Die Structure Design

CAA V5 based
V10.2 / V11.2

October, 2011
UNIADEX, Ltd.
Die Structure Design automates stamping die structure design.

- Complex shapes are automatically created such as sectional die and back face, etc.
- Automatic creation of parts list enables association with BOM.
- CAM automation achieved by attribute association with 2.5D/3D CAM.

V5 prerequisites: MD2
Unit Curve

- Smoothly connecting two or more curves into one

- Removal of duplication
- Rounding of a bend
- Interpolation of a gap
- Trimming
Create PF Offset curve

- Creation of a curve by offsetting a profile curve on a surface which is internally created by parallel sweep of given profile curve.

Gouges are automatically detected and removed.
Sectional die

- Sectional die shape creation by sweeping a cross section line along profile curves.

- Removal of gouges
- Assurance of surface width
Pierce placement

- Placement of a pierce part based on its symbol
  Modification of pierce type and pierce parameter are possible.

Coincidence angle with placement base axis can automatically be calculated.
- List display of all pierce parts belonging to the current document.

Listed information can be exported to an external file. The information can be imported into the list in order to reflect it to pierce parts.

- Pierce information can be displayed at any time.
- Specified data will be echoed.
- Customization into any size is possible.
- Synchronization with the latest information.
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.

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

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

 Evaluatin based on the specified sub coordinate

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

Rotation angles are set by selecting from the list.
Rib

- A rib is created by sweeping a top surface to specified sweeping direction, where the surface is created by offsetting specified curves to both sides with the direction determined by the curve direction and sweeping direction.
- Dynavista common direction input is used.
- A curve not on a plane can be specified.

Thickness direction is perpendicular to both curve direction and sweep direction.
Inside Rib

- Function for creating inside rib in a die structure at stamping die design.
- A lattice ribs will collectively be created.

-Rib distance and thickness are easily modified by sketch function after creation.
Rib Through

- Creation of hole at ribs in order to reduce its weight.

- Edges of surface or any curves are available as a cutting line. (Non rectangle shape can be used.)
Triangle Rib

- Triangle ribs are easily created by specifying any base position on a bottom surface, wall surface and their common boundaries.
- Two or more triangle ribs can be created in one time (1 rib feature).

- Equal/different width is selectable

- Slope angle can be specified
Component placement

Easy to place a registered standard part

Place the part by the use of Standard parts placement command.

Characteristics
- Placement position can be controlled by a sketch (Placement check in the same manner as a drawing)
- Collective placement and holing are possible

Check placement position of the part by a sketch

Collective placement by specifying a sketch as a placement position

After placement

Select "Collective holing" at a context menu of the base where standard parts are placed.

Collective holing is completed.
Part attribute/ BOM

- Information such as part identifier, material, order information, heat treatment, etc. can be defined as part attributes.
- BOM is created using the part attributes.
- The attributes can be customized by exporting to the Excel file.
Performance improvement

- Create and edit part attribute – Improvement of import performance from Excel file -

<table>
<thead>
<tr>
<th>Result</th>
<th>V8.0</th>
<th>V9.5</th>
<th>Performance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Body</td>
<td>19 sec</td>
<td>8 sec</td>
<td>Improved by 58 %</td>
</tr>
<tr>
<td>Part</td>
<td>16 sec</td>
<td>5 sec</td>
<td>Improved by 69 %</td>
</tr>
</tbody>
</table>

- Part attribute list – Improvement of activation performance -

<table>
<thead>
<tr>
<th>Result</th>
<th>V8.0</th>
<th>V9.5</th>
<th>Performance</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 sec</td>
<td>8 sec</td>
<td>-</td>
<td>Improved by 60 %</td>
</tr>
</tbody>
</table>

Environment
- Windows XP Professional x64 Edition
- CPU Pentium4 3.2GHz
- Memory 4GB
A box is created based on a specified coordinate system so that it contains specified elements.
Bolt/Dowel pin placement

- A bolt or dowel pin is selected from a catalog and its instance is placed at specified position.

- Part selection by filtering
- Dragging by a handle as well as coordinate value input from a panel.
- Positioning by preview
- Reference dimension will be defined by components and 2 boundaries or lines.
Axis Coincident

- Optimum placement of parts by adjusting the parts to a base part coordinate also utilizing movement and reversal function.

