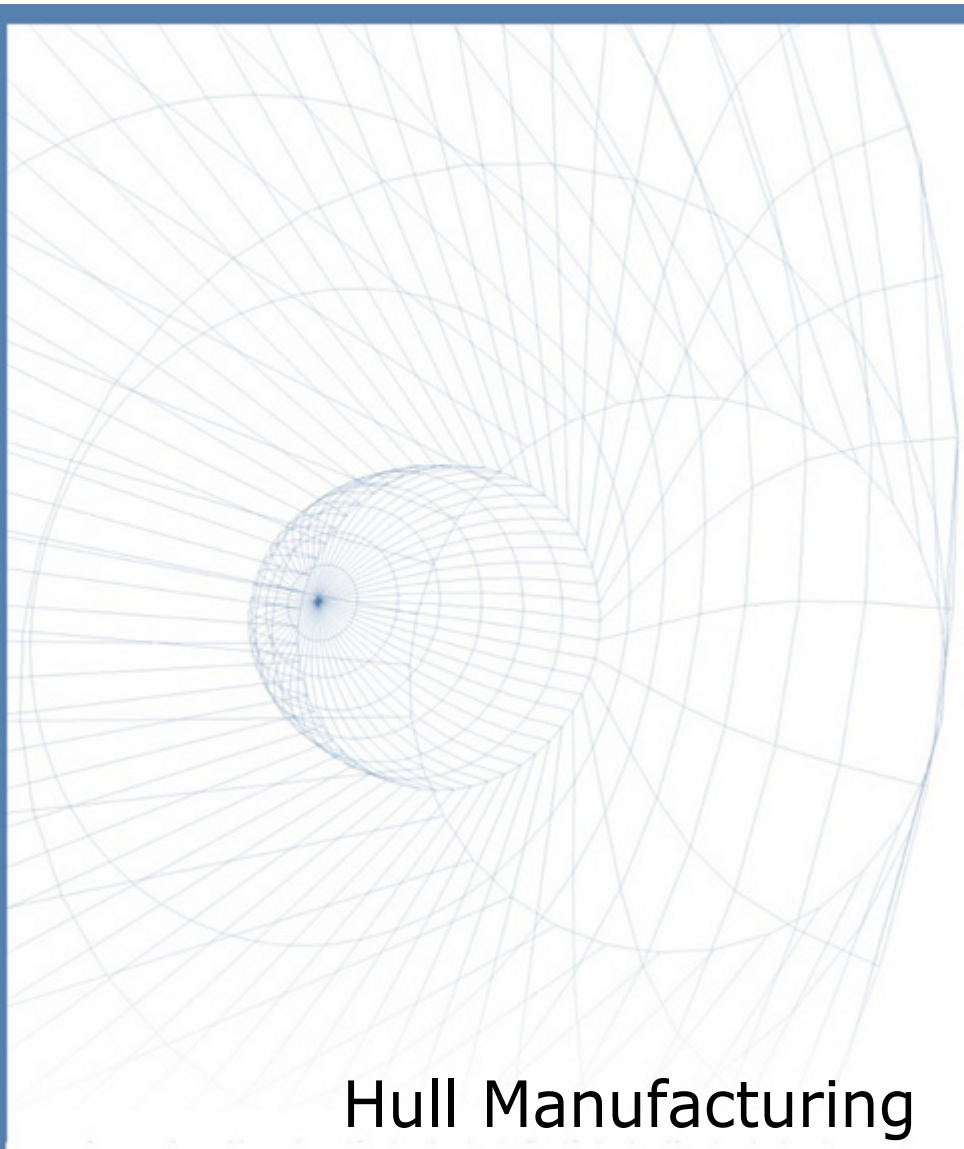

AVEVA MARINE

Hull Manufacturing

Nesting

User Guide

www.aveva.com

AVEVA Solutions Ltd

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Nest Job

New

Purpose:	To create a new nested plate.
Prerequisites:	A nesting session must be active. The nest name chosen for the new nest must not already exist on the data bank. Enter a different nest name if the chosen name exists.
Instruction:	Enter the name of the new nest.
Options:	Not available.
Result:	Select a parent plate from the stored plate catalogue or enter the plate definition data requested. A plate will be displayed on the screen, ready to receive nested parts.

Open

Purpose:	To open an existing nested plate.
Prerequisites:	A nesting session must be active.
Instruction:	Select **Open** from the **Job** menu. Enter the name of the nest to be opened, or choose a nest name from a filtered list.
Options:	Not available.
Result:	The chosen nest will be displayed on the screen. The geometry of the nested plate parts will be read from the plate parts data bank and the latest modifications of the parts will be displayed. If some parts have been changed after the nest was verified they will be shown in a separate colour.

Save

Purpose:	To store a current nest on the nested plates data bank.
Prerequisites:	A nest must be open.
Instruction:	If a nest with the same name exists on the nested plates data bank, confirm overwriting of the nest.
Options:	Not available.
Result:	The current nest will be saved on the nested plates data bank.

Rename

Purpose:	To rename a current nest to a new nest name. This may be used to duplicate existing nests, to allow modification without changing the original nest.
Prerequisites:	A nest must be open.
Instruction:	Enter a new name for the current nest. It is not possible to rename a current nest to a name which already exists on the nested plates data bank. Confirm when prompted if the old nest which is being renamed will become obsolete with the creation of the new nest.
Options:	Not available.
Result:	A current nest will be given a new name. Note that this nest is not stored on the nested plates data bank until **Save** is selected from the **Job** menu.

Data bank

List

Purpose:	To display a list of nested plate names, plate parts or standard plates on the data bank.
Prerequisites:	A nesting session must be active.
Instruction:	Enter a filter if required, to reduce the potential for a very large list.

Options:	Not available.
Result:	The list of nest fitting the specified criteria, if any, will be displayed on the screen.

Delete

Purpose:	To delete existing nests, plate parts or standard plates from the data bank.
Prerequisites:	A nesting session must be open.
Instruction:	Enter the name of the object to be deleted. Confirmation will be requested for the deletion. A wildcard may be used in this function to delete multiple nests. Confirmation will be requested individually for the deletion of each object in the group.
Options:	Not available.
Result:	The selected objects will be removed from the data bank.

Return to Drawing

Purpose:	To switch from viewing a nest to viewing a drawing.
Prerequisites:	A nest must be open.
Instruction:	Confirm exiting the current nest.
Options:	Not available.
Result:	The display will be returned to the current drawing in the work area. If there is no drawing current, the display will be returned to a blank screen.

Open Multiple

Purpose:	To open more than one existing nested plate.
Prerequisites:	A nesting session must be active.
Instruction:	Select **Open Multiple** from the **Nest Job** menu. Enter a search string for nested plates, press **OK**, a multiple selection dialog appears and there a selection of nested plates can be made.

Options:	Not available.
Result:	The selected nested plates will all appear for editing of layout. Addition of parts, transformation and move part within or between nested plates are possible to perform in multiple nest mode. **Note:** That a verify and nested sketch is not available.

Show plate/Show all plates

This function is only available when a multiple nesting session is open. The function Show plate can the be used to only display the nested plate which is indicated by the user. The user has then the possibility to e.g. create starts and verify the tool path which is not possible in a multiple nesting session. The function Show all parts will display all the nests in the multiple session.

Nesting Parts

Nest

Purpose:	To define parts to be included in a nested plate.
Prerequisites:	A nest must be open.
Instruction:	Select the method of part definition from the displayed sub-menu. Choose to define parts by entering the name of the part, to include parts from an existing data file or nest from a *parts menu* displayed on the screen. To include parts from an existing data file, either scroll through the unused parts in the data file or enter individual position numbers for parts in the file. If a data file has not been used to define parts in the nest previously, the name of the required data file must be entered before commencing. Select a part using either method described above, or nest parts from a *parts menu* by indicating the required part and locating it on the plate. Place the part on the nest and perform transformations to locate the part as required. Note that if a nested plate is linked to a specific data file, all parts nested on the plate must be contained in the data file.
Options:	When nesting from a *parts menu*, it is possible to select **All** to perform a feature referred to as 'Quicknest'. This will automatically locate parts in descending order of size on the plate, until no further parts can be added.
Result:	The chosen parts will be added to the nest. The changes made using this function will not affect the stored nest until **Save** is selected from the **Job** menu.

Parts Menu

Purpose: To display a menu of parts in the work area for subsequent nesting.

Prerequisites: A nest must be open.

Instruction: Choose parts by using the filter options in the dialogue.

Options: Not available.

Result: A menu of parts available for nesting will be displayed in a separate window, at reduced scale for viewing.

Transform

Purpose: To relocate a part on a nested plate.

Prerequisites: A nest must be open.

Instruction: Select a part or parts to transform on the nested plate.

Options: Not available.

Result: The selected parts will be available for transformation using the Transformation toolbar. The direction menu for **Bump moves** is displayed at the top of the work area at all times while this function is active.

Note: that any combination of the transformations in the Transformation toolbar can be performed before selecting **Operation Complete** to complete the transformation. This includes rotation and mirror sub-functions. **Exit** function will terminate the function without any transformation.

Verify

Purpose: To perform a verification of a nested plate as a visual check of the tool path. Also to create an output file for later post-processing and the framework of a shop drawing.

Prerequisites: A nest must be open. The nested plate must contain at least one marking or burning start.

Instruction:	Answer the prompts as necessary to provide the required outputs. Confirm each start in the tool sequence or select **All** at any point in the sequence to execute a complete verification.
Options:	Not available.
Result:	The nest will be verified and the outputs requested will be created.

Bump Curve

On

Purpose:	To switch the display of the bump curve on.
Prerequisites:	A nest must be open. The nest must contain nested parts and/or a parts menu.
Instruction:	Select **Bump Curve** from the **Nesting Parts** menu. Select **On** from the displayed menu.
Options:	Not available.
Result:	The bump curve will be displayed.

Off

Purpose:	To switch the display of the bump curve off.
Prerequisites:	A nest must be open. The nest must contain nested parts and/or a parts menu.
Instruction:	Select **Bump Curve** from the **Nesting Parts** menu. Select **Off** from the displayed menu.
Options:	Not available.
Result:	The bump curve will be turned off.

Clustering

On

Purpose:	To switch the clustering function on. Clustering allows the efficient nesting of similar parts during quick nesting, by rotating a part through 180° prior to placing it on the plate. Triangular parts such as brackets are particularly suited to clustering.
Prerequisites:	A nest must be open. A parts menu must exist in the work area.
Instruction:	Select **Clustering** from the menu of **Nesting Parts** menu. Select **On** from the displayed menu.
Options:	Not available.
Result:	Clustering will be switched on.

Off

Purpose:	To switch the clustering function off. Clustering allows the efficient nesting of similar parts during quick nesting, by rotating a part through 180° prior to placing it on the plate. Triangular parts such as brackets are particularly suited to clustering.
Prerequisites:	A nest must be open. A parts menu for automatic nesting must exist in the work area.
Instruction:	Select **Clustering** from the menu of **Nesting Parts** menu. Select **Off** from the displayed menu.
Options:	Not available.
Result:	Clustering will be switched off.

Exchange

Raw Plate

Purpose:	To exchange an existing parent plate with a new plate, selected from the list of available plates or defined interactively.
Prerequisites:	A nest must be open.

Instruction:	Select a method of plate definition from the displayed menu. If exchanging **By Name**, choose a plate thickness and grade from the displayed list, then choose the stock number of the plate required. If exchanging **By Dimensions**, enter the plate definition data as prompted.
Options:	Not available.
Result:	The parent plate for the nest will be exchanged for the new plate specified. Note that there may be overlaps after the plate is exchanged. An on-screen warning will inform of any overlaps.

Part

Purpose:	To exchange an existing part on a nested plate with an updated version of the same part or a different part.
Prerequisites:	A nest must be open.
Instruction:	Indicate and confirm the part to be exchanged. Select **Yes** if the new part is to be the same name as the existing part. Select **No** if the new part is to be a different part. No further action is required if selecting **Yes**. If selecting **No**, enter the name of the replacement part.
Options:	Not available.
Result:	The selected part will be replaced by either an updated version of the same part or the alternative part entered.

Autonest

Purpose:	To automatic create the layout of the given parts on the rawplate with minimum scrap.
Prerequisites:	A nest must be open. a parts menu must exist in the work area. An external pattern generator must be connected to the Nesting system using the Automatic Nesting Plug-In.
Instruction:	None.
Options:	Not available.
Result:	The result of the automatic layout of the parts will be displayed on the nest. The creation of tool path burning sketch etc. can be continued.

Nesting Tools

Bridge

Purpose:	To add bridges to a nest, to connect nested parts or bridge unwanted gaps.
Prerequisites:	A nest must be open.
Instruction:	Select a bridge type from the displayed menu. Indicate the location information requested by the system. Refer to further 'Help' information if detailed explanation of the available bridge types is required.
Options:	Select **Options** before selecting a bridge type to interactively define the width of the bridge, if the required bridge width if different from the stored default. Selecting **Options** when the system prompts for a position, will get the user a possibility to toggle between indicating one or two positions when defining the bridge.
Result:	A bridge will be added to the nested part as indicated.

Start

Purpose:	Add or modify a start for marking or burning of a nested part. starts for labelling text and symbols can also be added.
Prerequisites:	A nest must be open. Parts must be nested on the plate before attempting to add starts, unless **Cut Line** is chosen.
Instruction:	Select the type of start to add to the part, or select the method of modification for an existing start.
Options:	Not available.
Result:	A sub-menu will be displayed.

Burn

Purpose:	To add burning starts to a nested plate.
Prerequisites:	A nest must be open. One or more parts must be nested on the plate.

Instruction: Choose the method of adding starts. Three methods are available. Choose [1] to add individual starts, [2] to add starts in all holes on the nest, or [3]to cut free geometry.

For [1], Indicate the location of the start on a part. Note that the default hook type or the hook type chosen should be suitable for the location on the part, as dictated by specific customer requirements.

For [2], the starts are located automatically. Note that the location of the starts chosen by the system may not be suitable for specific customer requirements.

For [3], Indicate the free geometry to be cut.

Options: Options are only available when adding single burn starts, by selecting **Reject**. The options available allow selection of alternative hook types and location method for starts.

Note: that when an alternative hook type or location method is chosen, it is active for the addition of one start only.

Result: A burn start will be added to the nested part as specified. Note that, typically, the customer will require marking starts to be added prior to adding burning starts.

Mark

Purpose: To add marking starts to a nested part.

Prerequisites: A nest must be open. One or more parts nested on the plate must contain marking lines.

Instruction: Select **Mark** from the **Start** sub-menu. Select from [1] Single starts [2] or all marking starts. Function [3] does not add marking starts to the nested plate. This function will simply indicate at which end of a line the start should be placed to have the material thickness on the left of the line. When the nested plate is zoomed or repainted, these indications will disappear and the function must be selected again to re-display this information.

Options: Not available.

Result: Marking starts are added to the nested parts as specified.

Label

Purpose:	To add label starts to Labelled Text and Labelled Symbols.
Prerequisites:	A nest must be open. If there are no Labelled Text items or Labelled Symbols on the nest, this function will have no effect.
Instruction:	None.
Options:	Not available.
Result:	Label starts will be added to all Labelled Text items and Labelled Symbols on the nest. Note that labelled items should be marked on the plate before burning, and that the associated default 'WORKING_SEQUENCE' may need to be modified to suit.

Automatic

Purpose:	To add all marking and burning starts automatically the nested parts.
Prerequisites:	A nest must be open.
Instruction:	None.
Options:	Not available.
Result:	Any existing starts will be removed from the nest. New starts will be added to the nested parts. All marking starts will be added before burning starts in the nest. Note that if there are marking lines on the far side of a part, this function will not include them. The starts can be added in two ways, either all marking first and the burning part wise (holes first) or all starts parent wise (marking, holes, outers). It is also possible to control if marking lines on the far side should be included. These possibilities are controlled with default parameters.

Move

Purpose:	To relocate burning or marking starts on a nested plate.
Prerequisites:	Marking or burning starts must exist on the nest.
Instruction:	Select the start to be moved. Indicate a new location for the start.

Options: Not available.

Result: The selected start will be repositioned at the location indicated. Note that in some situations the current hook may not be suitable for the location chosen. The changes made will not affect the stored nest until **Save** is selected from the **Job** menu.

Sequence

Purpose: To rearrange the order of marking and burning starts on a nested plate.

Prerequisites: A nest must be open. Marking or burning starts must exist on the nest.

Instruction: Select **Sequence** from the **Start** sub-menu. Choose a function from the displayed list. [1] will display the current order of starts. [2] will allow the re-ordering of all starts on the nest. Note that this function will not be effective until all starts in the sequence have been indicated. If this function is exited by using OC or EF, none of the changes will be made. [3] allows the re-ordering of individual starts in the sequence.

Options: **Options** may be selected to redefine the text height for displaying the start numbers. When redefining the entire sequence, select **Reject** to step backwards through the sequence to correct a start indicated in error.

Result: Starts are renumbered as defined.

Labelling Text

Purpose: To add markable text to a plate part in a nest.

Prerequisites: A nest must be open. The nested plate must be displayed. It is not possible to use existing text on the shop drawing as labelled text.

Instruction: None.

Options: Not available.

Result: A sub-menu offering different methods of text definition is displayed.

Key in

Purpose:	To add markable user-defined text to a plate part.
Prerequisites:	A nest must be open. The nested plate must be displayed.
Instruction:	Enter the text required in the input window.
Options:	**Options** may be selected after the text has been entered, to change the characteristics of the text prior to positioning it on the nested plate.
Result:	Text will be added to the part. Items of labelled text can then have starts added to mark them on the part.

Plate

Purpose:	To add markable text to a nest, describing predefined features of a nested plate.
Prerequisites:	A nest must be open. The nested plate must be displayed. A file containing the types of information to be added must already exist. Please see the Hull Nesting Users Guide for the details on this file.
Instruction:	Due to the volume of information required to describe this function in suitable detail, refer to Nesting documentation for instructions.
Options:	Not available.
Result:	The predefined text will be added to the nested plate.

Part

Purpose:	To add markable text to a nested part, describing predefined features of the part.
Prerequisites:	A nest must be open. The nested plate must be displayed. A file containing the types of information to be added must already exist. Please see the Hull Nesting Users Guide for the details on this file.
Instruction:	None.
Options:	Not available.
Result:	The predefined text will be added to the plate.

Pos. No.

Purpose:	To add marking for the position number of a part to a plate part in a nest.
Prerequisites:	A nest must be open. The plate must be displayed. A position number must be assigned to the part stored in the data bank.
Instruction:	Indicate the required part on the nest. Place the text as desired.
Options:	Not available.
Result:	Text will be added to the part. Items of labelled text can then have starts added to mark them on the part.

Prod Info.

Purpose:	To add marking for any associated production information to a plate part in a nest.
Prerequisites:	A nest must be open. The plate must be displayed. It is not possible to use existing text on the shop drawing as labelled text. The production information type chosen for the plate part must exist on the part in the data bank.
Instruction:	From the next menu select, the type of production information to be added. Indicate the part required. Place the text as desired.
Options:	**Reject** may be selected to change the characteristics of the text prior to placing the text on the part. **Options** may be selected to choose the way in which the text is output, either on the shop drawing only, in the NC information only, or in both.
Result:	Text will be added to the part. Items of labelled text can then have starts defined to mark them on the part.

Copy

Purpose:	To make a duplicate of a Labelled Text item on a nested plate.
Prerequisites:	A nest must be open. The nested plate must be displayed.
Instruction:	Select the labelled text to copy. Indicate a location for the copy of the text.

Options:	**Options** may be selected to change the characteristics to the text prior to placing the copied text on the nested plate.
Result:	A copy of the selected labelled text item will be placed on the nested plate.

Move

Purpose:	To relocate labelled text on a nested plate.
Prerequisites:	A nest must be open. The plate must be displayed on screen. The nest must contain labelled text or this function will have no effect. Labelled text cannot be moved on the shop drawing.
Instruction:	Select the labelled text to move. Indicate a new location for the labelled text.
Options:	**Options** may be selected after the text has been indicated, to change the characteristics of the text.
Result:	The selected labelled text items will be relocated on the nested plate.

Labelling Symbol

Purpose:	To add markable symbols or position numbers to a plate part in a nest.
Prerequisites:	A nest must be open. The nested plate must be displayed.
Instruction:	None.
Options:	Not available.
Result:	A sub-menu will be displayed.

Symbol

Purpose:	To add markable symbols to a plate part in a nest.
Prerequisites:	A nest must be open. The plate must be displayed.
Instruction:	Choose **Add** from the displayed menu. Indicate the symbol type from the selection displayed on-screen. Place the symbol as desired.

Options:	**Options** may be selected to change the characteristics of the symbol prior to placing it on the nested plate. **Reject** may be selected to choose an alternative Font ID for symbols.
Result:	A symbol will be added to the part in the nest. Labelled symbols can then starts defined to mark them on the plate part.

Pos. No.

Purpose:	To add markable position numbers to a plate part on a nest.
Prerequisites:	A nest must be open. The plate must be displayed. A position number must be assigned to the part stored in the data bank.
Instruction:	Choose **Pos. No** from the displayed menu. Indicate the required part.
Options:	Not available.
Result:	A position number is added to the part. Note that there is no opportunity to modify the size, rotation etc. of the position number, and that the position number will be placed on the part exactly where the part was indicated.

Copy

Purpose:	To make a duplicate of a Labelled Symbol on a nested plate.
Prerequisites:	A nest must be open. The nested plate must be displayed.
Instruction:	Select **Label Symbol** from the **Copy** sub-menu. Select the labelled symbol to copy. Indicate a location for the copy of the symbol.
Options:	**Options** may be selected to change the characteristics of the symbol prior to placing the copied symbol on the nested plate.
Result:	A copy of the selected labelled symbol will be placed on the nested plate.

Move

Purpose:	To relocate labelled symbols on a nested plate.
Prerequisites:	A nest must be open. The plate must be displayed on screen. The nest must contain labelled symbols or this function will have no effect. Labelled symbols cannot be moved on the shop drawing.
Instruction:	Select the labelled symbol to move. Indicate a new location for the labelled symbol.
Options:	**Options** may be selected after the required symbol has been indicated, to change the characteristics of the symbol.
Result:	The selected labelled symbols will be relocated on the nested plate.

Auxiliary Function

Purpose:	To add a predefined Auxiliary Function to a nest.
Prerequisites:	A nest must be open.
Instruction:	Select a location type for the Auxiliary Function. Indicate the location for the Auxiliary Function. Key in the Auxiliary Function numbers requested at that location.
Options:	Not available.
Result:	Auxiliary Functions defined will be displayed on the nest at the location indicated. When the nest is verified, the verification will be delayed at the location of the Auxiliary Functions until confirmation is given.

Corner Loop

Purpose:	To add a loop in the corner of a nested part, to facilitate rotation of burning heads during bevelling.
Prerequisites:	A nest must be open.
Instruction:	Indicate the corner of a part to add the loop.

Options:	Not available.
Result:	A corner loop will be added to the part in the location indicated. Optionally it is possible to define auxiliary functions in three predefined points in the corner loop.

Hooks

Purpose:	To create or modify hooks to be used as burns starts.
Prerequisites:	A nesting session must be active.
Instruction:	None.
Options:	Not available.
Result:	A sub-menu will be displayed.

Open

Purpose:	To modify an existing burning hook.
Prerequisites:	A nesting session must be active.
Instruction:	Select a hook to modify from the displayed list of stored hooks. Modify the geometry of the hook as required. Save the modified hook.
Options:	Not available.
Result:	The modified hook will be updated on the data bank.

Save

Purpose:	To save a newly created or modified burning hook.
Prerequisites:	A nesting session must be active. A drawing with the same name as the required hook must be active in the work area.
Instruction:	Indicate the location of the start of the hook. The end of the hook should be in (0,0) with the positive u-axis as the continued tool path and the scrap side above the u-axis. Confirm overwriting of the existing hook if prompted.
Options:	Not available.
Result:	The newly created or modified hook will be stored on the data bank.

Parts

Plate

Purpose:	To create plate parts from a drawing.
Prerequisites:	A nesting session must be open.
Instruction:	Indicate the outer contour of the part. Holes are found automatically. Marking lines are defined by the user.
Options:	Not available.
Result:	The plate parts are stored on the plate parts data bank and can be used in the nesting.

Profile - Create

Purpose:	To create a new profile part.
Prerequisites:	A nesting session has to be open.
Instruction:	None.
Options:	Not available.
Result:	A new profile part is created from an existing rawprofile.

Profile - Edit

Purpose:	To edit an existing profile part.
Prerequisites:	A nesting session must be open.
Instruction:	None.
Options:	Not available.
Result:	

Profile - Delete

Purpose:	To delete an existing profile part.
Prerequisites:	A nesting session must be open.
Instruction:	None.

Options:	Not available.
Result:	

Check Post Processor

Purpose:	To create the output from the generic Post Processor.
Prerequisites:	A nest must be open. The tool path must be verified and the GPP must have been run twice, once to create the input to the burning machine, and once to create a generic file from the burning machine input. This can be done automatically in the verification.
Instruction:	None.
Options:	Pressing **Options** in the displayed window gives the user a possibility to indicate from where the check shall start.
Result:	The input to the burning machine is shown in a window. Selecting **All** will display the corresponding geometry on the screen. Then it is possible to step through the file and check that all auxiliary functions are present and in the correct position. The geometry is displayed in different colours or line type for burning and marking, any deviations from the original geometry can easily be detected.

