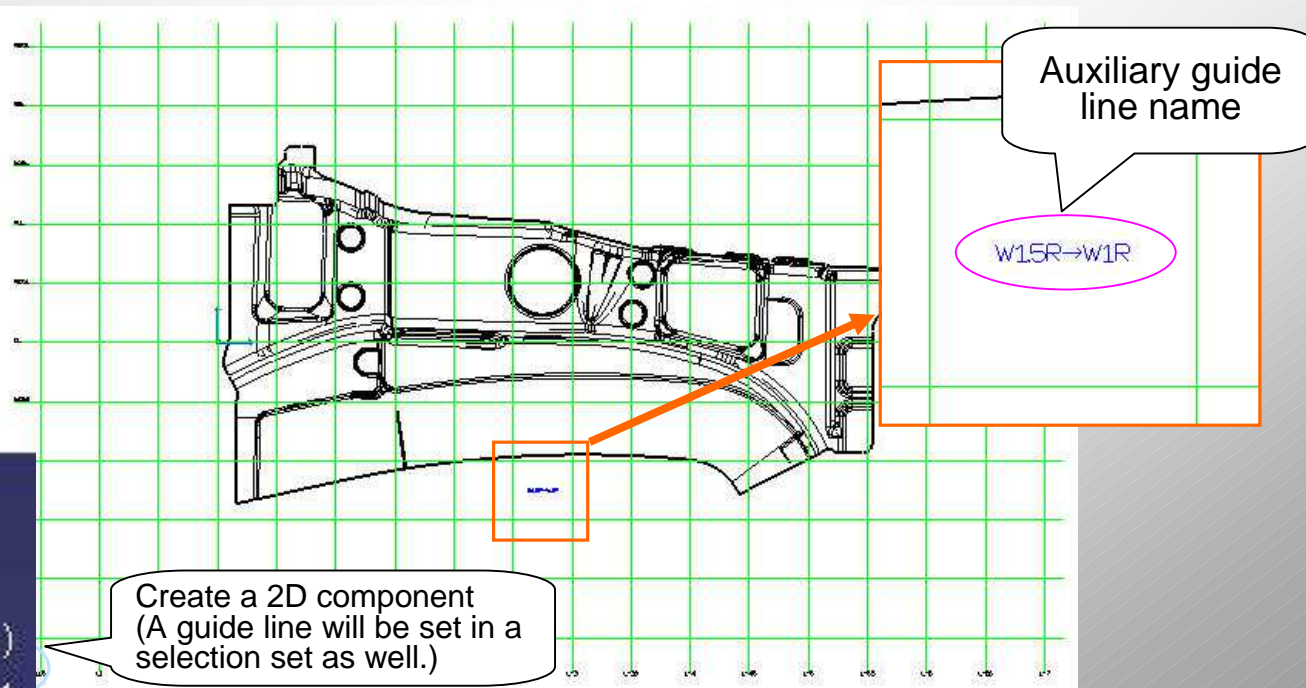
Layout support - Creation of 2D / auxiliary guide lines

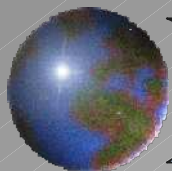


Create a guide line in a drawing (Drawing file), and input an auxiliary guide line name.



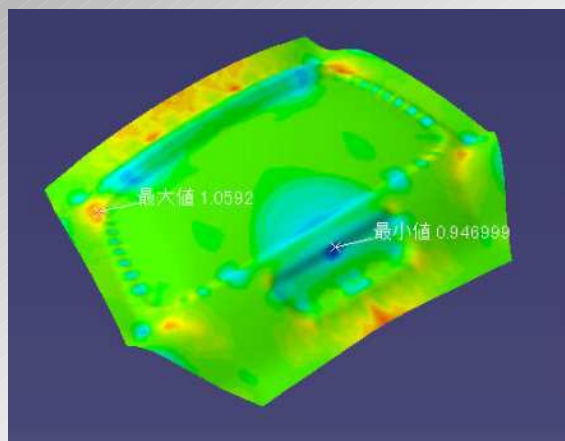
- Available in a workbench "Drafting".
- Guide lines like a lattice and guide line name will be created at a specified area in a drawing and with a specified pitch. (2D guide line creation)
- A text will be created as an auxiliary guide line name between adjacent guide lines. (Adjacent guide line creation)





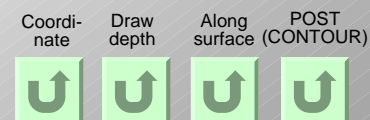
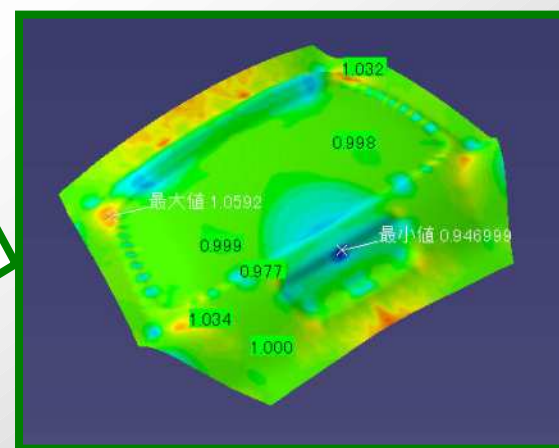
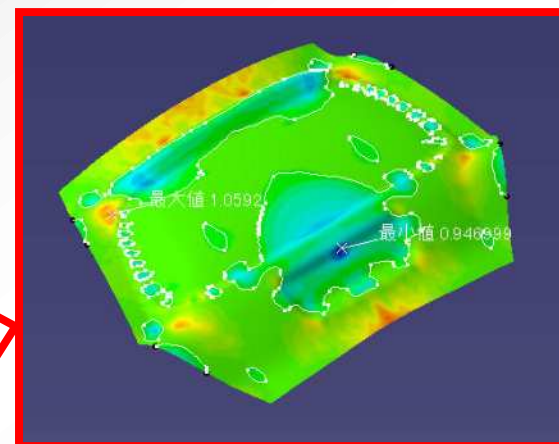
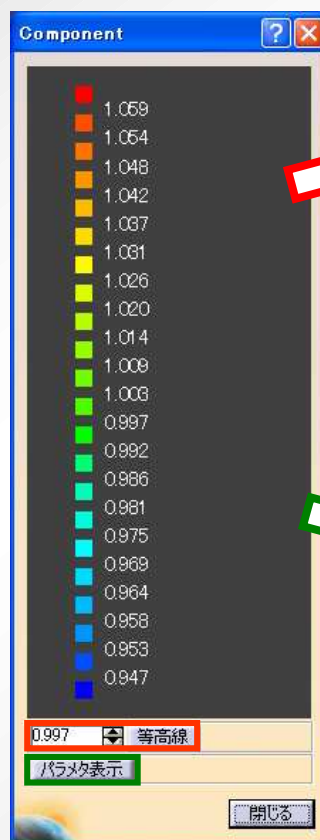
Color map

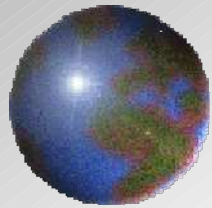
Contour display and parameters display are added.



Related command

- Die standard axis system
- Depth Measurement
- Distance along surface
- POST (CONTOUR)





Dynavista

CAA V5 based

<http://www.unisys.co.jp/e/dynavista/>

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