- Point and face offset constraints are defined for each axis after part position is fixed.
- To edit a placed part, select an item in a context menu displayed at clicking the right button.
Simple Bolt/Dowel pin placement

- Place a circle of Tap hole symbol with machining attribute to the specified position after specifying hole size of a bolt, dowel pin or hook.

- The size of each part can be customized.
- Created circle has a machining attribute.
- Height dimensions are calculated and temporarily displayed on three dimensional data.
3D Balloon

-A balloon is created for a specified part placed in a 3D model.

- Whole assembly or a part can be specified.
- A balloon direction is specified on a definition plane
- Parts category can be specified.

-A drawing is created from “definition plane” direction in V5 drawing function.
Collective management of parameters such as dimensions or measurement values, or information such as checks and rules in one table.
Distance from a target object to another is measured and its result is shown on the object with gradation.

**Distance Map**

- 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.
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.)
Thickness of a solid is shown with a color map.

- 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.

Thin portion of a product or a die can be detected. A range can be specified by maximum and minimum values.
- A casting shape which fills a gap at standard parts placement is created.
Correct Gap

- A shape connecting a casting structure and a base plane is created.

- Height of a correcting portion can be adjusted by machining requirement.
Layer table

- Operability improvement is possible by visibility control using a layer table
Display control – Type ON/OFF

- Display on/off control depending on the selection in the command dialog.
Temporal display control without modification of displayed elements.
The dialog is always displayed even when other commands are activated.
The command operation is allowed during editing a part file, a product file or a process file.
3D Note / View change

- Creation of a text with a leader line for specified data.

Created text retains view information at the time of creation.
Machining attribute

- Machining attribute is copied to a hole shape by adding the attribute to a cut solid and by executing Collective hole generation command.

Consideration of association with 2.5D CAM. Items can be customized such as addition of items and non display of an item.

*Direct setting to a hole shape is also possible*
Machining attribute check

- Machining attributes of planes where parts touch each other are checked, and the result is reported as a list.

Part instances where no machining attribute is set will be displayed as a list.
Machining attribute color check

- Machining attribute colors of parts under the specified Product are checked, and the result is reported as a list.

Parts where no machining attribute color is set will be displayed as a list.
Machining attribute – Finish mark and Coarseness

Display improvement by finish type (symbol, coarseness) (V8.1 -)

- If “Finish Type” is “Finish Mark”, “Finish Mark”: the value and “Coarse”: “-” is displayed.

- If “Finish Type” is Ra/Ry/Rz, “Finish Mark”: “-”, and “Coarse”: “Finish Type + Coarse” is displayed.
Simplified machining attribute

Machining attribute can be created and edited without regard to its type and portion.

- Case not to edit the attribute
  1. Hole machining attribute, bottom
  2. Pocket machining attribute wall and evade surface
  3. Hole portion created by Hole portion command

"A massage will be shown when a mouse pointer is located on the target element."

Create and edit of an attribute

Case of no edition allowed
Machining attribute Set From Color

Automatically setting Machining attributes according to corresponding face colors (RGB) by pairing a face color and a machining attribute in a customizing file.

Support general machining attribute only (Simplified machining attribute is out of support).

Effective in the case of importing data from other systems.

| Attr. Type | Portion | Nr. of steps | Step Category | RGB | | | | |
|------------|---------|--------------|---------------|-----|-----|-----|-----|
| Hole Tap   | 1 Wall  | 1            |               | 255 | 0   | 0   |     |
| Hole Tap   | 1 Bottom| 1            |               | 255 | 0   | 120 |     |
| Hole Tap, Socket hole | 1 Wall | 2            |               | 255 | 120 | 0   |     |
| Hole Tap, Socket hole | 1 Bottom| 2            |               | 255 | 120 | 120 |     |
| Hole Tap, Socket hole | 1 Wall | 3            |               | 255 | 120 | 220 |     |
| Hole Tap, Socket hole | 1 Bottom| 3            |               | 255 | 120 | 255 |     |
| Pocket Bottom of die | 1 Wall | 1            |               | 255 | 255 | 0   |     |
| Pocket Bottom of die | 1 Bottom| 1            |               | 255 | 255 | 120 |     |
| Surface Product surface | 1       | 1-2          |               | 255 | 255 | 0   |     |

Customization file

Colored face

Face with machining attribute
A boundary curve is added to a face in a surface by specifying a boundary curve to be added (constant parameter curve/curve on a surface) and a surface. The divided faces where an edge is added inherit attributes attached to the original face.