Delete

Purpose:	To remove existing items or functions from a nested plate.
Prerequisites:	A nest must be open.
Instruction:	None.
Options:	Not available.
Result:	A sub-menu will be displayed.

Bridge

Purpose:	To remove an existing bridge from a nested plate.
Prerequisites:	A nest must be open. If there are no bridges in the nest, this function will have no effect.

Instruction:	Indicate the bridge to be removed. Confirmation will be requested if the bridge has a start attached. Choose **All** to delete all bridges, in which case, no confirmation is requested.
Options:	Not available.
Result:	The indicated bridge will be deleted.

Start

Purpose:	To remove an existing start from a nested plate.
Prerequisites:	A nest must be open. If there are no starts in the nest, this function will have no effect.
Instruction:	Indicate and confirm the start to be removed. Choose **All** to delete all starts, in which case, no confirmation is requested.
Options:	Not available.
Result:	The indicated start will be deleted.

Part

Purpose:	To remove an existing part from the nested plate.
Prerequisites:	A nest must be open. Previously nested parts must exist on the plate.
Instruction:	Indicate the part to be removed from the nest. The part will be highlighted and is deleted when the next part is indicated. The last part will be deleted when **Operation Complete** is given.
Options:	Not available.
Result:	The selected part is removed from the nested plate. The deletion will not affect the stored nest, until **Save** is selected.

Auxiliary Function

Purpose:	To remove auxiliary functions from a nested plate.
Prerequisites:	A nested plate must be open. If there are no auxiliary functions used in the nest, this function will have no effect.
Instruction:	Indicate the auxiliary function to be removed. Choose **All** to delete all auxiliary functions, in which case, no confirmation is requested.
Options:	Not available.
Result:	The indicated auxiliary functions are deleted.

Sketch

Purpose:	To remove existing items from the shop drawing.
Prerequisites:	A nest must be open. A shop drawing must exist.
Instruction:	Select the type of item to be deleted from the next menu. Select the item to be deleted from the displayed sub-menu. Exercise caution during this function to avoid deleting an item accidentally. If, for example, the burn path, which would normally be required in the shop drawing, is deleted, it can be reinstated by reverifying the nested plate. Items such as position numbers must be redefined interactively if they are deleted in error.
Options:	Not available.
Result:	The selected items will be removed from the shop drawing.

Labelled Text

Purpose:	To remove existing 'Label Text' from a nested plate.
Prerequisites:	A nest must be open. If there are no labelled text items in the nest, this function will have no effect.
Instruction:	Indicate the labelled text to remove. No confirmation is requested. Choose **All** to delete all labelled text items.
Options:	Not available.
Result:	The indicated Label Text is removed from the nest.

Labelled Symbol

Purpose:	To remove existing 'Label Symbols' from a nested plate.
Prerequisites:	A nested plate must be open. If there are no labelled symbol items in the nest, this function will have no effect.
Instruction:	Indicate the labelled symbol to remove. No confirmation is requested. Choose **All** to delete all labelled symbols.
Options:	Not available.
Result:	The indicated labelled symbols are deleted. Note that labelled symbols added as position numbers exist as two separate entities, one labelled symbol and one labelled text item. The text element of the position number must be deleted as labelled text, even if selecting **All** symbol deletion.

Shop Drawing I/O

Display Sketch

Purpose:	To switch from viewing the nested plate to viewing the shop drawing.
Prerequisites:	A nest must be open.
Instruction:	None.
Options:	Not available.
Result:	The shop drawing will be displayed on the screen. If the nested plate has not been verified, a blank screen will be displayed.

Display Plate

Purpose:	To switch from viewing the shop drawing to viewing the nested plate.
Prerequisites:	A nest must be open.
Instruction:	None.
Options:	Not available.
Result:	The nested plate will be displayed on the screen.

Save

Purpose: To create a new shop drawing form in the nesting standards data bank.

Prerequisites: A drawing must be open. The drawing name must be the same as the name to be used for the form.

Instruction: After creating the geometry for the form and defining any $-text items to be used in the form, select **Save** from the **Form** sub-menu. Define the available working area of the form, by indicating two points. The points can be redefined until Operation Complete is selected.

Note: Refer to further 'Help' information for a full listing of $-text variables.

Options: Not available.

Result: The new form will be stored on the nesting standards data bank and will disappear from the work area. The form is now available for use in nesting. Care should be taken when storing 'new' forms to ensure that an existing form is not overwritten with a completely different form.

Add

Purpose: To add a form to the current burning sketch.

Prerequisites: A burning sketch must exist.

Instruction: Select the name of the form to be added from the displayed list.

Options: Not available.

Result: The form will be added to the burning sketch and the burning sketch will be displayed. Optionally a number of production information text and symbols can be created automatically.

Open

Purpose: To update an existing shop drawing form in the nesting standards data bank.

Prerequisites: The form to be modified must exist on the data bank.

Instruction:	Select the name of the form to be modified from the displayed list. After modifying the form as required, select **Save** from the **Form** sub-menu.
Options:	Not available.
Result:	The selected form is updated in the nesting standards data bank and will disappear from the work area.

Overlap Check

Purpose:	To check the nested plate and parts to determine if the gaps between the plate edge and the nested parts, or the gaps between adjacent parts are less than specified in the default file.
Prerequisites:	A nest must be open.
Instruction:	None.
Options:	Not available.
Result:	Any overlap is identified on the screen. Confirm acceptance of the overlaps or rework the nest to remove overlaps. Note that only one overlap is identified at a time. Rectification of one overlap may reveal further overlaps.

First Page/Next Page/Previous Page/Last Page

Purpose:	To view additional 'pages' of a shop drawing.
Prerequisites:	A nest must be open. The shop drawing must contain additional 'pages'. This will happen when there is a part table defined in the nesting form and the number of nested parts exceed the number of tables. If no additional 'pages' exist, the single page of the shop drawing will be displayed.
Instruction:	None.
Options:	Not available.
Result:	The selected 'page' of the sketch will be displayed on the screen. The geometry will be the same on all additional 'pages'. Only the information in the part tables will change.

Shop Drawing Information

Production Information

Purpose: To request production information relating to a specific part on a nested plate.

Prerequisites: A nest must be open. The nested plate must be displayed.

Instruction: Select **Prod. Info** from the menu of **Tools** sub-functions. Select the type of production information to be displayed.

```
[1] Bevel
[2] Excess
[3] Side Info
[4] Part ID
[5] Part Name
[6] Panel Name
[7] Pos no
[8] Weight
[9] Customer data
```

Indicate the part on the nested plate for which the information is to be displayed.

Options: Not available.

Result: The requested production information will be displayed in the dialog window at the bottom left of the work area. No text will be added to the nested plate or the shop drawing.

Text

Key in

Purpose: To add user-defined text to the shop drawing of a nested plate.

Prerequisites: A nest must be open. The shop drawing must be displayed.

Instruction: Select **Text** from the menu of **Shop Drawing** sub-functions. Enter text to be added to the shop drawing. Indicate the location for the text.

Options:	**Options** may be selected to modify text height, rotation etc. before placing the text on the shop drawing.
Result:	The defined text will be added to the shop drawing. The changes made using this function will not affect the stored nest until **Save** is selected from the **Job** menu.

Copy/Move

Purpose:	To relocate or copy text on the shop drawing of a nested plate.
Prerequisites:	A nest must be open. The shop drawing must be displayed.
Instruction:	Select the text to move or copy. Indicate a new location for the text.
Options:	**Options** may be selected after the text has been indicated, to modify the text height, rotation etc. prior to indicating the new location for the text.
Result:	The selected shop drawing text will be relocated on the shop drawing.

Starts / Plate name

Purpose:	To display the sequence of starts or the name of the raw plate on the shop drawing of a nested plate.
Prerequisites:	A nest must be open. The shop drawing must be displayed.
Instruction:	Select **Starts/Plate Name** from the menu of **Shop Drawing** sub-functions. Select **Starts** or **Plate Name** from the displayed menu. For the **Plate Name** function, locate the text on the drawing. Text for **Starts** is located automatically.
Options:	**Options** are available for **Plate Name** only, to modify the properties of the text prior to placing on the shop drawing.
Result:	The selected text items will be placed on the shop drawing.

Symbol

Put

Purpose:	To add symbols to a shop drawing.
Prerequisites:	A nest must be open. The shop drawing must be displayed.
Instruction:	Select **Symbol** from the menu of **Shop Drawing Information** sub-functions. Choose a symbol from the selection displayed at the top of the work area. Indicate the location for the symbol.
Options:	**Options** may be selected to modify symbol height, rotation etc. before placing the symbol on the shop drawing.
Result:	The selected symbol will be added to the shop drawing.

Move

Purpose:	To relocate symbols on the shop drawing of a nested plate.
Prerequisites:	A nest must be open. The shop drawing must be displayed.
Instruction:	Select **Shop Dwg Symbol** from the **Move** sub-menu. Select the symbol to move. Indicate a new location for the symbol.
Options:	**Options** may be selected after the symbol has been indicated, to modify the symbol height, rotation etc. prior to indicating the new location for the symbol.
Result:	The selected shop drawing symbol will be relocated on the shop drawing.

Pos. No.

Purpose:	To add position numbers to plate parts or marking lines on the shop drawing of a nested plate.
Prerequisites:	A nest must be open. The shop drawing must be displayed.

Instruction:	Indicate the part or marking line to have the position number. Locate the position number on the shop drawing.
Options:	Not available.
Result:	A position number will be added to the shop drawing for the part or marking line indicated.

Rest Plate

Create

Purpose:	To create a new raw plate on the standards data bank from remnant material on an existing nested plate.
Prerequisites:	A nest must be open. The raw plate currently used must be from the standards data bank. It is not possible to create a restplate on a raw plate defined by dimensions.
Instruction:	Select **Restplate** from the **Tools** menu.
Options:	Not available.
Result:	For special steel quality the rest is stored on the nestings standards data bank and some information is added to the burning sketch. For normal steel quality only the texts are added to the burning sketch.

Delete

Purpose:	To remove a restplate from the standards data bank.
Prerequisites:	A nesting session must be active.
Instruction:	Enter the name of the restplate or select a restplate to delete from a filtered list of existing restplates.
Options:	Not available.
Result:	The restplate will be removed from the standards data bank.

Status

Production Information

Purpose: To request production information relating to a specific part on a nested plate.

Prerequisites: A nest must be open. The nested plate must be displayed.

Instruction: Select **Prod Info** from the menu of **Tools** sub-functions. Select the type of production information to be displayed.

```
[1] Bevel
[2] Excess
[3] Side Info
[4] Part ID
[5] Part Name
[6] Panel Name
[7] Pos no
[8] Weight
[9] Customer data
[10] Block name
Ship number
Dates
```

Indicate the part on the nested plate for which the information is to be displayed.

Options: Not available.

Result: The requested production information will be displayed in the dialog window at the bottom left of the work area. No text will be added to the nested plate or the shop drawing.

Defaults

Purpose: Supervisor for the default system. The user can list, change and store the default file.

Prerequisites:

Instruction: None.

Options: Not available.

Result The current default settings will be listed. The default parameters belonging to the inferior default file can be changed.

Calculate Distance

Purpose:	To determine the location of a specific point on the nested plate, measured as X & Y coordinates from the origin of the nest (the bottom left corner of the raw plate).
Prerequisites:	A nest must be open. The nested plate must be displayed.
Instruction:	Indicate the point to be measured. It is possible to select a specific node, arc centre etc. by selecting from the menu of point definition methods available by clicking on the right mouse button.
Options:	Not available.
Result	The distance to the point from the origin of the nest will be displayed as X & Y coordinates in the dialog window at the bottom left of the work area. No information will be added to the nested plate or the shop drawing.

Utilities

Dump Plate

Purpose:	To store the existing nest, including all part and plate geometry. All plate part geometry is normally deleted before a nest is stored.
Prerequisites:	A nest must be open.
Instruction:	Confirm overwriting of the existing nest.
Options:	Not available.
Result:	The current nest will be stored, with all part and plate geometry.

Show Structure

Purpose:	To drawing with the data structure used in the bump function.
Prerequisites:	Nest must be open.
Instruction:	None.

Options: Not available.

Result: A drawing will be created.

Check Object

Purpose: To list the contents of attributes in the nested plate CAT object.

Prerequisites: Nest must be opened

Instruction: Choose a level from the displayed list.

```
[1] object attributes
[2] contour attributes
[3] segment attributes
```

Options: Not available.

Result: The selected attribute will be listed.

Way to Transform

Move

Bump

Purpose: To move a part adjacent to an edge of the parent plate or another part, holding the part at a distance specified in the nesting default file.

Prerequisites: A nest must be open. A part must be selected to be moved.

Instruction: Choose a direction for the move from the menu displayed at the top of the work area window. Choose additional transformations or select OC to complete the move.

Options: Not available.

Result: The part will be moved as indicated.

Parallel

Purpose:	To move and rotate a part to position it parallel to the selected edge of the parent plate or another part on the nested plate. The distance between the adjacent parts after the move will be as specified in the nesting default file.
Prerequisites:	A nest must be open. A part must be selected to be moved.
Instruction:	Indicate an edge on the selected part to be made parallel to another edge. Indicate the edge of the parent plate or another part. Choose additional transformations or select OC to complete the move.
Options:	Not available.
Result:	The part will be moved as indicated.

DX DY

Purpose:	To move a part by a distance relative to its current location.
Prerequisites:	A nest must be open. A part must be selected to be moved.
Instruction:	Enter the distance required for the move in the **X** direction (horizontal). Enter the distance required for the move in the **Y** direction (vertical). Choose additional transformations or select OC to complete the move.
Options:	Not available.
Result:	The part will be moved as indicated.

Two Positions

Purpose:	To move a part by definition of two specific points.
Prerequisites:	A nest must be open. A part must be selected to be moved.
Instruction:	Select a point definition method using the menu activated by the right mouse button. Indicate two positions with the cursor. Choose additional transformations or select OC to complete the move.

Options:	Not available.
Result:	The part will be moved as indicated.

Dynamically

Purpose:	To dynamically move a part on a nested plate.
Prerequisites:	A nest must be open. A part must be selected to be moved.
Instruction:	By default, dynamic moves are normally active. If this is not the case, select **Dynamically** from the menu of **Move** sub-functions. Indicate the already selected part, and drop it at a new location. Choose additional transformations or select OC to complete the move.
Options:	**Options** can be used to toggle between **Move Dynamically** and **Rotate Dynamically**.
Result:	The part will be moved as indicated. The changes made using this function will not affect the stored nest until **Save** is selected from the **Job** menu.

Rotate

Angle

Purpose:	To rotate a part through a specific angle.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Enter the angle in degrees through which the selected part is to be rotated. Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated, around the current rotation centre.

-90

Purpose:	To rotate a part by a predetermined angle, in this case -90°, around the current centre of rotation.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated.

45

Purpose:	To rotate a part by a predetermined angle, in this case 45°, around the current centre of rotation.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated.

90

Purpose:	To rotate a part by a predetermined angle, in this case 90°, around the current centre of rotation.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated.

180

Purpose:	To rotate a part by a predetermined angle, in this case 180°, around the current centre of rotation.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated.

Two Positions

Purpose:	To rotate a part by indicating two cursor positions.
Prerequisites:	A nest must be open. A part must be selected to rotate.
Instruction:	Indicate a first position for the rotation. Indicate a second position for the rotation. Choose another transformation or select OC to complete the rotation.
Options:	Not available.
Result:	The part will be rotated as indicated, around the current rotation centre. The angle of rotation applied will be the difference between an imaginary line drawn from the first position indicated to the current rotation centre, and an imaginary line drawn from the second position indicated to the current rotation centre.

Dynamically

Purpose:	To dynamically rotate a part around its current centre of rotation.
Prerequisites:	A nest must be open. A part must be selected to be rotated.
Instruction:	Rotate the part as desired. Choose another transformation or select OC to complete the rotation.
Options:	**Options** can be used to toggle between **Rotate Dynamically** and **Move Dynamically**.
Result:	The part will be rotated as indicated.

Define Centre

Purpose:	To define a new rotation centre for a currently selected part.
Prerequisites:	A nest must be open. A part must be selected.
Instruction:	Indicate the position of the new rotation centre on the selected part. Choose another transformation or select OC the complete the function.
Options:	Not available.
Result:	The selected part will be given a new rotation centre. This point will remain as the centre of rotation for the part until a different centre is defined using this function.

Mirror U-axis

Purpose:	To mirror a part on a horizontal line through the centre of the part.
Prerequisites:	A nest must be open. A part must be selected to be mirrored.
Instruction:	None.
Options:	Not available.
Result:	The part will be mirrored as described above. Choose another transformation or select OC the complete the function. Note that all marking lines will be now be on the opposite side of the part to their original position.

Mirror V-axis

Purpose:	To mirror a part on a vertical line through the centre of the part.
Prerequisites:	A nest must be open. A part must be selected to be mirrored.
Instruction:	None.
Options:	Not available.
Result:	The part will be mirrored as described above. Choose another transformation or select OC to complete the function. Note that all marking lines will now be on the opposite side of the plate from their original position.

Lock U-axis

Purpose: To restrict the transformation of a part so that no translation will occur in a horizontal direction.

Prerequisites: A nest must be open. A part must be selected to be transformed.

Instruction: Immediately select another transformation from the **Transform** toolbar.

Note: that this function is only active for one transformation. This function will not affect dynamic moves or rotations.

Options: Not available.

Result: The part will be transformed as indicated, except that no horizontal translation will occur.

Lock V-axis

Purpose: To restrict the transformation of a part so that no translation will occur in a vertical direction.

Prerequisites: A nest must be open. A part must be selected to be transformed.

Instruction: Immediately select another transformation from the **Transform** toolbar.

Note: that this function is only active for one transformation. This function will not affect dynamic moves or rotations.

Options: Not available.

Result: The part will transformed as indicated, except that no vertical transformation will occur.

4 Positions

Purpose: To transform a part by indicating four cursor positions. This transformation is similar to a combination of a 2 Position move and a 2 Position rotation.

Prerequisites: A nest must be open.

Instruction: Select the part to be transformed. Indicate the four points for the transformation.

Options: Not available.

Result: The part will be moved from the first indicated point to the third indicated and rotated by the difference between an imaginary line from the first point to the second point and an imaginary line from the third point to the fourth point.

Hull Plate Nesting

1 Nesting System - General

This document is a user-oriented description of the Nesting System system of Hull. It concentrates on the functional characteristics of the system and on the interfaces to other subsystems of AVEVA Marine. The graphical part is based on the Drafting System, see *User's Guide Drafting*. See also *User's Guide Drafting* for available colours.

2 Nesting System - Overall Characteristics

2.1 Purpose

The Nesting System is a tool for nesting plate parts and for creating information relevant to the nesting work.

The plate parts to be nested are usually supplied by other subsystems but can be created and modified within the Nesting System. With the system, parts are nested on a parent plate and the tool path is defined creating a generic file to be used as input to a suitable postprocessor. Optionally, a shop drawing (burning sketch) can be created and printed.

2.2 System Components

The Nesting System consists of programs, data banks and data files. The information flow in the system is illustrated in the following figure (where rectangles denote programs, circles denote data and double circles denote objects stored in data banks).

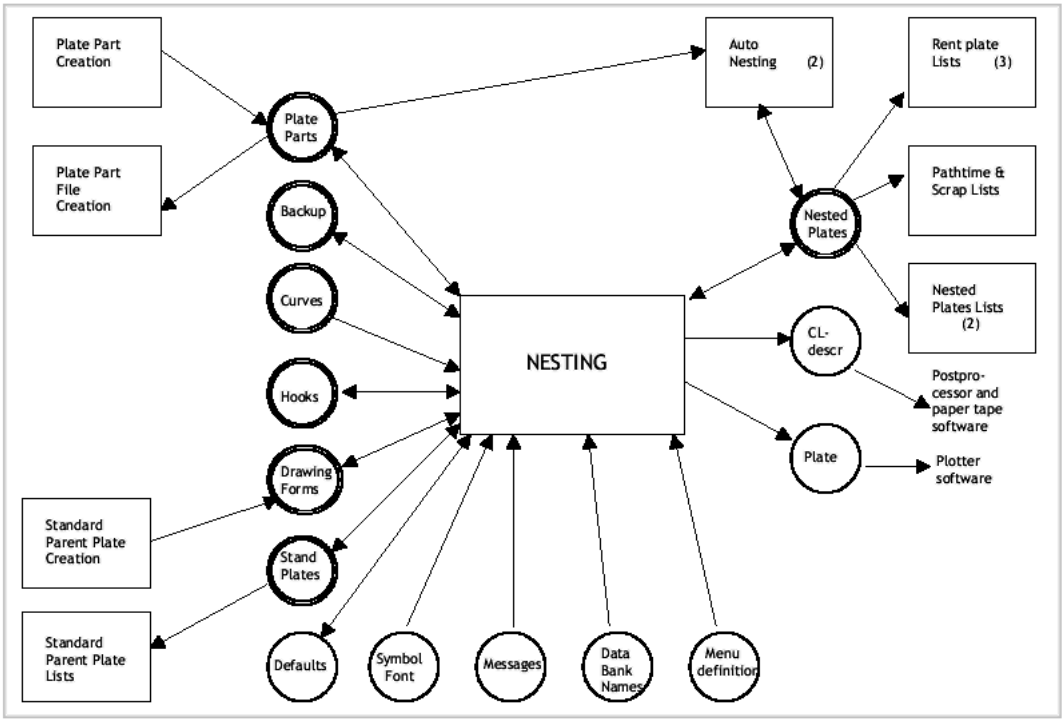


Figure 2:1. Figure showing the Information Flow in the Nesting System.

Only the nesting program must be run at a workstation.

The different types of data are either separate data files or organized in data banks.

The Nesting System has the following major interfaces to other subsystems of AVEVA Marine.

- Generic files created in the system are postprocessed and handled by the Generic Post Processor (see *Hull Post Processors, User's Guide*). The user routines in this context also depend on the burning machine control unit.

2.3 Input

The following data are input to the system:

- Plate parts.
- Standard parent plates.
- Standard hooks.
- Default information.
- Standard drawing forms.

  The system contains in itself functions for the establishment of standard parent plates, standard hooks, default information and standard drawing forms.
- Symbol font.

  The symbol font number 2 contains user defined symbols to be used on the shop drawing. In addition, the SBHM symbol font 92 is used to display different types of excess.

  In the Nesting System there is no special user font for symbols. All system symbols and user-defined symbols are placed in font number 2.

  It is however possible for the user to create his own symbol font in the drawing part of the system and use this font as font number 2. The symbol font current for a certain project must be named d012sy002.sbs.

  There are however, a number of system symbols which always must be defined in font No.2:

20-25	(restplate handling)
60-79	(position number)
80-99	(position number)

  These symbols must be taken from the font delivered by with the AVEVA Marine software.

  All other symbol numbers are free to use but it is advisable to use the numbers >= 100. In this way it is possible, if necessary, to introduce new system symbols with number < 100.
- Message and menu files.

2.4 Result

The results of the system are:

- Nested plates. The nested plates are stored and can be retrieved for modifications.
- Prints.
- Generic files.

- Plate parts.
- Standard plates.
- Lists.

2.5 Backup and Recover

The nesting program contains an automatic backup facility.

At predefined intervals, the plate in the work space is copied automatically to a backup data bank. If the system, for some reason (e.g. power failure), breaks down, the backup copy can be retrieved.

The backup is controlled via the VM Drafting default file.

2.6 Imperial Units

Imperial units are supported.

3 Nesting System - Application Functions

3.1 Fundamental Concepts

Some fundamental concepts in the Nesting System need further explanation.

3.1.1 Coordinate System

A nested plate is described in a coordinate system with the origin in the lower left-hand corner of the parent plate. A plate part which is created in the system, uses an operator defined coordinate system.

3.1.2 Automatic Renesting

The system always uses the latest possible versions of the plate parts when an old nested plate is brought up for modifications. This is the automatic renesting feature. The feature also implies the recreation of production information (burning bridges, burning starts, etc.), if a plate part has been modified since the last time it was nested or if an already nested plate part is transformed on the plate.

3.1.3 Burning Bridges

The system offers 9 kinds of burning bridges:

1. Gap
2. Double-centred bridge (positive or negative)
3. Single-centred bridge
4. Cross-centred bridge
5. Double asymmetric bridge (positive or negative)
6. Single asymmetric bridge
7. Cross asymmetric bridge
8. Double asymmetric bridge in cutout
9. Gap including cut free geometry

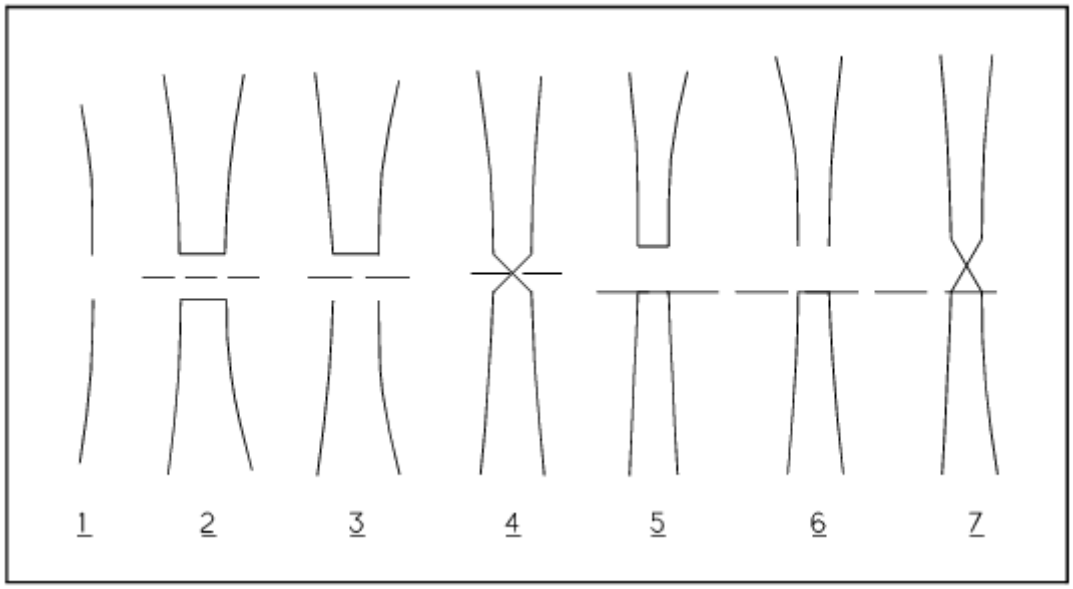


Figure 3:1. Illustration of the Bridge Types 1-7

Centred and asymmetric refer to how the bridge will be positioned relative to the cursor positions defining the bridge. A bridge can be defined with one or two cursor positions defining a "control line". A centred bridge is positioned so that the "control line" is in the centre of the bridge, while an asymmetric bridge is positioned so that the "control line" coincides with a bridge line or (for cross bridges) one end of the cross (see picture).

Bridges of types 3, 5, 6 and 7 also need a position to determine the side, i.e. the side where to place the gaps (types 3 and 6) or the side where to place the second bridge line (type 5) or side where to place the cross (type 7). The width of the bridges is part of the default information.

Type 1, gap, is the simplest of the *bridges* and can be defined using either one or two cursor positions. The system works in the following way, when a gap is defined.

- A given position is used to identify the closest part contour. The *foot point* on that contour will be calculated and the gap will be placed around that *foot point*.
- If the gap is defined by using two cursor positions, the first position is used to identify the closest part contour. If the activity code for node point preference is > 0 and the given point is within this distance from the node point, then the first gap line will pass through the node point. The second position is used to indicate on which side the second gap line shall be placed.
- If the distance from the node point is larger than the one given by the defaults the result is the same as with one cursor position.

A bridge defined with one cursor position will be treated by the system in the following way.

- The given position is used to identify the two closest part contours. On each contour, the *foot point* will be calculated and the line between the two foot points will be the control line.

A bridge defined with two cursor positions will be treated by the system in the following way.

- The line between the two given positions will be used to intersect the two closest part contours. The line between the two intersection points will be the *control line*.

Bridges can also be defined with a bridge line **from a node point on one contour to a node point on another contour** (e.g. at common cutouts). Such a bridge must be created using one cursor position, and the default information controls the feature.

Bridges can also be defined to have a **forced vertical direction** (types 2, 3, 5 and 6) by setting a default value. If no intersection between the contours exists or if the bridge lines get too long, an ordinary bridge will be created.

The **bridge of type 8** is defined in the same way as the bridge of type 5. The difference is that the bridge lines connect segments in the same contour in the bridge of type 8.

- This bridge is used in cutouts and when the tool path is verified (cf.VERIFY) the cutout will be burned as a hole.

When the **gap of type 9** is used the operator has the possibility to define a cut free geometry which has its origin in the gap. The gap itself is identical to the type 1 gap.

- The cut free geometry is a string line with an arbitrary number of line segments. When the last point is closer to a plate part or the raw plate edge than a default distance, this point will automatically be adjusted so that the point will be situated this default distance from the plate part/raw plate edge.
- The cut free geometry will automatically be included in the verification of the tool path. The automatic restart facility is then used to resume the burning of the plate part.

The **bridges of type 2 and type 5** can be defined either as positive or negative. It is possible to switch between positive and negative by pressing REJECT when the system prompts the user to give the first point defining the bridge. The default type is positive.

All bridges defined with two cursor positions can be made either horizontal or vertical. This is achieved by giving LOCK V or LOCK U *before* the second position is defined.

3.1.4 Auxiliary Functions

All "elementary" auxiliary functions (e.g. tool on/off, marking on/off, marking offset on/off, kerf compensation, rapid, stop, etc.) are generated automatically by the system. The function codes with which the Nesting System will supply the postprocessor are the symbolic codes used in the system.

Auxiliary functions for special purposes can also be defined by the operator. They can be defined at a node point, an arbitrary point on a contour or in a corner loop at 3 pre-defined points. The function codes (and possible parameters) are transferred to the postprocessor without change, except for function values in the range -900 to -1000, which are reserved for special auxiliary functions not treated by the postprocessor (e.g. corner loops).

To control bevel cutting for instance, it is necessary to give additional information to some auxiliary functions, using parameters.

When the auxiliary functions are defined, one function is given at a time, together with one or several parameters separated by a comma (,). The user interface for giving auxiliary functions in a corner loop is the same as for single auxiliaries.

The logical variable SBH_NESTAUX must be assigned to an arbitrary value and the auxiliary function should not be preceded by 99.

3.1.5 Starts

Marking and burning starts are either explicitly defined by the operator by indicating the position for a single start or automatically generated. The order in which the starts will appear in the tool path is the order in which they have been created. (The system contains a function for renumbering the starts.)

Marking starts will always be placed at the end of a marking line and are indicated on the display with an arrow-head. The following figure illustrates the definition of a single marking start:

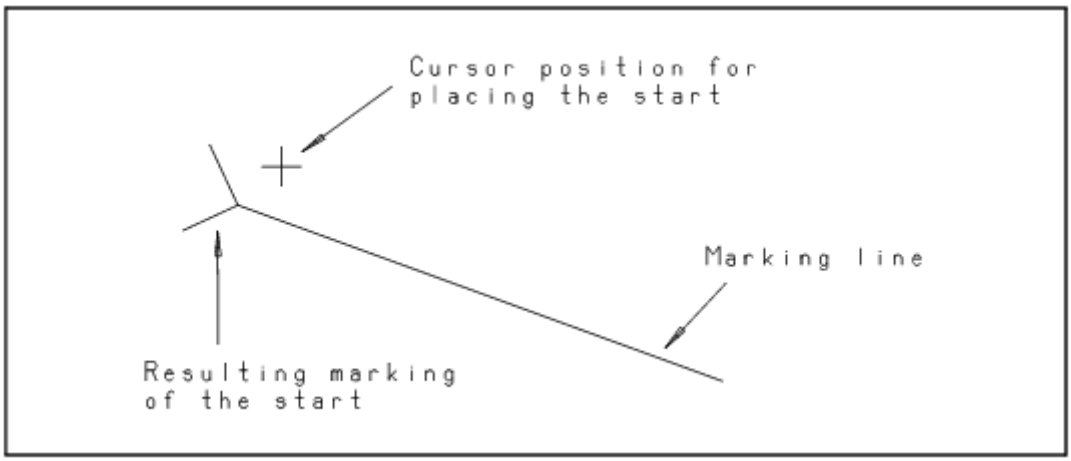


Figure 3:2. Illustration of a Single Marking Start.

Starts can be generated automatically for all marking lines. This operation implies the deletion of possible existing starts on the marking lines.

Burning starts can be placed in gaps, in node points or in arbitrary points. A burning start is indicated on the display with the hook at the start (or if no hook is used, an arrow-head just like marking starts).

The end of the tool path connected to a burning starts is defined at the same time as the start. The following three combinations of starts and ends can be used:

1. Start in a gap - end in the first following gap in the tool path (possibly the same gap as the start gap).

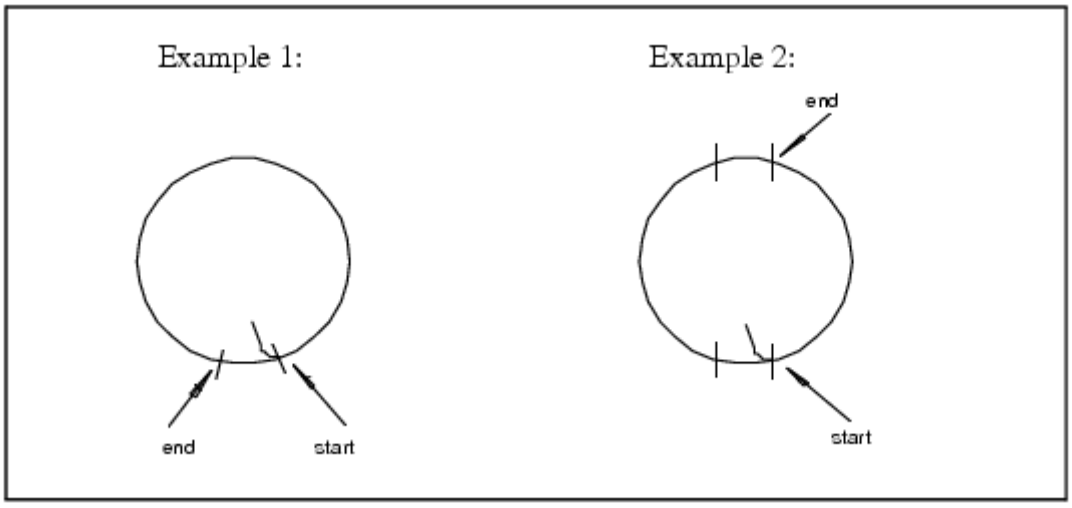


Figure 3:3. Illustrations of Starts in a Gap.

2. Start in a node point or arbitrary point end in the same point (if a gap has been placed in the path, the tool path will end at that gap unless the automatic restart feature is used).

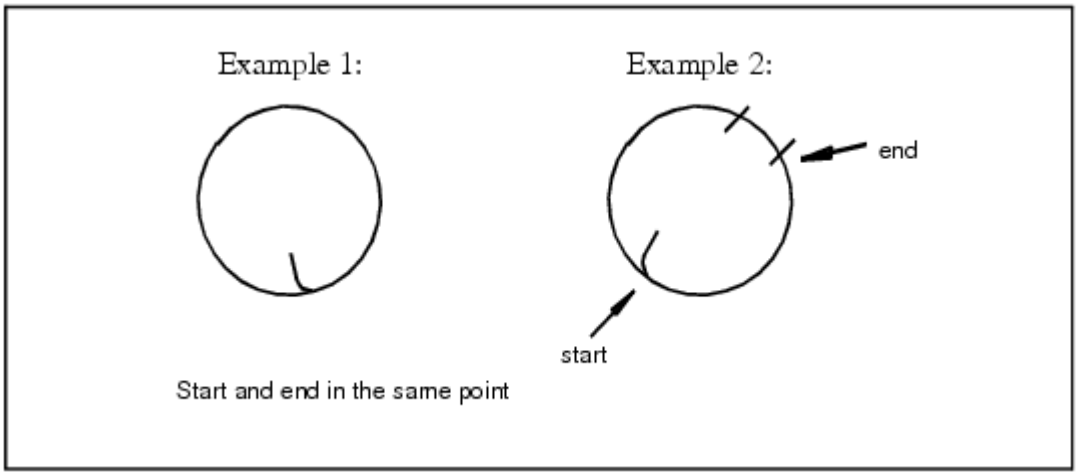


Figure 3:4. Illustrations of starts in a node/arbitrary point.

3. Start in a node point or arbitrary point end in another arbitrary point or node point. (If a gap has been placed in the end, the tool path will end in that gap unless the automatic restart feature is used.)

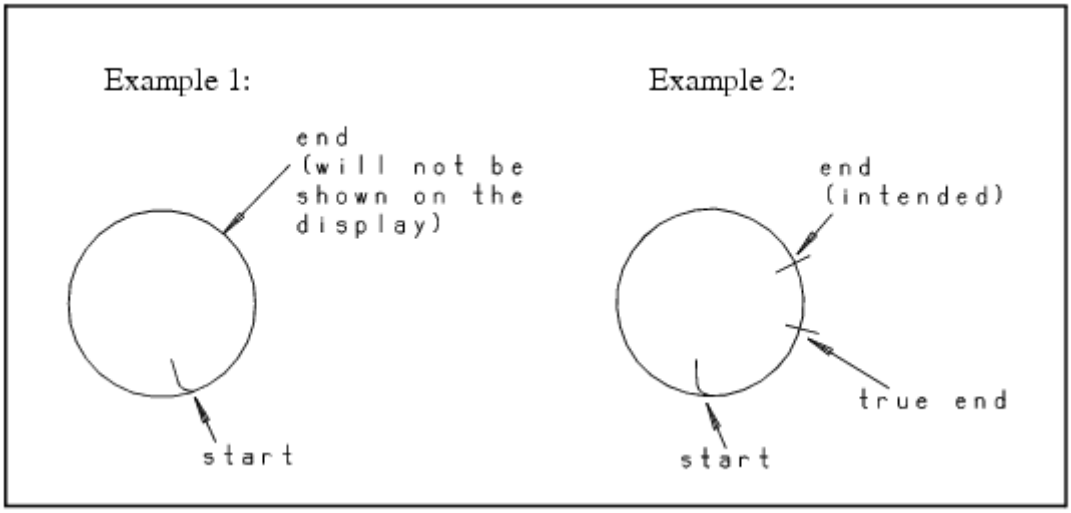


Figure 3:5. Illustrations of Starts in a Node/arbitrary Point.

Starts in all the holes can be generated automatically. If starts in gaps are used, gaps will be generated as well. The generation implies the deletion of possible existing starts in holes and gaps connected to starts.

When the burning starts are generated, either automatically or singly it is possible to guarantee a certain direction of the tool path. By setting one default value for holes and one for the outer contours, the operator is ensured that the cutting sequence will be either clockwise or counterclockwise.

Labelling starts can only be generated automatically. The generation implies the deletion of possible existing labelling starts.

It is also possible to define a cut line geometry when defining a start. The cut line geometry is a string line with an arbitrary number of line segments. When the first or last points are

closer to the raw plate edge than a default distance, this point will automatically be adjusted so that the point will be situated this default distance from the raw plate edge.

The cut line geometry will automatically be included in the verification of the tool path.

3.1.6 Automatic Restart

An automatic restart facility can also be used, if wanted. Each time a gap is encountered, the current burning path is terminated, the tool will go idle to the other end and the burning will restart.

3.1.7 Restplate Handling

In the Nesting System restplates can be handled. The treatment of the created restplates is dependent on the plate quality.

For the normal steel quality (which is defined in the default system) the created rests are only marked with texts in the burning sketch. Two types of texts are created, one for normal restplates and one for rests which are too small to be classified as normal restplates and yet too large to be considered as scrap. The latter rests are called workshop rests.

The names of the normal restplates are defined in an external file. The file name should be assigned to the environment variable

SBH_NSQ_RESTPLATE_TABLE.

The format of the file is:

```
<restplate 1>   <thickness 1>
<restplate 2>   <thickness 2>
....
```

One restplate and the thickness for which it is valid should be given on each line and each thickness may occur only once.

For the special steel quality the restplates are stored on the standards data bank and can later be used in another nest job in exactly the same way as any standard parent plate.

It is possible to define any number of rests in a nest job. The total number of generations is limited to 32767.

A restplate can be defined in a number of different ways:

1. Using original geometry
2. Using cut line geometry in a start
3. Using an imaginary line and original geometry (special steel quality only)

When a restplate is to be created using the original geometry of the plate parts the user must define a closed contour inside which the relevant parts of the plate part geometry is situated. This contour can either be a rectangle (defined with two points) or a polygon (defined with a string line). The defined contour may not intersect any plate part in more than two points.

The system then captures the plate parts that are inside the defined contour and automatically combines the original geometry of the plate parts included. The geometry between the plate parts will be a line between the points where the contour intersects the plate parts. The original geometry can then be combined with the existing parent plate to obtain the restplate contour.

If the restplate area is situated completely inside the parent plate the contour must be defined with a polygon which does not intersect the parent plate contour.

If bridges or cut free geometries in gaps are to be included in the original geometry the contour must be a polygon. The bridges must then be a part of the defined contour. This is obtained by pointing close to the endpoints of the bridges when defining the string line. For cut free geometries in gaps each segment must be identified.

Holes can also be defined as restplates. In this case it is sufficient to identify the hole.

When a restplate has been created that part of the parent plate is not available any longer. Therefore the parent plate contour is modified when restplates are created. For holes and restplates which are situated completely inside the parent plate the contour numbers are stored in the nested plate and the parent plate is not modified.

A restplate can also be created using a cut line geometry in a start. In this case the cut line geometry is combined with the parent plate contour to get the restplate contour. The parent plate contour is modified.

The third way to define a restplate (special steel quality only) is to define an imaginary line. The restplate contour is created exactly as when the original geometry was used. Furthermore, an imaginary line is created and stored with the restplate. It is defined with a string line in the same way as a cut free geometry. The bump and overlap functions will then use the imaginary line instead of the original geometry when the restplate is used in the Nesting System.

The name of the restplates consists of three parts:

```
<restname><delimiter><counting number>
```

The first part is common for all rests with the same thickness and quality. The maximum length of <restname> is 20 characters. The second part is a delimiter which is defined in the default system. The default is a hyphen ('-') but any character which is not included in first part may be used. It is not possible to use different delimiters for different restplates.

The last part is a counting number between 0001 and 9999. The counting number can either be given by the user or selected automatically by the system.

Thus each restplate will have a unique name.

Before any restplates can be created a standard parent plate with negative length and width must be created on the standards data bank. This is done with the program se012 (CREATESTDPL).

In this administrating object the following information about all created restplates with the same quality and thickness is stored:

- The name of the nest job where the restplate was created
- The name of the current parent plate
- The name of the nest job where the restplate has been used

If the total quantity of the nest job is larger than 1 there are two possibilities: either one restplate is created in N copies or N restplates with different names are created in one copy.

Nesting plate parts on a restplate works in principle as nesting on a standard parent plate. The only difference occurs when the total quantity is larger than 1. Either one restplate is used in N copies or N restplates are used in one copy.

The restplates used in a nest job will be locked from the beginning until the nest job is stored. In this way it is not possible that any rest is used in more copies than it is defined in.

When a restplate has been used in so many copies as it was created in, it is deleted from the standards data bank.

The administrating object is locked from the beginning of the restplate creation until the nest job is stored. This will ensure that the administrating object always is correct.

When a restplate has been created it is possible to make a classification of the rest. There are six different forms available:

- rectangle
- circle
- triangle
- symmetric T-form
- L-form
- trapeziform

The dimensions can be given either directly or by taking a measure.

It is also possible to create a restplate with one of the forms mentioned above when initiating a new nest job. This is made by giving the name of the administrating object as parent plate. The created restplate will be given a counting number.

It is also possible to delete restplates. The administrating object will also be updated.

When the verification of the tool path is made on a restplate the user must indicate the explicit start position. If OPERATION COMPLETE is given the values given in the default file are used.

3.2 Functions, Overview

Below, a survey of the functions of the system will be given. The functions correspond to the application part of the menu. To get an exhaustive description of how to proceed at a certain step in a function, see *Drafting / Operator's Instructions Drafting*.

(In the overview, all function names are printed in block letters.)

3.2.1 Transformation Functions

The transformation functions are all subfunctions of two of the nesting functions: NEST and TRANSFORM. Consequently, they cannot be used without actuating one of the two main functions first.

The transformation functions are used in a transformation sequence, when a plate part (or possibly a group of plate parts) is to be positioned. Thus, they are actuated one after the other, until the part is properly positioned.

The transformation functions are of two kinds: dynamic and static. A dynamic function implies that the part is moving on the screen during the transformation (e.g. rotating). A static function (e.g. mirroring) implies that only the positions before and after the function can be seen by the operator.

There are two dynamic functions: ROTATE DYNAMICALLY and MOVE DYNAMICALLY, of which one is always modal. This is normally the last one used but in the initial phase of a transformation sequence, i.e. when a part (or group of parts) has been newly appointed to be transformed, MOVE DYNAMICALLY is the modal one. To switch from one to the other, **OPTIONS** should be given. To actuate the modal one, however, a position at the part (or group of parts) currently to be transformed shall be given. Depending on which one of the

two functions that is modal, the part can then be moved or rotated. The next given position will uncouple the part and allow for a new choice of transformation function (or acceptance of the position). In these dynamic transformations the toggling between envelope and full display is solely controlled by mouse-button2.

The static functions will either be executed immediately after actuation (e.g. **MIRROR U** will immediately cause current plate part to be mirrored) or will prompt the operator to give some additional information to the system (e.g. positions, angles).

- **ROTATE DYNAMICALLY**

  This function is used to appoint rotation to be the modal dynamic function (cf above). To actuate the function, a cursor position must be given and the selected part(s) can be rotated around the current rotation centre (cf. ROTATION CENTRE). To finish the rotation, a new position must be given. Also see the function ROTATE 2 CPs.

- **ROTATE 2 CPs**

  With this function, a part can be rotated around the current rotation centre (cf. ROTATION CENTRE) using 2 cursor positions, which will be requested. The angle between the line from the rotation centre to the first position and the line from the rotation centre to the second position will be used to rotate the part. (This function is the static correspondence to ROTATE DYNAMICALLY.)

  The following figure illustrates the action of the function.

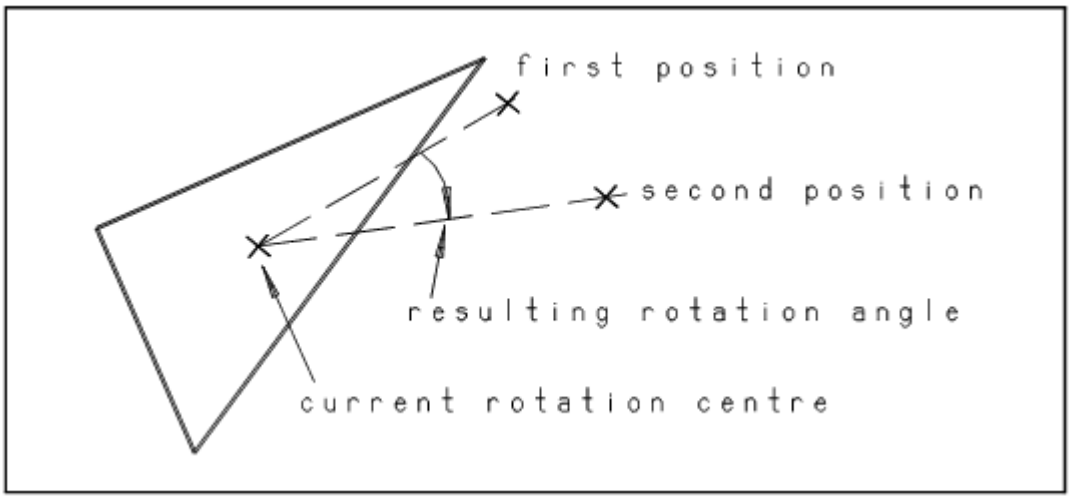


Figure 3:6. Figure showing the Action of the Function.

- **ROTATE BY ANGLE**

  This function is used to rotate a part by a given angle, which the operator will be requested to key in. The part will then be rotated around a current rotation centre (cf. ROTATION CENTRE).

- **90, -90, 180, 45**

  These four functions are used to rotate a part 90, -90, 180 or 45 degrees around the current rotation centre (cf. ROTATION CENTRE). Upon actuation, the rotation will be instantaneous.

- **ROTATION CENTRE**

  Upon actuation of this function, the current rotation centre will be displayed as a cross. To change the centre, the position of the new centre shall be indicated with the cursor. If the centre is not to be changed, OPERATION COMPLETE shall be used instead.

  The default rotation centre will always be the geometric centre of the part (or group of parts) to be rotated, i.e. the centre of the least circumscribed axis-parallel rectangle of the part before the first rotation.

- **MOVE DYNAMICALLY**

  This function is used to appoint translation to be the modal dynamic function (cf above). To actuate the function, a cursor position must be given. The part (or group of parts) can then be dragged. To finish the dragging, a new cursor position must be given.

- **MOVE 2 CPs**

  With this function, a part can be moved using 2 cursor positions. The vector from the first to the second position will defined the translation. (This function is the static correspondence to MOVE DYNAMICALLY.)

- **MOVE DX, DY**

  With this function, a part can be moved a desired distance horizontally and/or vertically. The operator will be prompted to key in the two distances, first the horizontal (negative if left), then a comma and finally the vertical (negative if down). The translation will then be instantaneous.

- **MIRROR U**

  This function is used to mirror the current part in a horizontal line through the centre of the part.