In case that a dividing curve is short and face is not divided into two or more areas.

Edge is added after extending the curve tangentially.

Used for separating a portion as another component surface.
Boundary addition (2)

Input of two or more curves

Two or more curves can be specified in one operation.

Range where boundary curves and added curves are lapping.

No curves are added at lapping range of the boundary and added curved. (The boundary is prioritized.)
If an end point of the lapping range and the added curve are separated, the end point and a non-lapping range of the added curve will be connected by a line.
Design check analysis

To analyse whether any checks in a specified product are correctly executed. The result is shown in a list.

- If one or more NG or non-executed checks exist among target check features in the target part or product, they are displayed in the result dialog.
Synchronizing Attributes

Machining attributes of copy (“as Result with link”) are synchronized to the destination.

A body with hole machining attribute is copied and pasted to another part by “as result with link”. -> But no hole machining attribute is copied at this time.

+ No change will occur by executing this command if figures are created other than copy or copied other than “as result with link” (AsResultWithLink).
Check difference between two shapes and display it for ease of view.

- Minimum distance is calculated between a target shape and a compared shape. Portions where the distance is no less than a threshold value are displayed on the target shape in red.
Hole list

Hole list is exported to an Excel file by specifying a plate in a die model after hole creation.

- The list can be exported by specifying machining direction and machining base point (base position + movement).

- Portions where non-through holes penetrate the plate are taken into account.

- Hole position is a position from hole base point.
- Initial depth is a hole start position from base point.
- Hole depth is a hole position from hole list base point.
Related command
- POST (CONTOUR)
- Depth Measurement
- Die standard axis system
- Distance along surface
Press standard parts

Standard parts (parts catalogue from makers) are equipped.

<table>
<thead>
<tr>
<th>Category for catalogue</th>
<th>Registered parts</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small parts</td>
<td>Side pin set, locating key, Back up key, Stroke end block, Washer for stripper bolt and coil spring, Stripper bolt with external screw, Stripper bolt with bush, Stripper bolt with internal screw, Magnet cobalt, Magnet alnico, Press-in magnet, Circular distance plate, square distance plate, Blank holder stopper bolt, Angle stay, Die clamp washer, Cushion pin, Lifter cushion pin for transfer, Lifter guide bush, Plate lifter pin, Lifter plate stopper</td>
</tr>
<tr>
<td>Material guide lifter</td>
<td>Nest guide, Rough guide, Material guide roller set, material lifter, Sheet lifter, Material transfer lifter unit, Curl code hook, Transfer attachment, Skid bracket set, Location pin, Input detection switch, Magnet lifter pin, Spring plunger - light load, Spring plunger - heavy load, Spring plunger - extra-heavy load, Spring plunger - nylon nose, Spring plunger - slope, Ball plunger</td>
</tr>
<tr>
<td>Holder guide post set</td>
<td>Guide post, Guide holder, Oil-less guide bush, Plane guide bush, Ball guide bush, Plane guide post set</td>
</tr>
<tr>
<td>Die guide parts</td>
<td>Guide post, Oil-less guide bush, Stroke end block distance guide ring, Oil-less heal guide plate, Heal guide plate</td>
</tr>
<tr>
<td>Oil-less slide plate</td>
<td>Oil-less slide plate, Oil-less thin slide plate</td>
</tr>
<tr>
<td>Cam related parts</td>
<td>Cam stopper, Cam forced returning block, Cam forced returning follower, Cam stroke plate – 15 types, Cam stroke plate 30, Cam side block, Cam slide plate, Cam side plate, Cam returning unit, Cam upper plate</td>
</tr>
<tr>
<td>Spring</td>
<td>Urethane without hole - heavy load, Square urethane – heavy load, Urethane – heavy load, Urethane with counter sunk hole, Economy urethane, Urethane torus - bending, Urethane stripper, Low backlash urethane, Low backlash urethane with counter sunk hole</td>
</tr>
<tr>
<td>Lifter related parts</td>
<td>Urethane without hole - heavy load, Square urethane – heavy load, Urethane - heavy load, Urethane with counter sunk hole, Economy urethane, Urethane torus - bending, Urethane stripper, Low backlash urethane, Low backlash urethane with counter sunk hole</td>
</tr>
</tbody>
</table>

* Now major part are registered except for piercing tools from Misumi (pierce, button die and retainer), JIS pats (bolt, dwell pin)
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