- **MIRROR V**

  This function is used to mirror the current part in a vertical line through the centre of the part.

- **TRANSFORM 4 CPs**

  This function is used to transform the current part giving four cursor positions. The translation part of the transformation is defined by the first and the third positions - the first is translated to the third. The rotation angle will be the angle between a line through the first two positions and a line through the last two.

  (The transformation can be thought as a transformation of one coordinate system into another. The first position indicates the origin and the second one a point on the positive u-axis of the original coordinate system. The third position indicates the new origin and the fourth position a point on the positive part of the u-axis of the new coordinate system.)

  The following figure illustrates the action of the function:

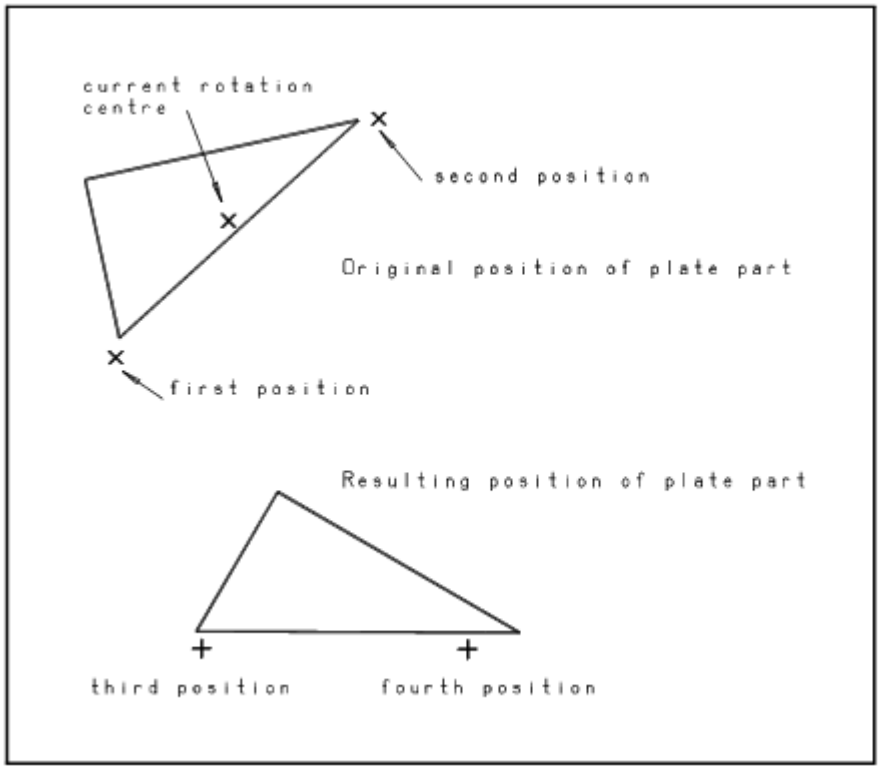


Figure 3:7. Figure showing the Action of the Function.

- **LOCK U**

  This function is used to restrict a transformation, so that no horizontal translation occurs (but the vertical translation and/or rotation is carried out as usual).

  This function must be actuated immediately before the one for which it shall apply and is only valid for one transformation at a time. It will have no effect on the dynamic transformations.

- **LOCK V**

  This function is used to restrict the vertical part of a translation and is analogous to LOCK U.

- **PARALLEL**

  This function is used to transform the current part with a parallel and distant constraint. The operator will be prompted to indicate first a line in the current part, then a line in another part or possibly a plate edge. (If an arc is indicated instead, the tangent at the indicated position will be used.) The operator will, in this case, be notified for acceptance. Then the transformation will occur instantaneously. The current part will be rotated, so that the two indicated lines become parallel (the least possible rotation angle will be chosen). Then the part will be moved, so that the distance between the two indicated lines will be the default distance and so that the indication points come as close as possible to each other. (The second indication defines on what side of the first line the second line shall be placed.) Answering REJECT instead of indicating the first line adds some other features to the function. The default distance can be replaced with a keyed in distance (or twice the kerf compensation for common cuts) or the rotation part of the function can be excluded, so that

the distance is measured from a node point in the current part (e.g. a corner) to the second line.

The following figure illustrates the action of the function in a simple case:

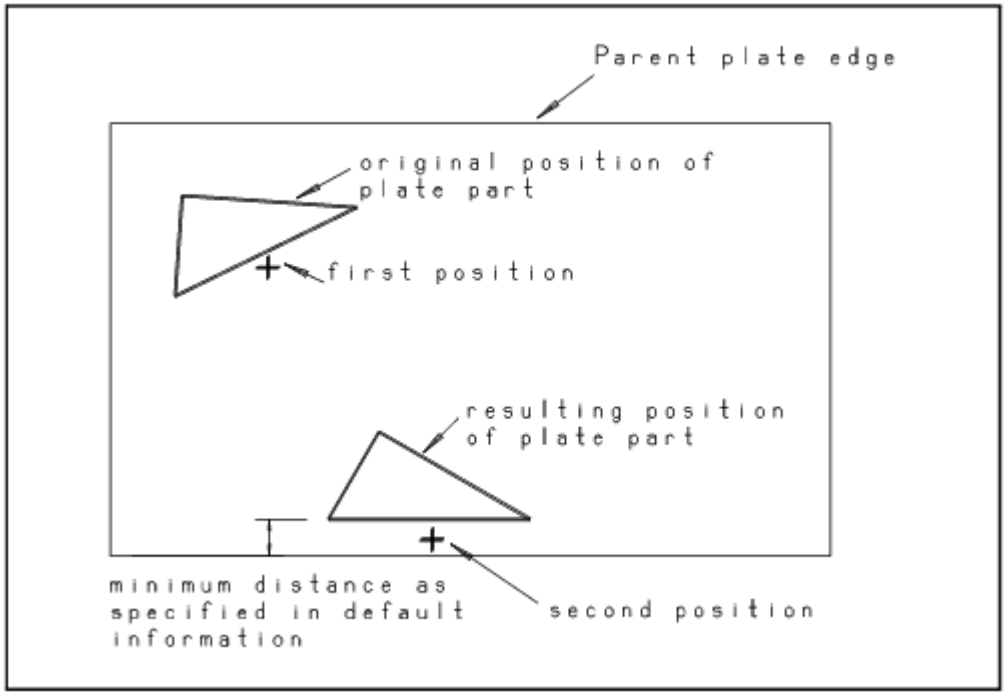


Figure 3:8. Figure Showing the Action of the Function in a Simple Case.

- **BUMP**

With this function, a part can be moved a distance determined by the system, in a specific direction given by the user. The distance is the largest distance the part can be moved without overlapping any other part, the parent plate or any cut free geometry. Default clearances between parts and between a part and the parent plate is taken into account when the distance is determined.

The BUMP function menu is displayed modally and all other transformations can be performed at any time.

The parts can be transformed in nine different directions:

`+x`	along the positive x-axis
`-x`	along the negative x-axis
`+y`	along the positive y-axis
`-y`	along the negative y-axis
`+x +y`	along the positive x-axis, along the positive y-axis
`+x -y`	along the positive x-axis, along the negative y-axis
`-x -y`	along the negative x-axis, along the negative y-axis

`-x +y` along the negative x-axis, along the positive y-axis

`dir` along any direction which is indicated by two cursor positions

In the four directions which are a combination of transformations along the x and y axes the system alternates between transformations in the x and y directions until it is not possible to move the part any further.

When the bump function is used in TRANSFORM it is possible to bump several plate parts simultaneously.

3.2.2 Nesting Functions

The nesting functions are used mainly to start a nesting job, to handle the parent plate and the plate parts to be nested, to supply the nested plate with burning starts, burning bridges, auxiliary functions, etc. and to produce a Generic file.

- **NEW NESTING JOB**

Context sensitive menus have been added to the nesting application. These menus are available both in nesting and burning sketch area in the application. The dialogues for New nest job and Open nest job have been changed.

- **Plate nesting**

The new dialogue for initialisation of a nest includes basic nest information, raw plate definition and burner selection. Each part of the dialogue is briefly described below the picture.

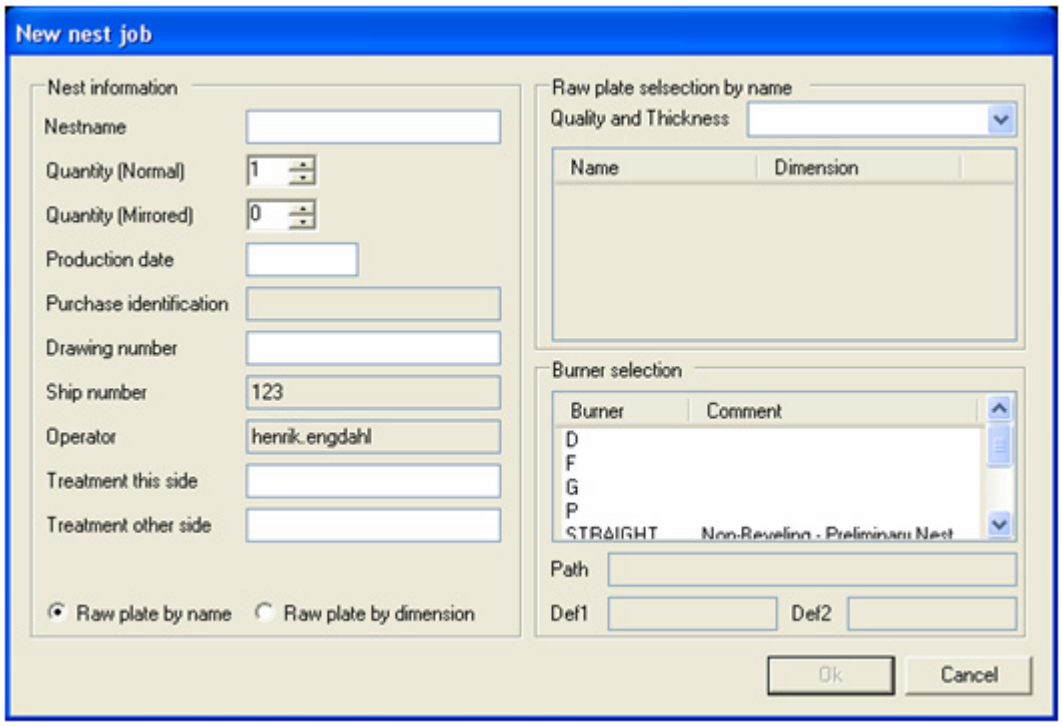


Figure 3:9. New nest job dialogue

Basic information

This part of the dialogue allows the user to input basic information, for example nest name, quantity and production date. Some of these input fields will be dimmed when the system provides the information or restrictions are made in the project setup.

Raw plate definition

The raw plate definition can be supplied in two ways, by dimension or by selection in a list of raw plates that matches given quality and thickness. Raw plate by dimension requires that length, width and thickness are given.

Burner selection

The burner selection can be done when a valid raw plate has been selected or given by dimension. Available burners are listed in a list view where the user selects one burner for cutting the plate parts.

- **Panel Line Control**

The new dialogue for initialisation of an assembly nest includes basic nest information, part selection and burner selection. Each part of the dialogue is briefly described below the picture.

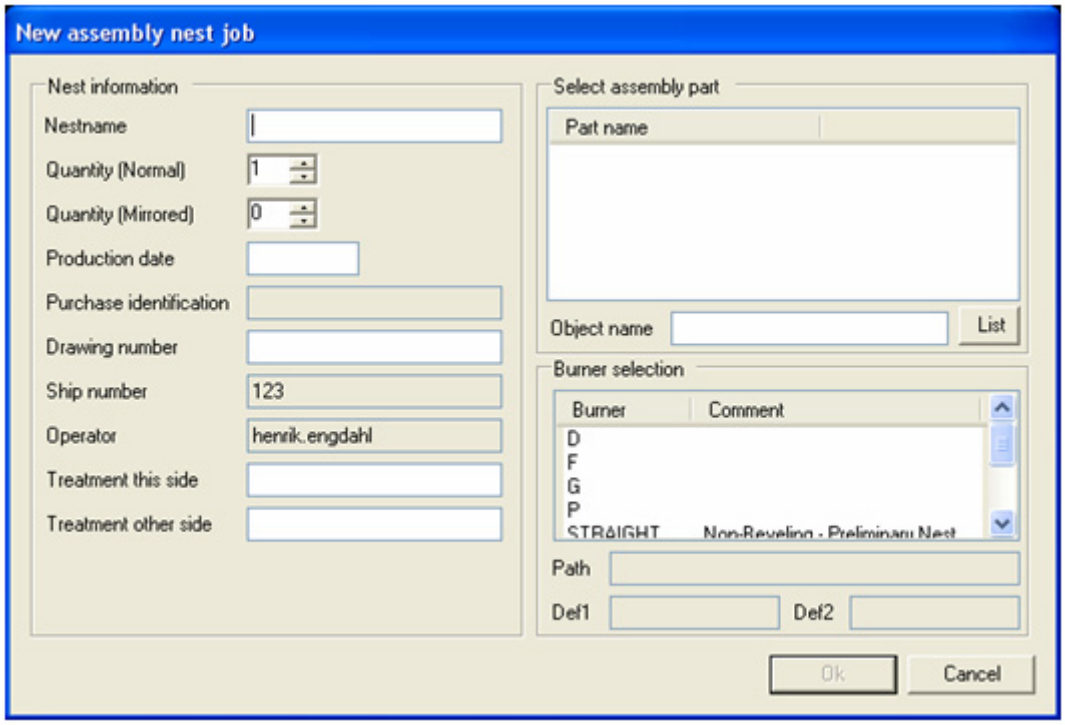


Figure 3:10. New assembly nest job dialogue

Basic information

This part of the dialogue allows the user to input basic information, for example nest name, quantity and production date. Some of these input fields will be dimmed when the system provides the information or restrictions are made in the project setup.

Part selection

List all assembly parts that matches the given filter string. Then the user selects one of the listed assembly parts to be nested.

Burner selection

The burner selection can be done when a valid assembly part has been selected. Available burners are listed in a list view where the user selects one burner for cutting the assembly part.

- **OPEN NESTING JOB**

The Open nest job dialogue is similar to The New nest job dialogue. It lists the stored information about the nest job. The user can edit some fields depending on settings in default files. Exchange of burner for nest job can be made if the settings allow it, otherwise the burner information will be listed in a similar way as the raw plate information.

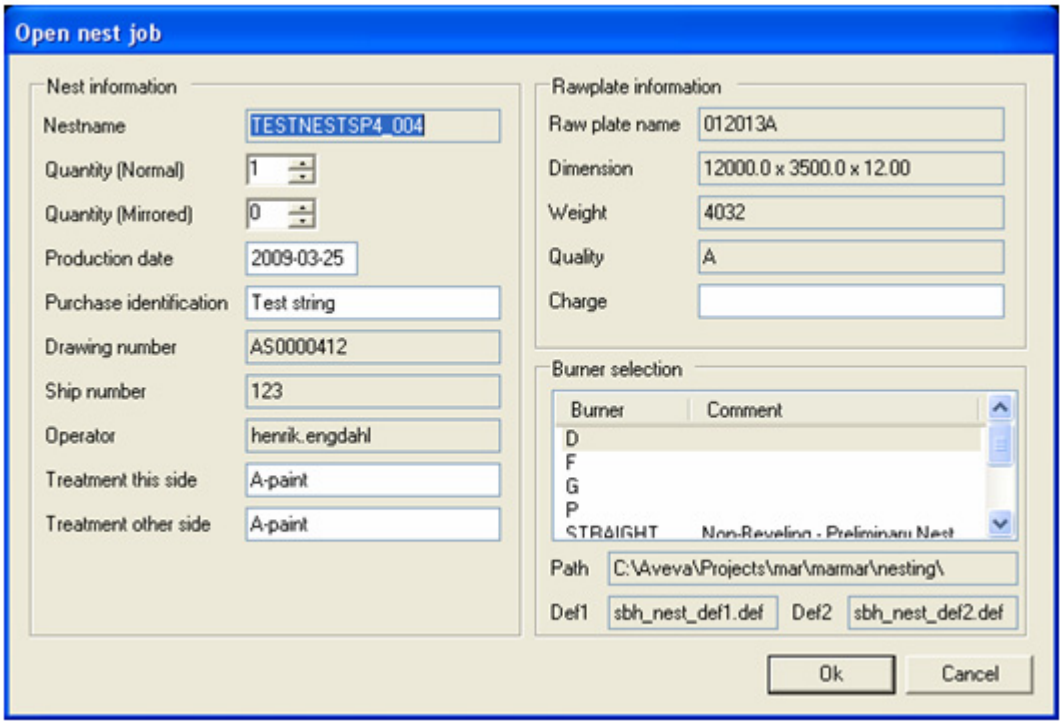


Figure 3:11. Open nest job dialogue

- **OPEN MULTIPLE**

This function is used to bring up a number of existing nested plates for modification. The nested plates will automatically be recreated in the same way as for normal open of existing nested plates.

If a multiple nesting session is open, the user will be prompted for starting a completely new session or to add more nested plates to existing multiple sessions.

- **SAVE NESTING JOB**

  This function is used to store the current plate, which makes it possible to retrieve the plate for modifications (cf. *OPEN NESTING JOB*).

  The function is also intended for backup purposes. It may be wise to store the current work as backup, if the system, for some reason, breaks down (e.g. power failure). In that way, only a small amount of work is wasted.

  Note: that the Generic file and the work shop drawing are not created or stored using this function (cf. *VERIFY*).

  When the nested plate parts have been fetched from a parts menu file, a comparison is made between the nested plate and the file. If some plate parts have been nested in too many copies the plate will be stored with the extension _ERR to the name. If the Generic file has been created it will be deleted.

  There will be given messages to the user which plate parts that have been nested in too many copies and also the number of copies missing in the plate parts data file.

  All plate parts nested on the plated will be marked with an asterisk in the data file and will not be processed the next time that file is used. This is valid both for the NEST and PARTS MENU functions.

- **RETURN TO DRAWING**

  The function returns control to the drawing part of the Nesting System. If a drawing was current when the user entered the nesting part of the system, this drawing will be available to the user.

- **EXCHANGE RAW PLATE**

  The Exchange/Select raw plate dialogue has two ways to supply raw plate information, by dimension or by name. By name means that the user selects a quality and thickness and selects one of the available raw plates listed. Raw plate by dimension means that the user inputs length, width and thickness. The purchase information for selected or given raw plate can be edited.

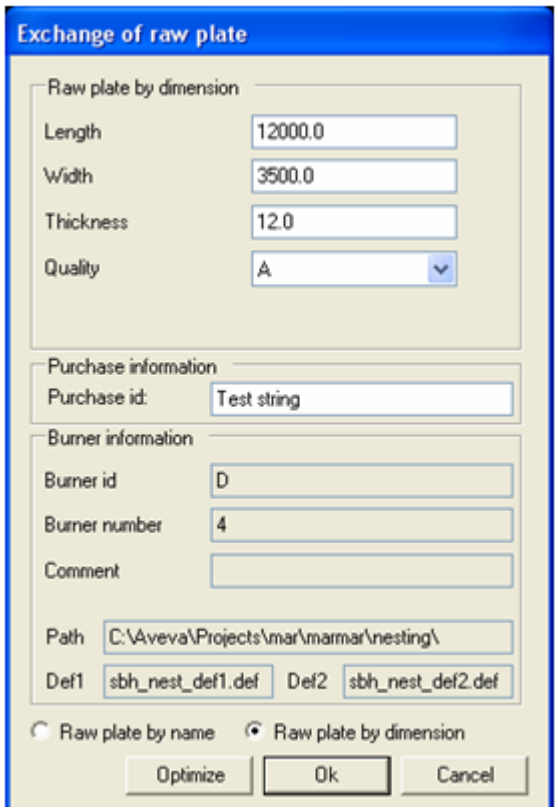


Figure 3:12. Exchange/Select raw plate

- **RENAME PLATE**

  This function is used to rename the current plate. The renaming will not affect the plate stored in the data bank, only the one in the work space.

- **DELETE ON DATA BANK**

  This function is used to delete any kind of objects from the data banks used in the system. The deletion will not affect objects in the work space. It is not possible to delete restplates with this function (cf. *DELETE RESTPLATE*).

- **LIST DATA BANK**

  This function is used to list any kind of objects in the data banks used in the system. It is possible either to list a single object by giving its full name or several objects by using wild cards (% replacing one character and * replacing any number of characters.

- **CALCULATE DISTANCE**

  This function is used to calculate the distance between two cursor positions and to get the coordinates of the positions.

- **DEFAULTS**

  This function is used to handle default information. Default files can be read, the current default information can be stored as a default file, the current information can be changed or the current information can be listed.

- **NEST**

  This function is used to nest one or several plate parts on the current parent plate. The part to be nested is identified by keying in the part name, by picking the part from the parts menu (cf. *PARTS MENU*) or by keying in the name of a data file containing part names and position numbers. When a data file is used it is possible to select between part name and position number.

  After a part has been identified, it can be transformed into its proper position using the transformation functions. When the part is at its final position and OPERATION COMPLETE has been given, the next part can be nested.

  When a plate part has been accepted it is possible to make an overlap check automatically. Intersecting segments will be displayed in a different colour.

  An optional modification of the NEST function is that the plate parts are automatically mirrored so that the side with the largest number of marking lines is shown.

- **QUICK NESTING**

  The basis for the **Quick nest** function is a selected number of plate parts, a 'parts menu', on the current plate.

  The function means that parts will be selected automatically from this menu in order of decreasing area/perimeter by answering ALL on the system prompt for a part identification. In this selection, it is possible to automatically cluster parts, i e combine parts two by two, with approximately the same size and shape in order to improve the utilization of the raw plate. This clustering of parts can also be used in manual nesting (OPTIONS is used to switch to clustering).

  Note: that these parts are only treated as a cluster during the nesting process. After positioning them on the plate no connection exists between the clustered parts.

  The **Nesting BUMP** function is then used automatically to place the plate part on the raw plate and the *best* position is selected. ‘Best’ means either that the extension of the nested parts in the x-direction or that the scrap is minimized.

  In the automatic process, the parts are put outside the right raw plate edge in a user-defined number of different positions from the upper edge down to the lower edge of the raw plate. At each starting position the part is rotated (the angle increment is user-controlled).

  Mirroring is optional. Nesting in holes is supported. The plate can contain parts nested previously.

  At any time the Quick nest algorithm can be interrupted (by pressing the space bar) and the user automatically gets the possibility to transform the last part(s) that was/were nested. The last nested parts are then made current and after making the wanted transformations, the Quick nest process continues.

  The interrupt facility can also be used to switch between clustering and no clustering (cf. *CLUSTERING ON/OFF*).

A serious drawback in this implementation is that free areas that are *shaded* by parts to the right of them, cannot be utilized by the automatic algorithm (with the exception of holes in the plate parts). This must currently be compensated for by manual intervention.

- **CLUSTERING ON/OFF**

  With these functions in the **Nesting parts** menu it is possible to switch between clustering and no clustering during the quick nesting. The nesting algorithm is interrupted (with the space bar) and the wanted function is selected from the menu.

- **PARALLEL NESTING**

  In the Nesting System it is possible to automatically nest identical parts in parallel in the same plate. Two parts are considered to be identical if they belong to the same block and have the same position number and outer geometry.

  The parallel nesting can be done in two different ways. The user can nest one part in the ordinary way and the system automatically finds an identical part if it exists and places it on the nested plate as close to the main part as possible. Two parts are never placed at a distance less than the minimum distance between the two tool sets of the burning machine. The user can then continue with the next part. It is possible to switch between nesting and parallel nesting by giving OPTIONS.

  Another possibility is to nest a number of parts singly. The parts are then connected with bridges. This group can now be identified and the system will automatically find an identical part for each part in the group, and nest them as close to the main group as possible. The bridges will also be copied.

  The parallel nesting is controlled via a number of default parameters. The restriction for the burning machine is defined with the parameters EQ1_MIN_X, EQ1_MIN_Y, EQ1_MAX_X, EQ1_MAX_Y, EQ2_MIN_X, EQ2_MIN_Y, EQ2_MAX_X, EQ2_MAX_Y, DIST_MAIN_PARALLEL.

  It is also possible to define a thickness interval where parallel nesting is possible. This is made by using the defaults parameters THICK_PARALLEL_MIN, THICK_PARALLEL_MAX.

  Two different single burners are supported, one main burner and one precision burner. The thickness intervals for respective burner is defined with the default parameters THICK_PRECISION_MIN, THICK_PRECISION_MAX, THICK_MAIN_MIN and THICK_MAIN_MAX.

  To make it easier to distinguish between single nested parts, the main and parallel part, colours can be defined for the three types of parts: NEST_SINGLE_COLOUR, NEST_MAIN_COLOUR and NEST_PARALLEL_COLOUR respectively.

  The post processor supports the NCE520 controller using ESSI format.

- **AUTOMATIC NESTING**

  This function is used for automatic nesting of a plate. To activate the function the user has to initiate a new nest job and add a parts menu (cf. *PARTS MENU*) with the parts to be automatically nested. No plate parts can previously have been nested on the plate.

  The automatic nesting function is optional. It requires that the user has access to a layout generator via the *Plug-In Interface,* see *Nesting Plug-In* .

- **BUMP CURVE ON/OFF**

  With these functions in the **Nesting parts** menu it is possible to turn on/off the bump curves on the nested parts. This curve is the maximum curve for a plate part with all bevel taken into account. It is defined when the part contains continuously varying bevel. In the verification of the tool path the guiding contour is always used.

- **TRANSFORM**

  With this function, an already nested part (or group of parts) can be transformed (repositioned). After actuation, the part is identified using the cursor. Each part will be marked with the circumscribed rectangle. When all parts to be transformed have been chosen, OPERATION COMPLETE will finish the identification and all transformation functions will be available. If e.g. starts and burning bridges have been defined for any of the parts, they will be recreated, if possible after transformation.

  A duplication can be obtained, if needed. Duplication implies that all transformations indicated, are applied to the original plate part (or group of parts) a predefined number of times leaving a copy every time.

  Note: that all transformations are merged into a transformation matrix. Repeated transformation implies the transformation of the matrix, which may lead to unintended results, if more than one of the basic transformations (rotation, translation, mirroring) are used. Duplication can be obtained by answering OPTIONS when the system prompts the user to identify a part to be transformed.

  In this function it is also possible to make the overlap check where intersecting segments will be displayed in a different colour.

- **OVERLAP CHECK**

  This function makes an overlap check for all plate parts and cut free geometries on a plate. The intersections are indicated in colour in the same way as when the overlap check is used in the NEST and TRANSFORM functions.

- **DELETE PART**

  With this function, a nested part can be deleted from the current plate. All starts, burning bridges and other information involved will be deleted as well. If a part was nested from a parts menu (cf. below) it is automatically put back in the same position in the menu as before.

  With this function a single part or all parts on the parts menu can be deleted as well.

- **EXCHANGE PART**

  This function is used to exchange a single part for a new version or a totally different part. The new part will be transformed exactly as the old one and all starts, burning bridges and other information involved will be recreated, if possible.

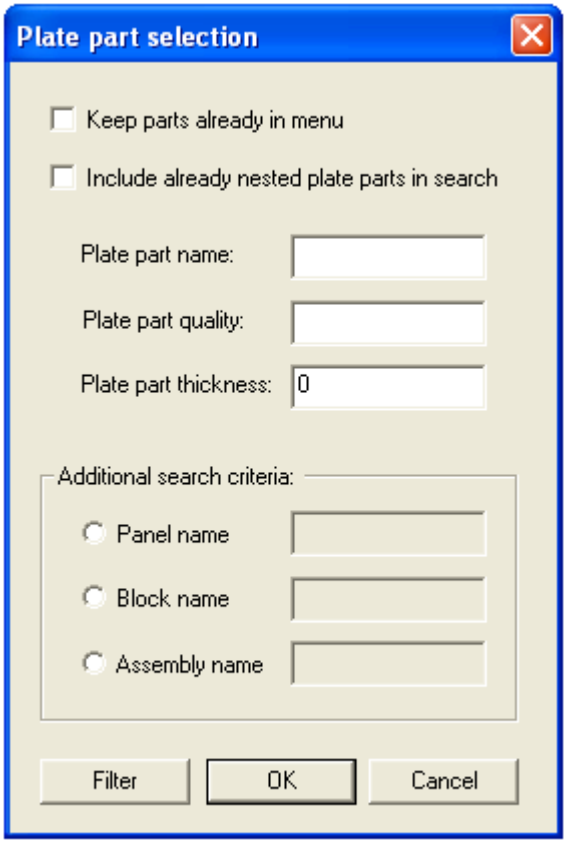
  Note: that if an old plate is brought up to be modified, the latest versions of all plate parts are automatically renested (an automatic exchange occurs). Also note that one part cannot be exchanged for a totally different part, unless they are both defined approximately at the same place in a common coordinate system.

- **MOVE PART**

  In a multiple nesting session it is possible to rearrange the nestings by moving one or several parts from one nesting to another. It is also possible to add new parts. No burning sketch can be created in this mode. The multiple nesting session must first be closed and then the individual nestings will be stored. It is then necessary to retrieve them individually to define the tool path and create the burning sketch.

- **PARTS MENU**

  This function is used to make a parts menu. A parts menu is a cluster of parts positioned in a separate window for later selection of single parts. The parts to appear in the parts menu are identified by selecting parts in a plate part selection dialog.



When **Filter** is selected the user gets the possibility to edit the Nesting Filter File. For a complete description of the filter possibilities, see *The Nesting Filter File in Chapter Initialisations for Nesting*.

Each part will be scaled down. Along with the part geometry, the part name will also be displayed in the menu. When a part later is selected to be nested (cf. NEST) it will be scaled up to its real size.

An optional modification of the PARTS MENU function is that the plate parts are automatically mirrored so that the side with the largest number of marking lines is shown.

The selection of plate parts can also be made using the assemblies on the top level. This is controlled with the default parameter PARTS_MENU_SELECTION. Possible values are PART and ASSEMBLY.

IWhen a plate part has been put in the menu by the user, the system automatically searches after identical parts among the remaining parts in the file. All other parts which are identical with the current part, are put into the parts menu behind the first one but they are initially invisible and not possible to access. The number of identical parts for each assembly is presented below the parts in the menu.

The criteria for two parts to be identical are:

- the same area and number of segments in the outer contour
- identical or mirrored segments

There are no checks on assemblies and position numbers.

When the first of the identical parts has been nested the next identical part will be visible and nestable. The number of remaining identical parts is updated.

The presentation of identical parts in the parts menu is controlled by the default parameter, PARTS_MENU_EQUAL. The possible values are YES and NO.

- **BRIDGE**

  This function is used to define a burning bridge giving one or two cursor positions. To switch between one and two positions the user answers OPTIONS when the system prompts for a position.

- **DELETE BRIDGE**

  This function is used to delete a burning bridge, which the operator will be prompted to identify, ALL is supported. Any cut free geometry defined in the bridge will also be deleted automatically.

- **CORNER LOOP**

  This function is used to define a corner loop. The operator will be prompted to indicate where the corner loop will be placed and to key in the radius and the length of the corner loop. See the figure below.

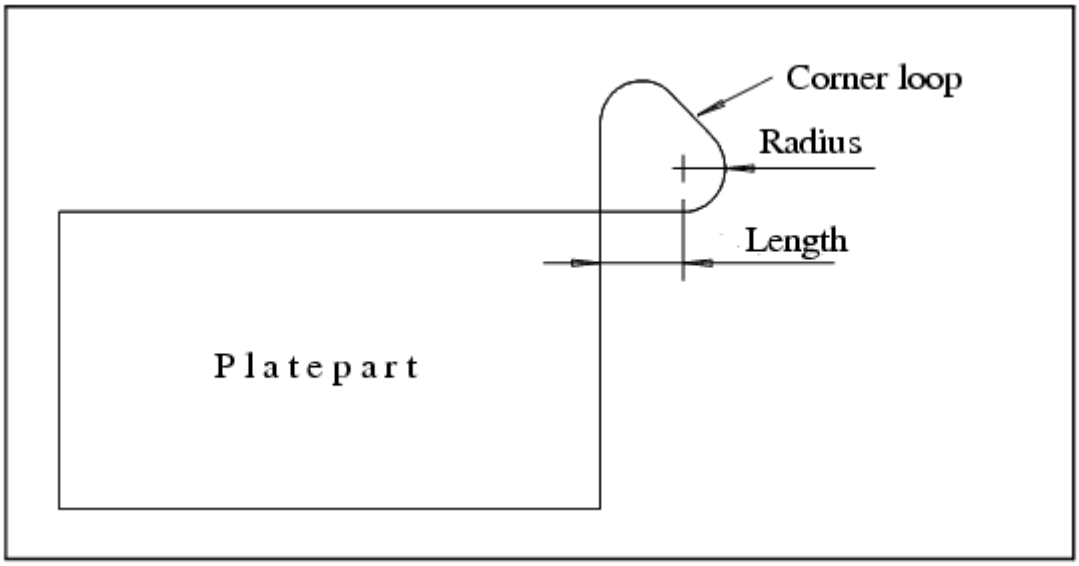


Figure 3:13. Illustration of a Corner Loop.

It is also possible to give auxiliary functions in three different points in a corner loop.

Corner loops can be added automatically when a plate part is nested. This is controlled with the nesting default parameter CORN_LOOP_CTRL. The default parameter CORN_LOOP_LENGTH must then be defined. If CORN_LOOP_RADIUS is zero a triangular corner loop will be created.

Corner loops can also be made in the function AUXILIARY FUNCTION by giving the function code -999 and the radius and length as parameters. If a corner loop will be made in this way, other auxiliary functions are not allowed in that node.

There is a possibility to modify the geometry of an existing loop if the user points within a certain distance from the loop. This distance controlled via the default parameter CORN_LOOP_MODIFY_DIST.

- **AUXILIARY FUNCTION**

  This function is used to define auxiliary functions. The operator will be prompted to indicate where the functions are to be valid and to key in the auxiliary function code together with possible parameters.

- **DELETE AUXILIARY FUNCTION**

  This function is used to delete an auxiliary function, which the operator will be prompted to indicate, ALL is supported.

  Note: that corner loops will be deleted as auxiliary functions.

- **START BURNING**

  This function is used to indicate a single burning start, in which case the operator will be prompted to indicate the start position and possibly the tool direction or to create burning starts for all the holes on the plate automatically.

If the user gives REJECT instead of a start position it is possible to temporarily change the type of the start or change the start and/or end hook.

The default information contains further information about how the start shall be defined, what hook to be used etc. The start order will be the order in which starts are defined but can be changed using START SEQUENCE.

- **START MARKING**

  This function is used to indicate a single marking start, in which case the operator will be prompted to indicate the start position or to create marking starts on all marking lines automatically. The start order will be the order in which starts are defined but can be changed using START SEQUENCE.

- **START BLAST**

  This function is used to indicate a single blasting start, in which case the operator will be prompted to indicate the start position or to create blasting starts on all blasting lines automatically. The start order will be the order in which starts are defined but can be changed using START SEQUENCE.

- **START AUTOMATIC**

  This function is used to create all marking and burning starts on a nested plate automatically. The order between the marking and the burning is controlled with the default parameter MARK_ALL_FIRST. If the value is YES, the marking starts will be created for the whole nested plate, and the burning starts afterwards part wise (holes first). If the value is NO, all starts are put part wise in the following order: marking, holes, outer contour. The parts are sorted in increasing x co-ordinates and decreasing y co-ordinates.

- **START CUSTOMISED**

  The function for creating customised starts is optional. It requires that the user has access to a sequence generator via the *Tribon Plug-In Interface*, see *Nesting Plug-In*.

- **START LABELLING**

  This function is used to create labelling starts automatically for all texts and symbols to be labelled. The start order will be the order in which starts are defined but can be changed using START SEQUENCE.

- **MOVE START**

  This function is used to move a burning or a marking start, keeping the start order. Parallel starts are supported. If the start is placed in a gap, the gap is moved as well.

- **DELETE START**

  This function is used to delete a start, which the operator will be prompted to indicate, ALL is supported. Any cut line geometry defined in the start will also be deleted automatically.

- **START SEQUENCE**

  This function is used to show the current start sequence or to change the sequence. The sequence is shown by displaying the sequence number at each start. The text height can be changed by selecting OPTIONS.

The sequence can be changed either by indicating all starts in the correct order or by keying in the order number of a single start and indicating the start on the screen.

- **SHOW PRODUCTION INFORMATION**

  This function is used to display production information selected by the operator. The following items can be shown:

  - The bevel information of a certain plate part.
  - The excess information of a certain plate part.
  - The side information of a certain plate part.
  - The name of a certain plate part.
  - The part name of a certain plate part.
  - The name of the panel to which a certain plate part belongs.
  - The position number of a certain plate part.
  - The weight of a plate part.
  - Customer data. The General Purpose Strings of a certain plate part. Either one of the four or all strings are chosen. In the latter case the strings are concatenated using the ':' character.
  - The block name of a certain plate part.
  - The ship number of a certain plate part.
  - The production date and the splitting date of a certain plate part.
  - Number of starts.
  - Total burning length.
  - Total marking length.
  - Total idling length.
  - True scrap percentage.
  - Efficiency scrap percentage (the scrap percentage calculated relative to the least circumscribed rectangle rather than the true parent plate).
  - Area of all nested plate parts.
  - Area of parent plate.
  - Area of the smallest axis-parallel rectangle circumscribing all the nested plate parts.
  - The date and time for the latest tool path verification.

  Some items are calculated during the tool path verification (cf. VERIFY) and reflect the situation at the verification rather than the present situation.

- **LABELLED TEXTS**

  This function is used to define texts to be labelled. It is found in the 'Nesting tools' menu.

  The text functions in the 'Shop dwg Info' menu have not been changed and can be used as before to create texts in the burning sketch.

  There are 3 types of labelled text:

  - Text to be printed
  - Text to be labelled but not printed
  - Text to be labelled and printed

The text type for a labelled text will be taken from default parameter TEXT_TYPE. The user has however, the possibility to temporarily change the type interactively for a single text by giving OPTIONS before the text is positioned in the nested plate.

When labelled texts are to be defined, the nested plate must be displayed. The reason for this is that starts have to be set for these texts so that a tool path can be defined. Therefore it is also not possible to use existing texts in the burning sketch as labelled texts.

The labelling of texts has the following characteristics:

- The text to be labelled can be the position number (text only), any production information, a user defined text, or semi-automatic labelling (plate part data or raw plate data).
- The texts can be changed, copied, moved and deleted.
- In order for a text to be labelled, a labelling start must be set.
- The labelling sequence in determined automatically by the system in the same way as for the burning sequence for holes.
- The labelling information will be processed in the order given by the default parameter WORKING_SEQUENCE.

In the verification of the labelled texts, output is created to the postprocessor and the tool path is shown in the burning sketch. If the text type is 0 or 2 the text to be labelled will also appear in the sketch.

The labelled text facility has not been implemented in the old Generic file but only in the new Generic format that can be created by the Nesting System.

The definition of the semi-automatic labelling text takes place via an ordinary text file, created and maintained in the standard editor of the computer system. The name of the file can be selected freely and its total name should be assigned to the logical variable SBH_NEST_LABEL_BLOCK.

The file is divided into a number of statements using a special language (cf. *"Interpretative Language"* in *"User's Guide Basic Features"*).

The file is essentially in three parts. In the first part a number of alternative delimiters are defined, in the second part the strings are set and in the last part the strings are attached to plate and part.

The syntax for defining alternative delimiters is:

```
ALT_DELIM, <designation> /TEXT='<delimiter>';
```

<designation>	This variable is used to reference the alternative delimiter in a STRING statement.
<delimiter>	This is the string defining the actual alternative delimiter.

If no alternative delimiter(s) will be used, this statement can be omitted.

The syntax for defining strings is:

```
STRING,                <designation>           /
CONTENTS=(<stringlist>)/
DELIM='<delimiter>';
```

<designation>	This variable is used to reference the string in the PLATE and the PART statements.
<stringlist>	This is a comma separated list of the actual data that will be presented in the string. Possible data are listed below. Alternative delimiters can be used in the list with the following exceptions: You cannot use two alternative delimiters in sequence and you cannot end the string with an alternative delimiter.
DELIM	This attribute specifies the delimiter between statements. It will be overridden when an alternative delimiter is place between two data.

The syntax for the plate and part statements are defined as follows:

```
PLATE (or PART) /STRINGS=(<list>) [/POSITION=<pos> [/
DELTA_X=<dx>]
                          [/DELTA_Y=<dy>] [ANGLE=<ang>]
```

<list>	A comma-separated list of the designations of the defined strings.
POSITION	(Optional) Used to position the text. For PLATE this can be one of CURSOR, LOWER_LEFT, UPPER_LEFT, LOWER_RIGHT, UPPER_RIGHT. For PART it can only be CURSOR. Default is CURSOR.

DELTA_X, DELTA_Y and ANGLE can be used together with POSITION to further specify the position. ANGLE is to be given in degrees.

Below follows an example of the input file.

Example:

```
ALT_DELIM, ALT1 /TEXT='*';
ALT_DELIM, ALT2 /TEXT='-';
ALT_DELIM, ALT3 /TEXT='-->';
STRING, STR1
          /CONTENTS=(NEST_NAME, RAW_THICKNESS, RAW_WIDTH,
ALT1,
                                RAW_LENGTH, ALT2, QUALITY) /
DELIM=' ';
STRING, STR2
      /CONTENTS=(OPERATOR, ALT2, SHIP_NO, NEST_DATE,
                                 SCRAP_BRUTTO) /DELIM=' ';
STRING,  STR11
         /CONTENTS=(PART_NAME, BLOCK_NAME, BLOCK_NUMBER,
ALT2,                                    POSITION_NUMBER) /
DELIM='::';
STRING,  STR12
      /CONTENTS=(ASSEMBLY_HIGH, ALT3, ASSEMBLY_LOW) /
DELIM=', ';
PLATE /STRINGS=(STR1,STR2) /POSITION=UPPER_RIGHT
      /DELTA_X=-30 /DELTA_Y=-30 /ANGLE=-90;
PART /STRINGS=(STR11,STR12);
```

Below follows the **keywords** to be used in the CONTENTS attribute.

For use in PLATE statement:

NEST_NAME
STANDARD_NAME
DEF_FILE_1
DEF_FILE_2
PURCH_INFO
CHARGE
DRAWING_NO
SHIP_NO
OPERATOR
THIS_SIDE
OTHER_SIDE
TOTAL_QUANTITY
NORMAL_QUANTITY
MIRROR_QUANTITY
FORM_SCALE
NEST_DATE
PRODUCTION_DATE
POSTPROC_DATE
START_POINT
QUALITY

```
DENSITY
RAW_THICKNESS
RAW_WIDTH
RAW_LENGTH
RAW_AREA
RAW_WEIGHT
USED_PLATE_LENGTH
USED_PLATE_WIDTH
USED_PLATE_AREA
USED_PLATE_WEIGHT
REST_PLATE_NAME
REST_PLATE_TYPE
REST_PLATE_NPAR1
REST_PLATE_NPAR2
REST_PLATE_NPAR3
REST_PLATE_NPAR4
REST_PLATE_LENGTH
REST_PLATE_WIDTH
REST_PLATE_AREA
REST_PLATE_WEIGHT
PLATE_PARTS_NUMBER
PLATE_PARTS_MIN_POSNO
PLATE_PARTS_AREA
PLATE_PARTS_WEIGHT
SCRAP_BRUTTO
SCRAP_USED
WASTE_AREA
WASTE_WEIGHT
BURN_STARTS
BURN_LENGTH
BURN_IDLE
BURN_NAME
BURN_NUMBER
BURN_TYPE
BURN_ID
BURN_PROCESS_NAME
BURN_TOOL_ID
BURN_PARALLEL_STARTS
BURN_PARALLEL_LENGTH
BURN_PARALLEL_IDLE
MARK_STARTS
MARK_LENGTH
MARK_IDLE
```

MARK_PARALLEL_STARTS
MARK_PARALLEL_LENGTH
MARK_PARALLEL_IDLE
BLAST_STARTS
BLAST_LENGTH
BLAST_IDLE
BLAST_PARALLEL_STARTS
BLAST_PARALLEL_LENGTH
BLAST_PARALLEL_IDLE
GSD_STARTS
GSD_LENGTH
GSD_PARALLEL_STARTS
GSD_PARALLEL_LENGTH
SIGN_STARTS
SIGN_NO_OF_CHARACTERS
SIGN_IDLE
For use in PART statement:
NEST_NAME
STANDARD_NAME
DEF_FILE_1
DEF_FILE_2
PURCH_INFO
CHARGE
DRAWING_NO
SHIP_NO
OPERATOR
SPLIT_DATE
PRODUCTION_DATE
EFFECTIVE_AREA
PART_NAME
PANEL_NAME
POSITION_NUMBER
BLOCK_NAME
BLOCK_NUMBER
SHIP_NUMBER
LOCATION
DRAWING_NUMBER
ASSEMBLY_HIGH
ASSEMBLY_LOW

For use in PART statement:

ASS_DESC_LOW
ASS_DESC_LOW1

```
ASS_DESC_LOW2
ASS_DESC_LOW3
ASS_DESC_LOW4
CUST_DATA1
CUST_DATA2
CUST_DATA3
CUST_DATA4
WORK_LOCATION
POSITION
```

- **LABELLED SYMBOLS**

  This function is used to define symbols to be labelled. It is found in the 'Nesting tools' menu.

  The symbol functions in the 'Shop dwg Info' menu have not been changed and can be used as before to create symbols in the burning sketch.

  There are 3 types of labelled symbol:

  - Symbol to be printed
  - Symbol to be labelled but not printed
  - Symbol to be labelled and printed

  The symbol type for a labelled symbol will be taken from default parameter TEXT_TYPE. The user has however, the possibility to temporarily change the type interactively for a single symbol by giving OPTIONS before the symbol is positioned in the nested plate.

  When labelled symbols are to be defined, the nested plate must be displayed. The reason for this is that starts have to be set for these symbols so that a tool path can be defined. Therefore it is also not possible to use existing symbols in the burning sketch as labelled symbols.

  The labelling of symbols has the following characteristics:

  - The symbol to be labelled can be the position number (symbol and text) or any symbol from a symbol font.

    The symbols can be changed, copied, moved and deleted.
  - In order for a symbol to be labelled, a labelling start must be set.
  - The labelling sequence is determined automatically by the system in the same way as for the burning sequence for holes.
  - The labelling information will be processed in the order given by the default parameter WORKING_SEQUENCE.

  In the verification of the labelled symbols, output is created to the postprocessor and the tool path is shown in the burning sketch. If the symbol type is 0 or 2 the symbol to be labelled will also appear in the sketch.

- **DELETE LABELLED TEXT**

  This function is used to delete a labelled text which the user will be prompted to indicate. All labelled texts can be deleted by answering ALL.

- **DELETE LABELLED SYMBOL**

  This function is used to delete a labelled text symbol which the user will be prompted to indicate. All labelled symbols can be deleted by answering ALL.

- **VERIFY**

  This function is used to perform a number of different (but related) operations which are operator selectable:

  1. The tool path can be verified visually on the screen, with the beam simulating the tool. The operator will be prompted to verify each start and each explicitly defined auxiliary function. The geometry will be saved temporarily in the picture object. It will be removed when storing the nested plate or verifying again.
  2. A generic file can be created. It will get the same name as the plate and can be transferred to the postprocessor at a later stage.
  3. The frame work of a shop drawing (burning sketch) can be created, containing the burning path (solid lines), the marking path (dashed and dotted lines), the idling path (dashed line) and the rectangle of the parent plate (solid lines) and labelled texts and symbols. The frame work can later be completed using the shop drawing functions.

  An optional addition is that marking lines caused by a folded flange will get a text from the default file.

  4. When the parent plate is a restplate the user has the possibility to redefine the starting point for the burning machine. The new point will be the node point on the restplate which is closest to the position indicated by the user. If the user presses OPERATION COMPLETE the default value will be used.

  In the burning sketch the whole restplate is displayed (dashed and double dotted lines).

  Furthermore, at the verification some production information is calculated (cf. *SHOW PRODUCTION INFORMATION*).

- **CREATE RESTPLATE**

  This function is used to create restplates. A number of texts will be created in the burning sketch. The restplate will be stored on the standards data bank for special steel quality only.

- **DELETE RESTPLATE**

  This function is used to delete restplates from the standard data bank. The administrative object will be updated. The deletion will not affect objects in the work space. It is not possible to delete standard parent plates with this function (cf *DELETE ON DATA BANK*).

- **OPEN HOOK**

  This function is used to retrieve a hook from the hook object D003HOOKS. A drawing is created with the hook number as drawing name. The geometry of the hook can then be changed using the drawing part of the Nesting System.

- **SAVE HOOK**

  This function is used to save a drawing as a hook in the hook object D003HOOKS. The drawing name will be used as contour number in the hook object.

A hook must be described in its own coordinate system, ending at the origin with the positive u-axis as the continued tool path and the scrap side above the u-axis. No check of this is made when the hook is stored.

- **CHECK POST PROCESSOR**

  This function is used to interactively check the result from the Generic Post Processor (GPP). The GPP must be run twice to generate both the input to the burning machine and the input file to this function. The input to the burning machine is displayed in a list and the corresponding geometry is displayed at the screen when ALL is selected. The user can step by step check that the resulting geometry is correct and that the various auxiliary functions are given at the correct positions. Different colours and line types can be used for the different types of movement as well as for the different types of bevel.

- **DEFINE BEVEL**

  When some bevel needs to be defined in a very late stage of the production this function can be used. The user indicates the contour of the part and then selects the appropriate bevel from a list with the available bevels taken from the bevel control file. This should be repeated fro all bevel intervals in the contour, also where no bevel is wanted. Dotori is supported and for each dotori interval both start and end bevels must be defined. OPTIONS is used to toggle between dotori and normal bevel. The system bevel code 999 is used by this function and it must be defined in the bevel control file.

- **CREATE CURVE**

  For a complete documentation of this function, see *Hull Planar Modelling.*

3.2.3 Create Parts and Profiles

- **Create Parts**
  - General

    This function is used to create plate parts with, optionally, holes and marking lines. Some information is also asked for and added to the part.

    The plate parts are then stored on the plate parts data bank associated with the logical name SB_PLDB.

    This function must not be used for the updating of plate parts created in hull modelling.
  - Function

    The function asks the user to indicate the boundary of the plate part. All closed contours that are located totally inside the contour describing the boundary of the plate part will be stored as holes. All other lines, arcs or contours will be stored as marking lines. These will however be cut at a distance given in the default file.

    When the plate part is approved by the user, the function asks for name, quality, position number, thickness, build number and date of production.

    The side where the plate part is to be valid may be given by the user, i.e. port side, starboard or symmetric.

    The location of the plate part may be given by the user. The system asks for a plane parallel to the plate part and for a point at the plate part. By giving three coordinates to this reference point, the plate part will be defined in three dimensions.

    Bevel codes may be given along indicated parts of the plate part boundary contour.

It is possible to create plate part copies without answering all questions twice. To make a copy of the recently created plate part, just give the new name of the copy. If some information is to be changed in this copy, press **OPTION** when asking for name.

Prompt	**Operation/explanation**
Choose function	PLATE PARTS Enter the function.
Indicate contour and area	`<cursor position>` Collect geometry that describes the boundary of the plate.
Indicate contour and area	OPERATION COMPLETE All geometries collected.
Holes OK?	YES
Key in name of plate part:	`<name>`
Key in quality:	`<quality>` May be given as a code.
Key in position number:	`<position number>`
Key in thickness:	`<thickness>` Has to be given.
Key in build number:	`<build number>`
Key in production date:	`<date>` Format: `YYYY-MM-DD`.
Valid on side: 1=Port side 2=Starboard 3=Symmetric	Side where plate part is valid.
Define plane	A plane parallel to the plate part.
Define reference point Cursor Position	Indicate a 2D point in the plane.
3D point: Key in	Give the indicated point three coordinates.
Define end point Cursor position	Define bevel interval along the plate part boundary contour.
Key in bevel code:	Key in a bevel code for the defined interval.
Indicate marking line and material side	Indicate a 2D point close to a marking line.
Valid side for the marking line information: 1=Front 2=Back 3=Both	Valid side of the plate part marking line. Front is the side facing the user.

Prompt	Operation/explanation
Kind of component that causes the marking 1=Stiffener 2=Panel	Give the type of component that causes the marking.
Marking position number:	<position number> position number of the component that causes the marking.
Indicate contour and area	<cursor position> Collect geometry that Describes the boundary of the plate.
Indicate contour and area	OPERATION COMPLETE All geometries collected.
Holes OK?	YES
Key in name of plate part:	<name>
Key in quality:	<quality> May be given as a code.
Key in position number:	<position number>
Key in thickness:	<thickness> Has to be given.
Key in build number:	<build number>
Key in production date:	<date> Format: YYYY-MM-DD

- **Create Profiles**

- **Description**

Stiffeners to be manufactured are normally modelled via the Hull Modelling and stored together with the panel they are generated on. Panels are then split into parts to be manufactured and profiles are in this process placed on a data bank for profiles. Programs for manufacturing (profile nesting, profile sketches, etc.) are fetching information about profiles from this data bank.

There are situations in a shipyard, when there is no obvious need to create a model (ship repair, etc.) Still, the profile parts are to be manufactured in the same automatic way as the modelled parts. Manufacturing lists, link files for robots, profile sketches, etc. are to be produced with the same sequence of operations by the user as for profiles from the model.

The Profile Part Programming Tools as this function is called is intended to cover this need as described below.

- **Scope**

The tool contains functionality to edit already existing profiles on SBH_PROFDB as well as creating new profile objects. These new ones will then exist without references to any panel. Editing profiles includes the possibility to change the configuration of holes, notches,

cutouts, marking and signing information possessed by the profile. Already existing profiles can be deleted.

A profile is handled during a session. This session can be left unchanged in the back while performing other activities.

The main entrance to the tool is one of 3 possibilities:

- Create
- A new session will be initiated. This will give the possibility to create a new profile object. In case a profile already is current, you will be informed about this.
- Edit
- In case you have a session already active, this one will be resumed. Otherwise you will be shown the current contents of the profile data bank and asked for selection of one of these. The characteristics of this profile will now be displayed by the main form and you have a chance to edit the profile.
- Delete
- A selection box will appear with the current contents of the profile data bank. All selected profiles will be deleted after confirmation.

In case the profile is sufficiently defined, it will be displayed on the screen. The web of the profile will be displayed with end 1 to the left, stretching rightward. If the profile contains one or two flanges, these will be displayed in separate views, located above the view of the web. These will show the flange(s) as they appear from the outside of the profile (i.e. from above and below).

If Edit was chosen to view a profile, this view will be created upon the selection, otherwise pressing OC will try to draw the view.

This view (or views) of the profile will later on be used for highlighting the features of the profiles, if any. Please see section 2.

- **Input**

All information about a specific profile is maintained by a form as explained in section 1.6.

Since the profile will be displayed on the screen, a drawing must be current.

- **Naming and Storage of Profiles**

Naming of profiles can either be made freely by the user of in an automatic way. When using the automatic way of naming, the name will be built up as described below:

`<Proj><Block No.>-0<Auto. number>/S<rno><SB/PS information>`

`<Proj>`	Project ID from fetched form SB_PROJECT
`<Block No.>`	Block number fetched from the initial form of this function (see below).
–	Separator (minus sign).
`0`	Identifier for coded profiles.
`<Auto. number>`	Automatically generated number. The number is generated in a similar way as when using automatic naming in the module for Profile Nesting.
`/`	Separator (slash).

<rno>	Running number. Starting from one. In case of individual storage, equal profiles regarding parameters and geometry will be stored with increasing running number.
<SB/PS information>	P is used for portside profiles and S is used for starboard profiles.

Automatic naming is used when the logical SBH_PROFILE_NAMES is assigned to a file used for generation of automatic numbers.

When retrieving and storing profiles on the profile data bank, this can be done using a data type different from the one normally used for profiles. If this is done, also the Edit and Delete subfunctions will be restricted to this object type. The system manager will by this have a possibility to prevent editing or deletion of profiles generated from the model. If a data type different from the default is to be used (default is 90), this can be set by the use of a restriction file (see section 1.5).

The concept of individual storage is used when the following environment variables are set to something:

SBH_EXPL_PANEL_SYM

SBH_SPEC_SYMM_PARTS

- **Restriction File Usage**

The restriction file does not, as its name indicates, restrict any part of the functionality from being accessed. Instead, this is the means by which the system manager can tailor the tool by setting a few global options. The restriction file in itself is indicated by the environment variable

SB_PPPT_RESTRICT

and its contents can be used to rule the following:

- DATA_TYPE=<number>
- Sets the profile data type as used by these tools to <number>. Default is 90.
- ASSEMBLY_OPT=<YES/yes>
- Allows the user to neglect to specify assembly information in the main form.
- BLOCK_OPT=<YES/yes>
- Allows the user to neglect to specify block belonging in the main form.
- EXCESS_OPT=<YES/yes>
- Allows the user to neglect to specify excess information (for both ends) in the main form.
- BEVEL_OPT=<YES/yes>
- Allows the user to neglect to specify bevel information (for both ends) in the main form.
- POS_OPT=<YES/yes>

Allows the user to neglect to specify positions number in the main form.

Comment in the file are allowed in any of two ways, either if the line starts with a '#' or '!', or as anything that follows after a blank on each line. This means that all the possible options listed above **must be supplied as a consecutive word** (i.e. no internal spacing allowed).

The restriction file can contain any or all of these options in any internal order. In case any option is not specified or no restriction file exists, defaults will rule the user.

- **Main Input Form**

The main input form looks as follows:

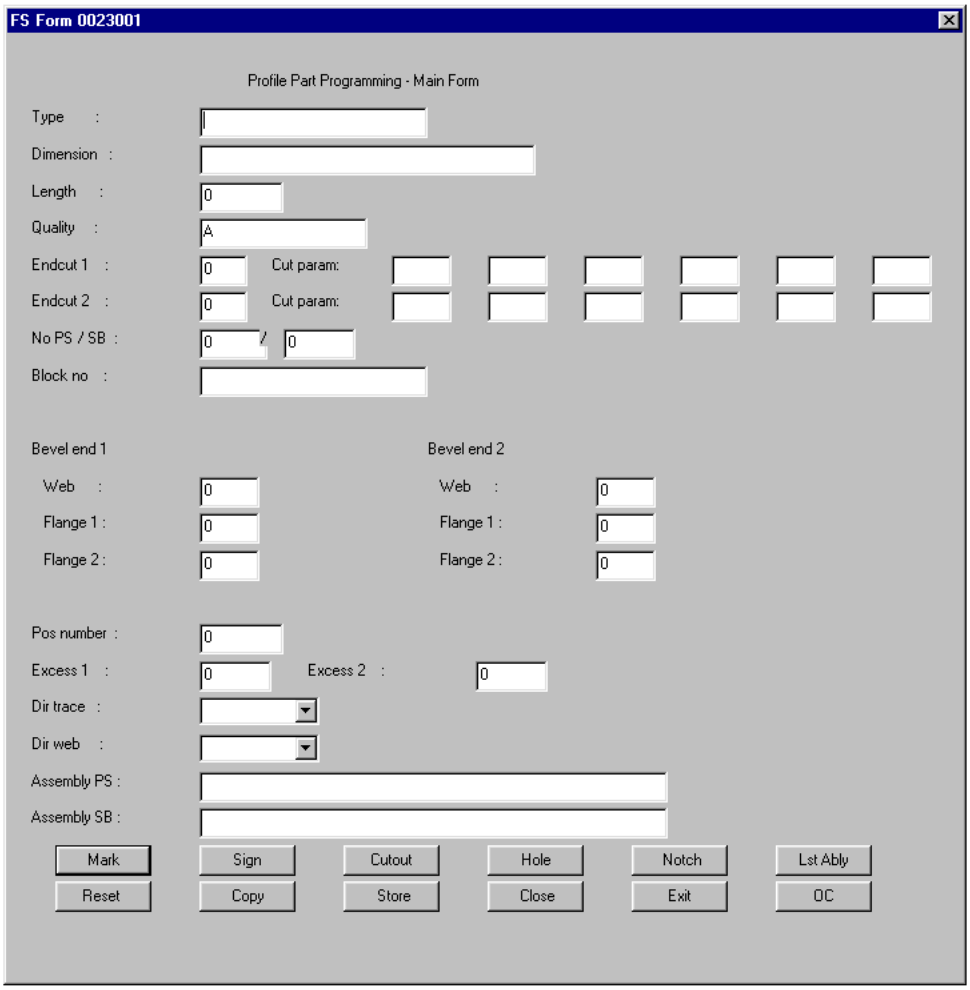


Figure 3:14. Profile Part Programming - Main Form.

where the key-in fields have the following meaning:

- Type

  Type string of profile (FB, HB, etc.).
- Dimension

  Dimension string (profile parameters separated by ',').

  If the Type and/or the Dimension field are empty, new forms with data from a file connected to SBH-PROF_CTRL will be displayed and the user is given the possibility to pick data from these values.
- Length

  Moulded length of the profile.
- Quality

  Quality string.

  If left empty, possible qualities are fetched from the quality control file (SBH_QUALITY_CTRL) and presented in a pick area of a form.

- Endcut <n>

  Endcut code for end <n>, possible with parameter set as according to the Standard.

  If the Endcut fields are empty, data from the endcut control file connected to SBH_ENDCUT_CTRL are displayed and made available for picking by the user. See *Setup and Customisation / Selection of Active Endcuts / _Endcut Control File* for further details.
- No PS/SB

  Number of portside respectively starboard specific profiles (all equal images).
- Block

  Block that the profile belongs to.
- Bevel end <n>

  Bevel code for web and eventually flange 1 and 2 for end <n> according to the Standard.
- Pos number

  Position number for the profile.
- Excess <n>

  Excess value for end <n>.
- Dir trace

  Direction of profile trace from end 1 to end 2 (Top, Bot, Aft, For, SB, PS).
- Dir web

  Direction of web (DOWN, UP).
- Assembly PS/SB

  If any of the assembly fields is empty, the user will have an option to pick an assembly to which the profile shall belong.

Note: that this is a one-way reference that not will be reflected in the assembly itself. The intention is purely to get assembly data to the part name definition via Part Name Control.

Furthermore, the following command buttons are available to select in the form:

- Sign

  View/edit the signing information for the current profile.
- Mark

  View/edit the marking information for the current profile.
- Hole

  View/edit the hole information for the current profile.
- Notch

  View/edit the notch information for the current profile.
- Cutout

  View/edit the cutout information for the current profile.
- List Asm

  Allows the user (for any of the assembly fields that are empty) to pick PS respectively SB assembly belongings among the currently defined. This function is automatically invoked when verifying that the profile properties are sufficiently specified.
- Reset

  The Reset button resets all values to an initial state.
- Copy

The Copy button displays a new form, which allows the user to key or pick the name of an existing profile. All data from this selected profile will be added to the form.

- Store

  The current contents of the form will be used to update the profile data bank. In case an already existing profile was used to fill the form (Edit), this object will be updated.

  In case a new profile has been defined (Create), automatic naming is selected if possible; otherwise the user will be prompted for a name.

  Individual storage is selected according to the criteria's listed.

  In any case, the form will be checked to contain all properties (although these are not validated in themselves).

- Close

  Close the current session. In case an already existing profile was used to fill the form, this will be unlocked on the data bank. The form will be resettled to an initial state. The displayed profile (all views of it) will be removed from the picture.

- Exit

  The Exit button leaves the subfunction without performing any further operations. The session is though still active in case you wish to continue afterwards.

- OC

  The OC button (operation Complete) is the positive answer to complete the subfunction. The profile will be created if all necessary values are supplied by the user and displayed on the screen. In case this is not possible, the user will be notified and guided through a number of new forms as required to fill in all necessary profile properties.

- **Profile Features**

  By the term profile features, all additional properties are included. This includes:

  - Holes
  - Notches
  - Cutouts
  - Marking information
  - Signing information

  Each of these feature types is supported by a separate type of form, in which its main characteristics are displayed. Along with this, the feature in question is highlighted in the drawing.

  Before the features of a specific type are displayed, the profile in the main form is checked for validation.

  In case the current form (of any feature type) does not represent any existing feature, the message field at the bottom of the form will indicate this.

  Common information for all types of features is:

  - Dist along

    Distance from end 1 to the origin of the feature. E.g. for cutouts this indicates the point of intersection, thereby the actual needed hole will depend on the intersecting profile's mould line etc.

  - Dist above

    If relevant, this will indicate the distance from the trace line to the origin of the feature.

  Common operations for all features are:

  - First / Next / Prev / Last

Retrieves the wanted feature of the specific type. If found, the form will display the feature specific data and the feature will be highlighted in the profile view.

Features of a specific type are sorted by increasing coordinate values along the trace line of the profile.

- Reset

  Resets all the current form fields. Doing this while editing an already stored feature might cause strange behaviour since all values will be set to 0 or empty strings, all while the feature still exists. Useful though when creating new features and wanting to get a fresh start.

- Copy

  Allows the user to pick an already existing profile from the profile data bank. After doing this, all features of the current type will be copied into the current profile.

- Delete

  Delete the current feature. The form will position itself on the next feature, if any. If that was the last feature on the specific type, the previous will be selected.

- Create

  A blank form will appear in which you can specify data for a new feature of the current type in question. After specifying the relevant data, pressing OC will reveal if the keyed in data was sufficient for the creation. If so, the feature will be sorted into the current list according to distance along trace line.

  In case the first field is left empty (thereby leaving out the geometry of the feature) when pressing OC afterwards, the standard drafting identification procedures are invoked to let user indicate geometry/text to form the feature.

  The user is thereafter asked to indicate on which surface (web or any of the flanges) that the feature should be positioned. This means that it will be possible to define holes, markings etc. on the flanges as well.

Note: Please note that AVEVA Marine in general does not support this yet, though.

- Exit

  Return to the main form.

- OC

  Used when creating brand new features. The contents of the current form are checked and the feature gets registered with the current profile.

- **Holes**

Used for traversing the holes within the profile.

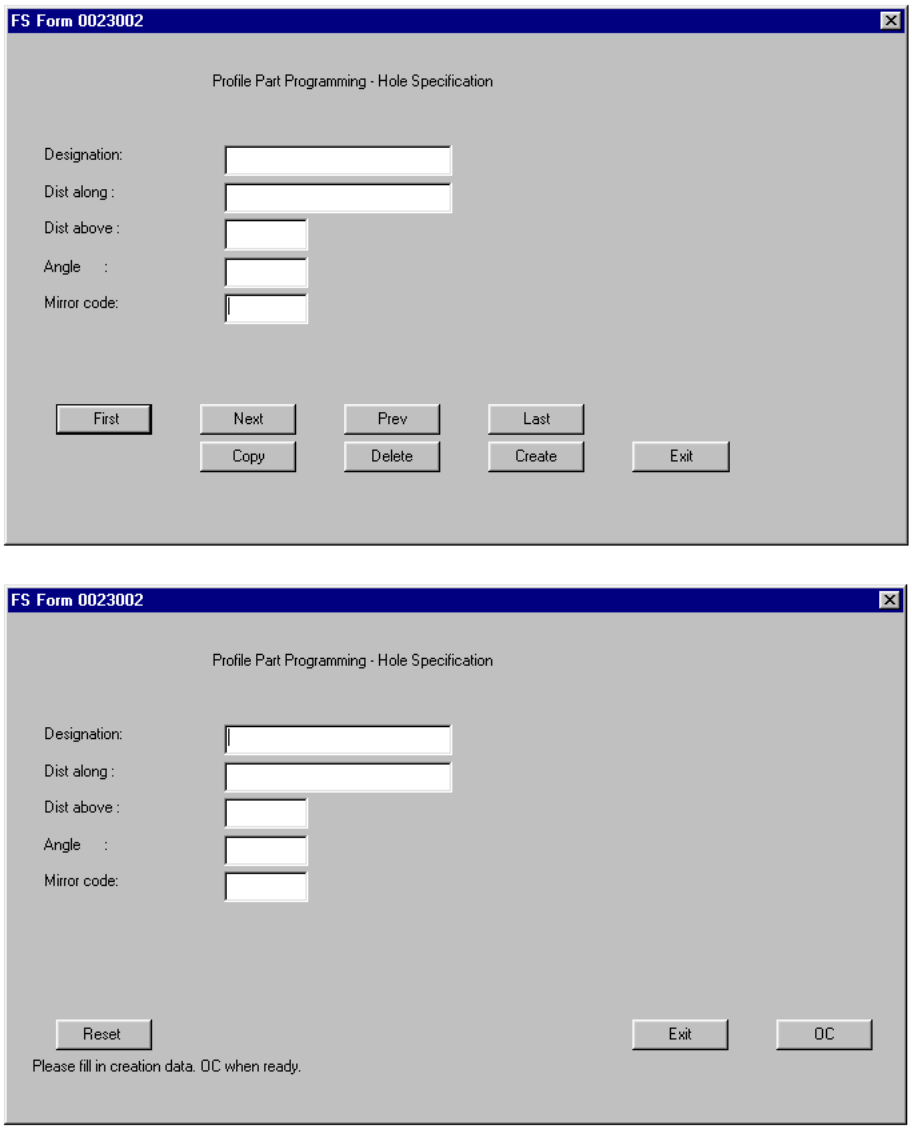


Figure 3:15. Profile Part Programming - Hole Specification.

Specification of the hole specific fields:

- Designation
  Designation of the hole, if any.
- Angle
  Rotation angle, if any, for the hole.
- Mirror code
  Indicates whether the hole should be reflected in the v-axis of the local coordinate system.

- **2.2. Notches**

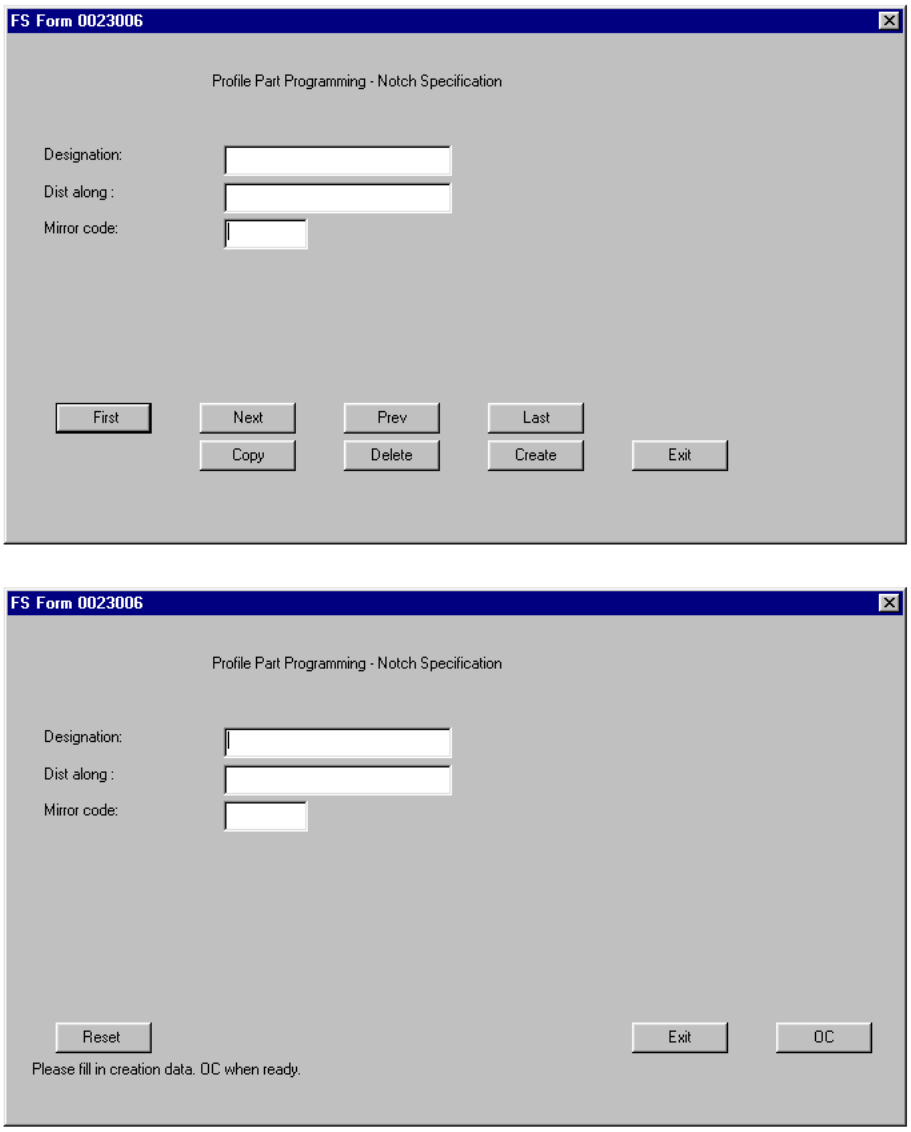


Figure 3:16. Profile Part Programming - Notch Specification.

Specification of the notch specific fields:

- Designation
  Designation of the notch, if any.
- Angle
  Rotation angle, if any, for the notch.
- Mirror code
  Indicates whether the notch should be reflected in the v-axis of the local coordinate system.

- **2.3. Cutouts**

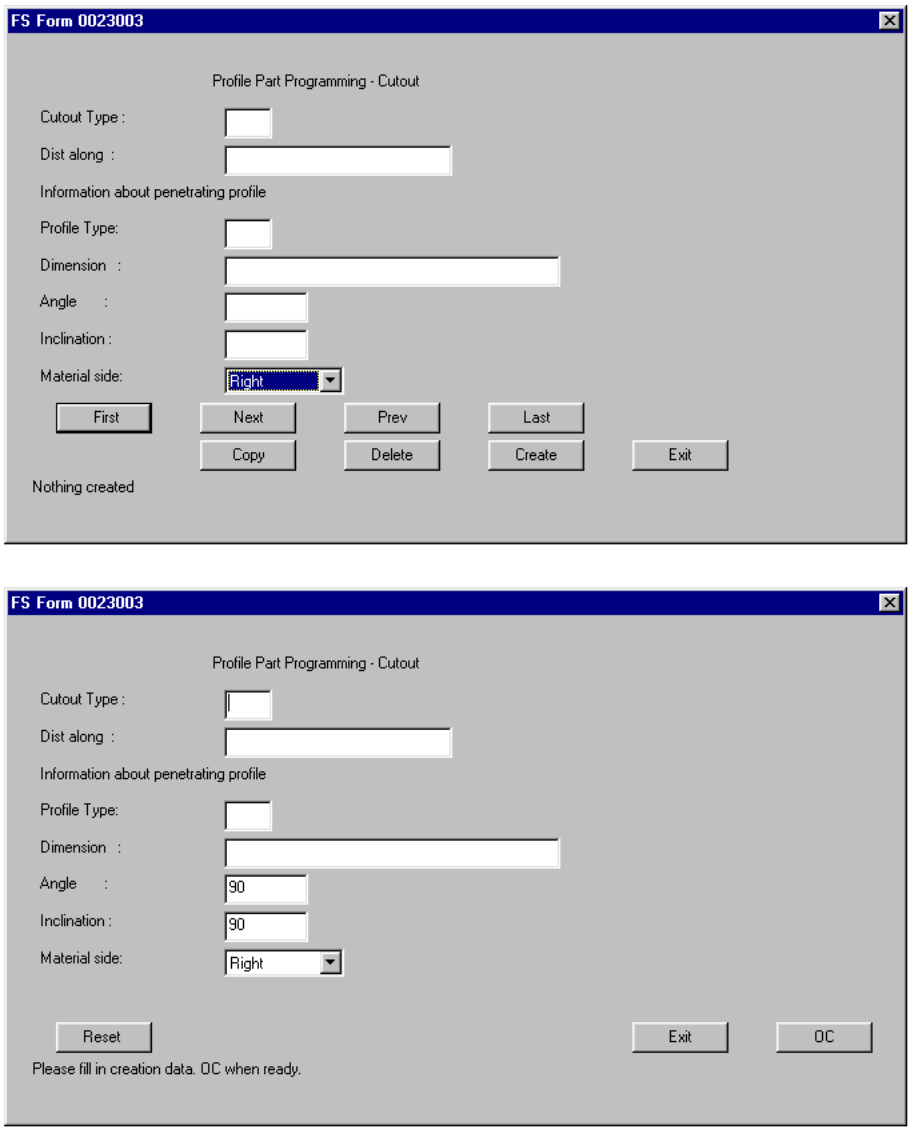


Figure 3:17. Profile Part Programming - Cutout.

Specification of the cutout specific fields:

- Type
  Profile type of the intersecting profile.
- Dimension
  Dimension of the intersecting profile. Provided in similar way as in the main form, i.e. profile parameters separated by ','.
- Angle
  Angle between the trace lines of the two intersecting profiles.
- Inclination
  The Angle that the intersecting profile deviates from the plane of the original profile's mould plane.

- **Marking**

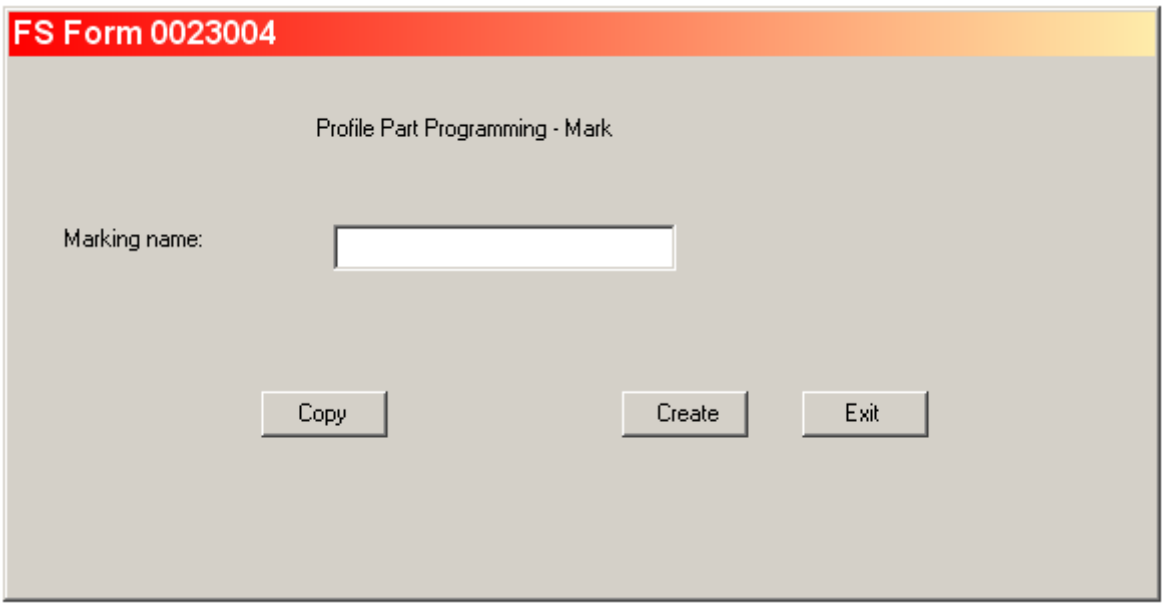


Figure 3:18. Profile Part Programming - Marking.

Specification of the marking specific fields:

- Designation
  Designation of the marking, if any.
- Angle
  Rotation angle, if any, for the marking.
- Mirror code
  Indicates whether the geometry of the marking should be reflected in the v-axis of the local coordinate system.

- **Signing**

Signing is used to place arbitrary text on the surface of the profile.

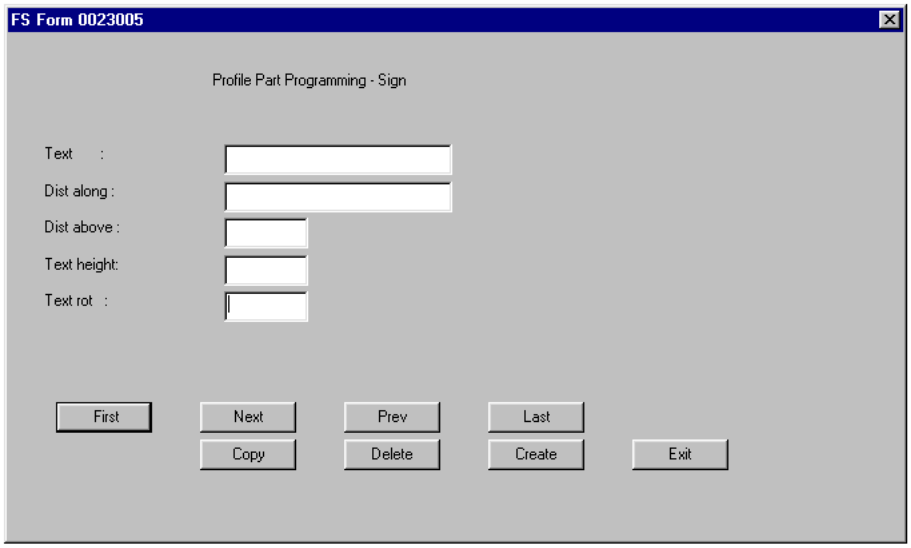


Figure 3:19. Profile Part Programming - Sign.

Specification of the signing specific fields:

- Text
  The textual information that is to be put on the profile.
- Text Height
  Text height.
- Text Rot
  Rotation angle of the text.

3.2.4 Shop Drawing Functions

The shop drawing functions are used to interactively create information on the shop drawing (burning sketch). The frame work of the shop drawing containing the tool path is generated during the verification (cf. *VERIFY*).

Automatic information in shop drawings

Specific production information such as bevel, excess, side etc. can be added automatically to the burning sketch by applying a form to the sketch. The forms are in the same format as the standard drawing forms and are stored in the same data bank. To create, edit and add a form the Drafting functions for these tasks are used.

However there is a set of form rules that is nesting specific and will only work when a form is activated in nesting. For further information about how to create and use forms in the Nesting application see *Initialisations for Nesting*.

- **PRODUCTION INFORMATION**

  This function is used to add production information to the shop drawing. The various items that can be added are the same as in the function SHOW PRODUCTION INFORMATION except for the block name, the ship number, the spitting date and the production date. These items can only be used in the function SHOW PRODUCTION INFORMATION.

When the desired item has been chosen, the corresponding text can be dragged into the proper position. The text height and the text rotation angle will be the modal values (cf. *TEXT*).

- **DISPLAY NESTED PLATE**

  This function is used when the work with the shop drawing is finished. After actuation, the nested plate will be available on the screen.

- **DISPLAY SKETCH**

  This function is used when the shop drawing is to be displayed on the screen instead of the nested plate. An automatic autoscale is performed.

- **FIRST/NEXT/PREVIOUS/LAST SKETCH**

  If the nesting drawing form contains part tables it is possible that the number of parts on the nested plate exceeds the number of part tables in the form. In this case, these functions can be used to display the different 'pages' of the burning sketch.

- **DISPLAY PLATE/SHOW ALL PLATES**

  This function is available only when multiple nesting have been activated by reading multiple nest jobs into workspace. It is used to switch between the multiple nesting mode and the normal mode of showing a selected nest job in standard plate nesting mode.

- **DETAILED SKETCH**

  A detailed sketch can be added to the form for small parts. To enable this functionality the following default settings has to be applied..

DETAILED_SKETCH	=	YES NO	Enabled Disabled
DETAILED_SKETCH_AREA	=	Parts smaller than given area will be drawn in a detailed sketch.	
DETAILED_SKETCH_SCALE	=	Scale of detailed sketch, given as a scale relative to sketch.	

  Along with these settings sequence numbers must be added.

AUTO_CUTSEQ_INFO	=	YES

- **TEXT**

  This function is used to define and place a text, to move or copy a text, to define the modal text height, text rotation angle or text font, to place the start sequence or to place the plate name. It is also possible to change the text height, the text angle, the contents or the text font of an existing text.

  A text to be defined is keyed in and dragged into the proper position.

  The modal text height can be defined by keying in the height, by indicating an existing text, the height of which shall be modal, by giving cursor positions at the bottom and the top of

the text or by referring to a standard height. The modal text rotation angle is keyed in. The modal text font is keyed in.

The start sequence can be placed in one operation.

The plate name can be placed in the same way as any text that has been keyed in.

Choosing REJECT when a text is to be placed allows for a change of height and/or rotation angle. The standard height in the system is 20.

- **PUT SYMBOL**

  This function is used to place a symbol. Upon actuation of the function, a symbol menu is displayed a the top of the graphic display. A symbol is chosen by indicating it with the cursor, and it can then be dragged into the proper position. The function can also be used to define the symbol height or symbol rotation angle in a way corresponding to the text heights and rotation angle. It is also possible to mirror the symbol in the u-axis or the v-axis.

  The UNDO facility is supported.

- **MOVE SYMBOL**

  This function is used to move an existing symbol. A symbol is identified in the burning sketch with the cursor, and can then be dragged into the proper position. The possibilities to change symbol height, symbol rotation angle and to mirror the symbol, are also available in this function.

- **POSITION NUMBER**

  This function is used to place position numbers in the burning sketch. It is possible to create position numbers, to change the reference symbol, to change the position number symbol and to re-create the reference line for an existing position number.

  When a position number is to be placed in the sketch, the operator identifies the item (plate part or marking line) which position number is to be placed. It is then possible to create a string line from this position by giving successive cursor positions. By using REJECT at any stage, it is possible to delete the last reference line segment. OPERATION COMPLETE will finish the string line creation. If a position number is defined for the item, it will be displayed at the end of the last reference line segment and automatically be surrounded by the position number symbol. A reference symbol is also automatically added at the beginning of the string line. If no position number is available, the operator is prompted to give it.

  In the case above, the position number symbol must be defined with four connection points (see *User's Guide Drafting*). If the symbol has been defined with less than four connection points the position number symbol will be placed at the last reference line segment but to the left and at a distance above it. This distance is controlled by a default value.

  The UNDO facility is supported.

- **AUTOMATIC POSITION NUMBERS**

  In this function position numbers can automatically be added to the burning sketch. The function is activated when the burning sketch form is added (cf. *Add in Chapter Operator's Instructions - Nesting*) and is controlled with the default parameter AUTO_POSNO.

  The position numbers are put on all marking lines existing in the burning sketch at the relative distance AUTO_POSNO_DIST from the starting point of the marking line. For the plate part itself, either the position number (= short part name) or the full part name is put in

the Centre of Gravity of the part. The type is given by the default parameter AUTO_POSNO_PART. No reference lines are used for the position numbers.

If necessary, the position numbers can be moved with the POSITION NUMBER function. The reference line will then be re-defined.

- **DELETE SKETCH**

  This function is used to delete information from the shop drawing. A single item or all items of a kind can be deleted. The information of the nested plate (parent plate, plate parts, starts, burning bridges, restplate texts (special steel quality only), position numbers, measures etc.) can also be deleted.

3.2.5 Nest PPI

- **CREATE STANDARD PLATES**

  This function is used to create standard parent plates. The standard parent plate data is supposed to exist in a text file which is selected by the user.

  The following information about a plate must be given in the file:

plate name	(max 24 characters)
length	(real)
width	(real)
thickness	(real)
quality	(max 25 characters)
density	(real)
charge	(max 75 characters)
preparation	(max 25 characters)
purchase information	(max 25 characters)

  All terms must be given on one line and be separated by a blank or a comma. Strings must be surrounded by ' characters (e.g. 'A10' for quality). If a term has been assigned no value, then integers should be given as 0, decimals as 0.0 and strings as ''. There is no restriction on how many plates that can be defined in the text file.

  Plates to be used in the restplate handling must have negative length and width.

Below is an example of such an input file.

Example:

```
'STD001'   8.00E+03   2.50E+03   4.00E+01 'D  '   8.00000000E-06 'TEST' '' ''
'STD002'   8.00E+03   2.50E+03   4.50E+01 'D  '   8.00000000E-06 'TEST' '' ''
'STD003'   1.25E+04   2.95E+03   2.00E+01 'D  '   7.99999952E-06 'TEST' '' ''
'STD004'   1.25E+04   2.95E+03   2.20E+01 'D  '   8.00000000E-06 'TEST' ''
'STD005'   8.00E+03   2.95E+03   2.00E+01 'A  '   8.00000000E-06 '' '' ''
'STD006'   7.50E+03   2.50E+03   1.10E+01 'A  '   8.00000000E-06 '' '' ''
'STD007'   8.75E+03   2.95E+03   1.55E+01 'A  '   7.99812508E-06 '' '' ''
'STD010'   4.00E+03   2.00E+03   1.20E+02 'D  '   8.00000000E-06 '' '' ''
'RESTPLATE'  -1.00E+03  -1.00E+03   1.20E+02 'D  '   8.00000000E-06 '' '' ''
```

- **Generic Post Processor**

  This function is used to run the Generic Post Processor for a number of generic files. The files are selected by the user.

  For a complete description of the Generic Post Processors see also *Hull Post Processors.*

- **Batch Nesting**

  This menu alternative is used to activate different batch nesting functions. When activated, a menu is displayed with the following entries:

  - Select
  - Show
  - One Part
  - 3-Axis
  - Plug-in

  The *Select* alternative will invoke the standard selection tool for production data. It is further described in *Hull - Hull Manufacturing - Hull Production Program Interface - General Selection Tool*.

  The *Show* alternative is used when inspection of shall be made of resulting sketches and drawings. It is further explained in Hull - Hull Manufacturing - Hull Production Program Interface - Presentation of Graphical Results.

  The other entries in this menu, One Part, 3-Axis and Plug-in all invoke a batch nesting routine. One Part and Plug-in have common user interaction concerning how names of resulting nestings shall be made and how raw material shall be selected. In 3-axis nesting the system extracts the developed parts according to the selection made. For each quality-thickness combination the user is then prompted to select the part(s) to be nested and the raw plate to be used.

  When creating names, the following constitutes can be given:

  - Prefix
  - Suffix
  - Base name
  - Starting number
  - Number of digits

  which will be used in the following way:

  `<prefix><Base Name><Number><suffix>.`

If starting number is 1 and number of digits is 3, the first nesting will use 001 as its number.

Raw material is selected in the same way as when selecting it during interactive nesting. Any number of materials can be given. Execution starts when leaving this menu by pressing Cancel.

The One Part Nesting is further described in *One-Part Nesting*.

The 3-Axis Nesting is further described in *Manufacturing Data for 3-axis Burning Equipment*.

The Plug-In Nesting is further described in *Nesting Plug-In*.

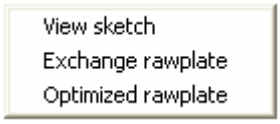
3.3 Right Click

A quick way to perform certain actions on objects in the nesting or in the burning sketch is by indicating a nesting component and use the right-click. The context menu alternative depends on which object type the user has clicked on.

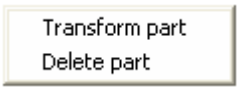
3.3.1 Nesting

The context menus are available for nested parts, bridges, starts, auxiliary functions and labelled texts. By right-clicking on an object in the nest a pop-up menu is displayed showing the alternatives that can be performed

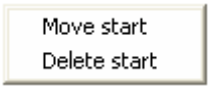
.



Menu viewed for click on the raw plate.



Menu viewed for click on a Part.

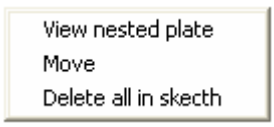


Menu viewed for click on a start.

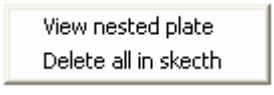
3.3.2 Burning sketch

The context menus are available for text/symbol and production information. By right-clicking on an object in the burning sketch a pop-up menu is displayed showing the alternatives that can be performed.

Menu viewed for click on the drawing form.



Menu viewed for click on production information.



Menu viewed for click on the raw plate.

4 One-Part Nesting

4.1 General

Nesting jobs, each consisting of one single plate part, may be nested and output data for the workshop created automatically.

The program will combine a selection of plate parts on a selection of standard plates. The nesting function will place the parts on the smallest possible raw plate.

The selection of plate parts is made in the standard selection form of the Production Program Interface of the Hull system. It is further explained in Manufacturing, General About the Production Program Interface.

The names of the produced nestings are created automatically based on the input from the user. This is further explained in Manufacturing, AVEVA Hull Hull Nesting, Hull Plate Nesting, Nesting System Application Functions, Functional Overview, NestPPI.

4.2 Set-up of Program

The name of the executable of this program is se016. It communicates via an input file and resulting files. The program is normally activated through the Job Launcher (JL) where the following set-up is required:

Name recognised by JL: **Nesting One-Part**

AVEVA Marine logical	JL set-up and explanation
SB_INPUT1	Input file to be set up with extension .dat in JL
SB_OUTPUT1	Output file with run-time information. To be set up in JL as first output file with extension .log

4.3 Input

Input to the program is based on the general selection tool and this input is normally generated automatically when activating this function through the interactive nesting application via a production program interface. The selection possibilities are described in *Manufacturing, Hull Production Program Interface, General Selection Tool*.

4.4 Output

The program will make complete nesting jobs, generating NC data and a Burning Sketch.

The NC data file, by default placed on the directory assigned to the logical SB_TAPE, will be presented in the output window of the JL.

The Burning Sketch is stored in the same way as for any other nesting object together with the actual nesting. By executing this software via the nestPPI menu of Interactive Nesting, the possibility to view and plot the sketches via a show function is available.

5 Nesting Plug-In

5.1 Plug-In Interface for Automatic Nesting Algorithms

5.1.1 General

The Automatic Nesting Plug-In makes it possible to connect the VM Nesting System to an external pattern generator. Using this function part data are transferred to the pattern generator which returns transformation data for each part for proper positioning on the raw plate. The Plug-In is part of the Nesting System and can be used in either interactive or batch mode.

The VM Automatic Nesting Plug-In makes it possible to connect the Nesting System to an external pattern generator. Using this function part data are transferred to the pattern generator which returns transformation data for each part for proper positioning on the raw plate. The Plug-In is part of the VM Nesting System and can be used in either interactive or batch mode.

- **Data Transfer**

The Plug-In is supplied with the following data from the Nesting System:

- Plate part geometries (the bump contour, i.e. the maximum extension of contour in case bevel cutting)
- Plate part quality, thickness, assembly, production date etc.
- Raw plate sizes and quantities
- Nesting defaults (e.g. reflection flags, rotation increments, part-to-part/plate distances)

The Plug-In application will then return the following information to the Nesting System:

- Co-ordinates of the plate part origin
- Rotation angle
- Reflection flag

5.1.2 Input

- **Interactive Mode**

In interactive mode the user initialises a nest job and prepares a parts menu in the normal way. The menu function Automatic nesting will then transfer the necessary data to the Plug-In. After optimising the nesting pattern using its own algorithm, the Plug-In returns the transformation data for the nested parts to the Nesting System. The nested parts are presented on the screen and the user can continue processing the nested plate. This processing may include modification of the layout, nesting of additional parts, either manually or using Quicknesting. The standard nesting functions are then available to set

bridges, add labelling information, create starts and generate the tool path. The nesting sketch and generic file may be created as in any nesting.

- **Batch mode**

  In batch mode an input file is used to define the plate parts to be nested and the raw plates to be used. In this mode the user has the option to let the VM Nesting automatically set starts, define the tool path, create the burning sketch, generate a plot of the sketch and create the generic file, see also Robotics, User's Guide, Section III, Chapter 2. The output is the same as in the interactive mode and the nesting can be changed interactively if necessary.

- **Language**

  The definition of the input language can be found in Robotics, Section III, Chapter 1. In addition the following keywords have been added:

  Statements/attributes Usage

RAW_PLATE	Use this statement to define the available raw plates. Wild cards can be used.
QUANTITY	Attribute to the RAW_PLATE statement. It must be used when the quantity of the raw plates is larger than 1.

  The name of the syntax file is `d0320174.sbx` and the corresponding message file is `d0300174.sbm`.

Example:

The following input will select all plate parts within the given block and nest them on the specified raw plate. If the number of raw plates is insufficient or if the size is too small some plate parts will not be nested. The reason for not nesting some plate parts will be specified in the log-file for each part.

```
RAW_PLATE, 'NORM1'/QUANTITY=10;

RAW_PLATE, 'NORM2'/QUANTITY=10;

RAW_PLATE, 'NORM3'/QUANTITY=5;

RAW_PLATE, 'NORM4'/QUANTITY=10;

RAW_PLATE, 'SPEC1';

BLOCK, 'ES123';
```

5.1.3 Environment

- **Variables**

  The following environment variables should be defined at run time:

Name	Description
SB_PLDB	Name of the plate part data bank.
SB_OGDB	Name of the hull structure data bank.
SB_ASSDB	Name of the assembly data bank.
SBH_QUALITY_LIST	Name of the quality list file.
SBH_QUALITY_EXCH	Name of the quality exchange file.
SBH_NEST_DEF1	Names of the nesting default files.
SBH_NEST_DEF2	
SB_DEFSTD	Name of the nesting standards data bank.
SBH_AUTONEST_PLUGIN	Name of the Plug-In to be connected.
SB_DEFNPL	Name of the nesting data bank.
SB_TAPE	Name of the directory for the generic files
SB_PLOT	Name of the directory for the plots.

Furthermore, general environment variables as SB_SHIPDATA and SB_SHIPPRINT are used. The databank associated with SB_OGDB should contain object for bevel (__SBH_BEVEL_CTRL__) and part name control (SBH_PARTNAME_CTRL).

5.1.4 Plug-In Interface

This section describes the interface through which the Plate Nesting can communicate with external software plug-in that makes the layout of a nested plate. It consists of a number of C++ routines for the transfer of data between Nesting and the external layout generator and also a definition of a Microsoft Visual Studio project for building the plug-in dll.

See also the document Autonestdoc.pdf.

5.2 Plug-In Interface for Automatic Nesting Sequence Algorithms

5.2.1 General

The Automatic Nesting Sequence Plug-In makes it possible to connect the Nesting System to an external sequence generator. Using this function part data are transferred to the sequence generator which returns sequence data for each start for the calculated starting sequence The Plug-In is part of the Nesting System and can be used in either interactive or batch mode.

- **Data Transfer**

  The Plug-In is supplied with the following data from the Nesting System:
  - Nesting name
  - Number of individual parts

- Raw plate size and quality
- Nesting defaults (e.g. cutting direction, default starting points etc.).
- Plate part geometries as defined on nested plate
- Bevel information

The Plug-In application will then return the following information to the Nesting System:

- Number and types of starts
- Positions for each start
- Sequence information

5.2.2 Input

- **Interactive Mode**

In interactive mode the user initialises a nest job and fetches a nesting with a ready layout. The menu function Sequence nesting will then transfer the necessary data to the Plug-In, it will sequence and put the starts for the current nesting. After optimising the starting sequence using its own algorithm, the Plug-In returns data for the nested plate to the Nesting System. The nested plate are presented on the screen with the starts placed and sequenced and the user can continue processing the nested plate. This processing can include modification of the starts or making the production data for the plate by verifying the tool path, creating the sketch and the generic output and if connected the NC-output. The standard nesting functions are available after running the sequence plugin, above a common working procedure are described.

5.2.3 Environment

- **Variables**

The following logical/environment variables should be defined at run time:

SBH_NEST_DEF1 SBH_NEST_DEF2	Names of the nesting default files.
SB_DEFNPL	Name of the nestings data bank.

5.2.4 Plug-In Interface

Below follows the definition of the Plug-In Interface. It consists of a number of C++ routines for the transfer of data between Nesting and the external sequence generator.

See also the document Sequenseplugindoc.pdf.

6 Initialisations for Nesting

6.1 Defaults

Default information (defaults) consists of system parameters and default values controlling the overall performance of the system, giving the users the possibility to adapt the system to different working conditions and different workshop cutting procedures.

The default system stores the default information in default files identified by their file names (device name, name of directory and name of file). The length of a file name should not exceed 75 characters. The default information is stored as one assignment statement per row in the default file. An assignment statement consists of a keyword identifying the variable followed by an equal sign, followed by the default value. Since the default information is identified by a keyword, the order of the default information in the default file is irrelevant.

The default system is divided into two levels with one default file each. The higher, superior level (the system manager level) consists of global default variables that may not be changed by the operator. The inferior level consists of default variables that may be changed by the operator.

The superior default file must be assigned to the logical variable SBH_NEST_DEF1 and the inferior default file to the logical variable SBH_NEST_DEF2.

The keywords of the default variables are given below in alphabetic order.

ACCURACY_OF_RAW_PLATE_LENGTH	Accuracy in mm used when optimising the length of the rawplate. 0 means that no rounding will be applied to the length.
ACCURACY_OF_RAW_PLATE_WIDTH	Accuracy in mm used when optimising the width of the rawplate. 0 means that no rounding will be applied to the width.
ACT_AFTER_LABELLING	Code for activity after labelling: **0 =** Continue with idle movement to next start. **1 =** Make an idle movement to an explicitly defined position.
ADDITIONAL_EXCESS	The values of the additional excess in the u- and v-direction, respectively, which are applied for some plate parts.

ALT_LONG_NAME	Option of how to present longitudinal name in burning sketch.
ALT_TRANS_NAME	Option of how to present transversal name in burning sketch. **Yes =** <prefix><long number> <trans number> **No =** Use longitudinal part name transversal
ANGLE_CHECK	This code controls if the angle between consecutive segments shall be checked in the VERIFY function. With this check it is possible to detect direction changes that are too large. **0 =** No check made. **1 =** Check the angle.
ATTACHMENTINFO_COLOUR	Colour of attachment information.
ANG_CONSEC_SEG	The smallest angle between consecutive segments which is accepted in VERIFY when the angle is checked.
AUTO_GRINDING_INFO	Code for adding grinding information automatically in the Nesting burning sketch: **No =** Not automatic. **Yes =** Automatic
AUTONEST_CLUSTER	This code controls the clustering in the Automatic nesting function. **No =** No clustering **Yes =** Clustering
AUTONEST_FILL_TYPE	Fill type of automatic nesting: **1 =** Normal, fill exclusively with parts as defined by the parameter AUTONEST_PART_SET. **2 =** Add also parts from another set (same quality and thickness but different assembly or production date).
AUTONEST_FORMNAME	The name of the nesting form to be used automatically when a for is inserted. The user will be prompted to give the formname if no name has been given.
AUTONEST_IN_HOLE	Code for automatic nesting of parts in holes:

	No = Do not use holes.
	Yes = Use holes.
AUTONEST_LAYOUT_ONLY	Code for sketch creation in automatic batch nesting:
	No = Create an automatic sketch after the layout.
	Yes = Stop after the layout.
AUTONEST_MAXSCRAP_MIRROR	Maximum allowed scrap for mirrored single part nesting.
AUTONEST_MAXSCRAP_MULTI	Maximum allowed scrap for multi part nesting.
AUTONEST_MAXSCRAP_SINGLE	Maximum allowed scrap for single part nesting.
AUTONEST_NESTNAME_PREFIX	The prefix of the automatically created nestings in batch. A running number will be added automatically.
AUTONEST_OPT_TYPE	Code for optimisation type to be used in automatic batch nesting:
	0 = Do not optimise.
	1 = Press down all parts towards the lower left corner.
AUTONEST_PART_ANGLE	The rotation angle increment of the autonested parts.
AUTONEST_PART_SET	Code for the part set used in automatic batch nesting:
	1 = Normal, based on part quality and thickness.
	2 = Based on part assembly.
	3 = Based on part production date.
	4 = Based on part assembly and production date.
AUTOSCALE_SKETCH	This code controls how the drawing sketch should be scaled.
	0 = Half automatic scaling. A scale is suggested, the operator is prompted to decide the scale and to place the shop drawing in the form.

	1 = Automatic scaling. The scaling of the shop drawing and placing of it is made automatically. Shop drawing is scaled to best fit the drawing area.
	2 = Automatic scaling, same as for 1, but scale is adjusted to be even, e.g. 1:10, 1:20 etc.
AUTO_ATTACHMENT_INFO	Automatic attachment information in burning sketch
	= Yes
	= No
AUTO_BEVEL_INFO	Code for adding bevel information automatically in the Nesting burning sketch:
	No = Not automatic.
	Yes = Automatic.
AUTO_COMPENSATION_INFO	Automatic compensation information in burning sketch.
	= Yes
	= No
AUTO_CUTSEQ_INFO	Automatic cutting sequence information in burning sketch also required to get detailed sketches.
	= Yes
	= No
AUTO_EXCESS_INFO	Code for adding excess information automatically in the Nesting burning sketch:
	No = Not automatic.
	Yes = Automatic.
	The length of the bevel limit tag used for Continuously Varying Bevel Angles.
AUTO_GEN_COUNR	This code controls how the counting number for restplates is to be given.
	No = Key in.
	Yes = Automatically.
AUTO_HULL_MARK_INFO	Code for adding names of Hull Marks automatically in the Nesting burning sketch:

	No = Not automatic.
	Yes = Automatic.
AUTO_NEST_NAME	Code for automatic naming of nested plates:
	No = Key in.
	Yes = Automatic.
AUTO_POSNO	This code controls the usage of automatic position numbering in the nesting burning sketch.
	No = No automatic position numbering.
	Yes = Automatic position numbering.
AUTO_POSNO_DIST	The relative distance from the marking line start where the position number symbols and text is placed.
AUTO_POSNO_PART	This code controls the type of position number to be put for the plate part.
	0 = The long part name.
	1 = The position number.
AUTO_POSNO_POSITION	Parameters to control how much the origin of the position number symbol shall be moved from COG. The values (du & dv) are given as factors of the position number height (0.5 means half the position number symbol height).
AUTO_RECT_PART_MEASURE	Control for automatic dimensioning of rectangular parts.
	No = Not automatic.
	Yes = Automatic.
AUTO_SIDE_INFO	Code for adding side information automatically in the Nesting burning sketch:
	No = Not automatic.
	Yes = Automatic.
AUX_COLOUR	The colour for auxiliary functions.
AUTO_VERT_MARK_MEASURE	Control for dimensioning of first vertical marking.
	No = Not automatic.
	Yes = Automatic.

AUX_FUNC_RANGE	Auxiliary functions range. This value allows the creation of auxiliary functions at a start, that has been created at an arbitrary point on a contour. When a start is processed, an auxiliary function is considered to belong to the start provided the distance between the two items is less than the default range.
AUX_COLOUR	The colour for auxiliary functions.
BEVELINFO_COLOUR	The colour of the bevel information in the burning sketch.
BEVELINFO_LINTYPE	Linetype For bevel information.
BEVEL_ARROW_SYMBOL	Symbol number for bevel arrow. 0 indicates that no arrow is wanted.
BEVEL_LIMIT_TAG_LENGTH	The length of the tags indicating the limits of a bevel interval (CVBA only).
BEV_OS_COLOUR	Colour for those parts of the plate part contour which have bevel defined on the other side.
BEV_TS_COLOUR	Colour for those parts of the plate part contour which have bevel defined on this side.
BRIDGE_COLOUR	The colour for bridges.
BRIDGE_NODE	Node point preference for bridge creation. Very often, a bridge is wanted between the corners of two plate parts, e.g. at a common cut. However, it is difficult to exactly indicate the corner when the bridge is created. This default value controls whether the closest node should be preferred to the closest foot point of a segment when a cursor position indicates the position for a bridge. The code can control the following activities: **< -1** Closest foot point is always chosen. **-1 to 0** Closest node is always chosen **> 0** Closest node is chosen provided the distance from the given cursor position to the node is less than the default value, else closest foot point is chosen.
BRIDGE_RADIUS	The radius to be used in bridges to avoid sharp corners. If 0, no radius will be used.

BRIDGE_WIDTH	Width of bridge. This is the default bridge width.
BRIDGE_WIDTH_HOLE	The bridge width in holes.
BRIDGE_YDIR	Bridge direction code. This code controls if forced vertical y-direction is wanted or not, when creating a bridge. **0 =** Not wanted **1 =** Wanted
BURN_BEVEL_I	Code for burning I-bevel: **No =** Not supported **Fixed =** Fixed value
BURN_BEVEL_MIN_SEG_LENGTH	Minimum length of segment to be included in the bevel interval statistics.
BURN_BEVEL_X	Code for burning X-bevel: **No =** Not supported **Fixed =** Fixed value **VBA =** Varying Bevel Angle **CVBA =** Continuously Varying Bevel Angle.
BURN_LEVEL_Y	Code for burning Y-bevel: **No =** Not supported. **Fixed =** Fixed value. **VBA =** Varying Bevel Angle. **CVBA =** Continuously Varying Bevel Angle. **Fixed (V only) =** Fixed value, only V-bevel. **VBA (V only) =** Varying Bevel Angle, only V-bevel. **CVBA (V only) =** Continuously Varying Bevel Angle, only V-bevel.
BURN_BEVEL_X_MAX_ANGLE_OS	Maximum supported angle for Y-bevel, other side.
BURN_BEVEL_X_MAX_ANGLE_TS	Maximum supported angle for Y-bevel, this side.
BURN_BEVEL_X_MIN_ANGLE_OS	Minimum supported angle for Y-bevel, other side.

BURN_BEVEL_X_MIN_ANGLE_TS	Minimum supported angle for Y-bevel, this side.
BURN_BEVEL_Y	Code for burning Y-bevel:
	No = Not supported.
	Fixed = Fixed value.
	VBA = Varying Bevel Angle.
	CVBA = Continuously Varying Bevel Angle.
BURN_BEVEL_Y_MAX_ANGLE_OS	Maximum supported angle for Y-bevel, other side.
BURN_BEVEL_Y_MAX_ANGLE_TS	Maximum supported angle for Y-bevel, this side.
BURN_BEVEL_Y_MIN_ANGLE_OS	Minimum supported angle for Y-bevel, other side.
BURN_BEVEL_Y_MIN_ANGLE_TS	Minimum supported angle for Y-bevel, this side.
BURN_NO_BEVEL_TEXT	Text automatically placed in the burning sketch when the bevel is not supported.
CF_RESTART	Gap restart code for gap including cut free geometry (cf. GAP_RESTART).
CHANGE_SHIP_NR	Defines if the ship number is changeable by the user or not.
	No = Ship number not changeable
	Yes = Ship number changeable
CHANGE_OPERATOR	Defines if the operator name is changeable by the user or not.
	No = Operator name not changeable
	Yes = Operator name changeable
CLIP_GRINDING_SIDE	Defines how the grinding side of the clip should be displayed. Possible values are UP, DOWN or NONE.
	UP = The clip will automatically be mirrored if the clip is located on the non-moulded side of the panel.
	DOWN = The clip will automatically be mirrored if the clip is located on the moulded side of the panel.

	NONE =	The clip will be displayed as it is defined.
COLOURS_IN_SKETCH	**Yes =**	Use colour settings in sketch. Allows selection of colours for different type of actions.
	No =	No colours.
COMPENSATIONINFO_COLOUR	Colour for compensation. information in burning sketch.	
COMPENSATIONINFO_SUFFIX	Suffix for compensation information in burning sketch.	
CONT_DIRECTION	This code controls the cutting direction in outer contours.	
	0 =	No check made.
	1 =	Forced cutting counter-clockwise.
	2 =	Forced cutting clockwise.
CORNER_ANGLE	The least knuckle angle that will be regarded as a corner.	
CORN_LOOP_CTRL	Code for automatic creation of corner loops:	
	-2 =	No corner loops are created.
	-1 =	As 0 but no corner loops are created inside cutouts.
	0 =	All corner loops are created.
	1 =	Corner loops are created only in corners where there is change in bevel.
	= 10	PLCM, as 0 but loops will be deleted when burning starts are created.
	= 11	PLCM. as 1 but loops will be deleted when burning starts are created.
	= 20	PLCM, as 0 but the corner loops will be symmetric with fixed length.
	= 21	PLCM, as 1 but the corner loos will be symmetric with fixed length.
CORN_LOOP_LENGTH	Corner loop length.	

CORN_LOOP_MAXNO_AUXFUNC	The number of allowed auxiliary functions in the corner loop points P1, P2 and P3, respectively. -1 means an unlimited number.	
CORN_LOOP_MINANGLE	The minimum angle for a knuckle between two segments to be treated as a corner.	
CORN_LOOP_MODIFY_DIST	The corner loop modifying distance. If there exists a corner loop within this distance, the user gets the opportunity to modify its geometry.	
CORN_LOOP_RADIUS	Corner loop radius.	
CORN_LOOP_RECREATE	**No =**	Do not recreate corner loops when opening an existing nest (System default).
	Yes =	Recreate corner loops when opening an existing nesting.
CUTLINE_COLOUR	The colour for cutlines.	
CUTSEQINFO_COLOUR	The colour for the cutting sequence information in the burning sketch.	
CUTTING_SEQUENCE_SYMBOL	Symbol for cutting sequence information.	
	S1 =	Symbol for burning sketch.
	S2 =	Symbol for detailed sketch.
CVBA_INTERVAL	The angle interval for continuously varying bevel angle.	
CVBA_TOLERANCE	The tolerance used in the interpolation of the continuously varying bevel angle.	
DELETE_IDLE	Code for automatic deletion of the idle movement from the Nesting burning sketch after VERIFY:	
	No =	Do not delete.
	Yes =	Delete.
DENSITY	The density. The current default value will be stored on the nested plate. The stored value is used in all calculations. The unit should be kg/ mm^3.	
DETAILED_SKETCH	Show detailed sketch in form.	
	No =	Do not show.
	Yes =	Show

DETAILED_SKETCH_AREA	Plate parts with area smaller than given value will be shown in a detailed sketch.
DETAILED_SKETCH_SCALE	Scale of detailed sketch, relative to the sketch scale.
DIRECTION_DEF	The default parameter DIRECTION_DEF indicates the directions that should be available when directions are inserted into the burning sketch. To get all directions the parameter shall have the value 1234567.
DIRECTION_MIN_AREA	Minimum area of plate for which side information texts are added to sketch.
DIRECTION_NAME1	= TOP
DIRECTION_NAME2	= BOTTOM
DIRECTION_NAME3	= FORE
DIRECTION_NAME4	= AFT
DIRECTION_NAME5	= CL
DIRECTION_NAME6	= PS
DIRECTION_NAME7	= SB If only TOP, AFT, CL and SB are to be shown then the value should be 1457. The order in which numbers are given is irrelevant.
DIRECTION_NO_DEF	The number of directions to be shown in the burning sketch. If value is equal to 2 and a cutting sequence symbol with two insertion points for texts will be added to symbol.
DIRECTION_TEXT	Height of direction texts.
DISPL_AUX	Code for auxiliary function display. **No =** Auxiliary functions will not be displayed in the VERIFY function. No confirmation is needed **Yes =** Auxiliary functions will be displayed in the VERIFY function. The result on the CL-file is independent of the value of DISPL_AUX.

DISPL_CODE	Parallel displacement code. This code controls if parallel displacement of contours will be performed when a plate part is brought up from the data bank and positioned on the parent plate (the functions NEST, OPEN NESTING JOB and PARTS MENU). If the code is set to 1, the outer contour is displaced outwards and all holes are displaced inwards. Marking lines are not modified.
DISPL_DIST	Parallel displacement outdistances is the default distance of the parallel displacement.
DIST_BLAST_PLATE	Distance between blasting line and raw plate edge.
DIST_CF_PLATE	Distance between cut free geometry and raw plate edge.
DIST_MAIN_PARALLEL	The minimum distance between the main and parallel burner.
DIST_MARK_PLATE	Distance between marking line and raw plate edge.
DIST_PART_PART	Distance between parts. This distance is a minimum distance which is default for the transformation functions **BUMP** and **PARALLEL**. The functions transform plate parts in a way that the distance between the indicated parts is never less than the default distance.
DIST_PART_PLATE	Distance between parts and raw plate edge. This distance is a minimum distance which is default for the transformation functions BUMP and PARALLEL. The functions transform plate parts in a way that the distance between the part and the raw plate edge is never less than the default distance.
DIST_PLATE_DETSK	Distance from raw plate edge in burning sketch to detailed sketch.
DIST_PLATE_VERT_MARK	Distance from raw plate edge in burning sketch to dimension for first vertical marking.
DIST_PRODINFO_TEXTS	The minimum distance between production information texts generated automatically.
DRAW_RAW_PLATE_RECT	This code controls if the rectangle of the raw plate should be drawn as a solid line in the burning sketch or not. **No =** Do not draw rectangle. **Yes =** Draw rectangle.

DWGNO_ASK	This code controls if the system should ask for the drawing number in NEW NESTING JOB and OPEN NESTING JOB. **No =** Do not ask for drawing number. **Yes =** Ask for drawing number.
EDIT_NEST_FILTER	This code controls if the filter file can be updated by the user. **No =** Update not possible **Yes =** Update possible
END_MARK_POS	End marking position. This is the explicitly given position between marking and burning connected to the start/end position code.
END_POS	End position. This is the explicitly given end position (u- and v-coordinates) connected to the start/end position code.
EQ1_MAX_X	Maximum x value for equipment 1.
EQ1_MAX_Y	Maximum y value for equipment 1.
EQ1_MIN_X	Minimum x value for equipment 1.
EQ1_MIN_Y	Minimum y value for equipment 1.
EQ2_MAX_X	Maximum x value for equipment 2.
EQ2_MAX_Y	Maximum y value for equipment 2.
EQ2_MIN_X	Minimum x value for equipment 2.
EQ2_MIN_Y	Minimum y value for equipment 2.
EXCESSINFO_COLOUR	The colour for the excess information in the burning sketch.
EXCHANGE_BURNER	Code for exchanging the burner id for old nests: **No =** User not allowed to exchange the burner id **Yes =** User allowed to exchange the burner id
EXPORT_FORMAT	Code for the type of output file from VERIFY: **XML =** XML format **Generic =** Generic file format.

GAP_IN_HOLE_START	Code for automatically creating a gap in a hole start: **No =** Do not create a gap **Yes =** Create a gap
GAP_RESTART	Gap restart code. The system includes a feature for restart at every gap encountered in the tool path. The restart includes an ending hook at the beginning of the gap, a starting hook at the end of a gap and resumed burning. **0 =** No automatic restart in gaps. **1 =** Automatic restart in gaps.
GEN_REST_PLATE	This code controls how to generate restplates when the total quantity of the parent plate is larger than one. **0 =** One restplate with n copies. **1 =** n restplates with one copy each having a unique name.
GET_STORE_FORM	This code controls if forms can be updated by the user. **0 =** Update not possible. **1 =** Update possible.
GET_STORE_HOOK	This code controls if hooks can be updated by the user. **0 =** Update not possible. **1 =** Update possible.
GPP_3AX_RESTRICTION_FILE	Restriction file for 3-axis burning machine.
GPP_BLAST_COLOUR	The colour used in the presentation of the blasting.
GPP_BLAST_PARALLEL_COLOUR	The colour used in the presentation of the parallel blasting.
GPP_BURN_BEVEL_BOTH	The line type used in the presentation of bevel burning (BOTH sides).
GPP_BURN_BEVEL_CVBA	The line type used in the presentation of bevel burning (Continuously Varying Bevel Angles).
GPP_BURN_BEVEL_OS	The line type used in the presentation of bevel burning (other side).

GPP_BURN_BEVEL_OS_COLOUR	Colour for contours denoting bevel other side in function check generic post processor.
GPP_BURN_BEVEL_TS	The line type used in the presentation of bevel burning (this side).
GPP_BURN_BEVEL_TS_COLOUR	Colour for contours denoting bevel this side in function check generic post processor.
GPP_BURN_COLOUR	The colour used in the presentation of the burning.
GPP_BURN_PARALLEL_COLOUR	The colour used in the presentation of the parallel burning.
GPP_CHECK	Run check postprocessor command verify process. **Yes =** Run. **No =** Do not run.
GPP_CTRL_FILE	Default file for generic post processor. Must be in SB_SHIP directory.
GPP_IDLE	Line type denoting idle movements in function check generic post processor.
GPP_IDLE_COLOUR	Colour for lines denoting idle movements in function check generic post processor.
GPP_IDLE_PARALLEL_COLOUR	Colour for lines denoting parallel idle movements in function check generic post processor.
GPP_KERF_FILE	Kerf file for generic post processor. Contains kerf and speed information, must be placed in SB_SHIP directory.
GPP_LABEL_SYMBOL_COLOUR	The colour used in the presentation of labelled symbols.
GPP_LABEL_TEXT_COLOUR	The colour used in the presentation of labelled text.
GPP_MARK	Line type denoting the marking movements In function check generic post processor.
GPP_MARK_BLAST_COLOUR	The colour used in the presentation of the simultaneous marking-blasting.
GPP_MARK_COLOUR	The colour used in the presentation of the marking.
GPP_MARK_PARALLEL_COLOUR	The colour used in the presentation of the parallel marking.
GPP_RUN	Run generic post processor after verify process.

	Yes =	IRun.
	No =	Do not run.
GRINDING_BEVEL_CODES	Edge grinding should be done when any of the following bevel codes are defined.	
GRINDING_COLOUR	Colour for grinding information in burning Sketch.	
GSD_IN_MARK_TRACE	Code for the option to include GSD geometry in the marking line trace:	
	No =	Do not include.
	Yes =	Include.
HOLE_BURNING_DIRECTION	Code for the direction of burning holes on plate.	
	None =	None.
	X-direction =	x-direction.
	Y-direction =	y-direction.
HOLE_BURNING_TOLERANCE	The tolerance for determining the direction when sorting the holes centre points.	
HOLE_DIRECTION	This code controls the cutting direction in holes.	
	0 =	No check made.
	1 =	Forced cutting counter-clockwise.
	2 =	Forced cutting clockwise.
HOOK_END	Ending hook. This is the standard hook number for the hook to be used at the end of a tool path on an outer contour. -1 indicates that no hook shall be used.	
HOOK_END_GAP	The number of the ending hook when the restart gap facility is used.	
HOOK_END_HOLE	Ending hook in holes. This is the standard hook number to be used at the end of a tool path on a hole. -1 indicates that no hook shall be used.	

HOOK_START	Starting hook. This is the standard hook number for the hook to be used at the start on an outer contour. -1 indicates that no hook shall be used.
HOOK_START_GAP	The number of the starting hook when the restart gap facility is used.
HOOK_START_HOLE	Starting hook in holes. This is the standard hook number to be used at the start of a hole. -1 indicates that no hook shall be used.
IDLE_COLOUR	Idle colour in toolpath shown in burning sketch.
KEEP_VERIFY_COLOUR	Code for the verification colour: **No =** Do not keep the colour. **Yes =** Keep the colour (the colour is automatically deleted when the **VERIFY** and **STORE** functions are used).
KERF_COMP	Kerf compensation. This is the value of the kerf compensation.
KNUCKLE_LINE_ANGLE	Controls which angle to show in burning sketch. **Knuckle =** Show knuckle angle. **Opening =** Show opening angle.
KNUCKLE_LINE_TEXT	Text to show with knuckle angle in burning sketch.
LABEL_BACKWARDS	This code controls if backwards labelling is supported. **No =** Backwards labelling not supported. **Yes =** Backwards labelling supported.

LABEL_BEVEL_DEGREE	Code for including the degree character in the labelling bevel text: **No =** Do not include. **Yes =** Include.
LABEL_EXCESS_SIGN	Code for including the +/- character in the labelling of the excess text: **No =** Do not include. **Yes =** Include
LABEL_POSNO_SYMBOL	Code for displaying the position number symbol in the burning sketch, for labelled position numbers: **No =** Do not display **Yes =** Display
LABEL_START_REF_SYMBOL	Code for drawing a reference symbol at a labelled start: No = Do not draw. Yes = Draw.
LABEL_SYMB_HEIGHT	Symbol height for labelled symbols.
LABEL_SYMB_ROTATION	Symbol rotation angle for labelled symbols.
LABEL_TEXT_GEOMETRY	Code for creating the labelled text geometry in the generic file: **No =** Do not create. **Yes =** Create.
LABEL_TEXT_HEIGHT	The height of the labelled text.
LABEL_TEXT_HGT_CODE	Value transferred into generic file, can be used to control labelling equipment.
LABEL_TEXT_LINE_DIST_FACT	Factor used in automatic placing of labelled position numbers (the corresponding Drafting default is multiplied with this parameter).
LABEL_TEXT_ROTATION	Text rotation angle for labelled text.
LABEL_TEXT_ROTATION_FIXED	Controlling the text rotation of labelled Position numbers. **No =** Rotate position numbers to be aligned with marking lines.

	Yes =	Always put position numbers according to angle set by LABEL_TEXT_ROTATION.
LIST_BEVLEN_EXT	Extension of file containing list of bevel lengths for all plate parts in nest.	
LOCATION_CL_DIST	Minimum distance across CL for an assembly part to be regarded as a CL part (used in the workshop lists).	
LOCATION_CL_TEXT	Text used for indicating parts extending over CL in automatic labelled texts.	
LOCATION_PS_TEXT	Text used for indicating for port side parts in automatic labelling.	
LOCATION_SB_TEXT	Text used for indicating for starboard side parts in automatic labelling.	
MARKING_ALL_FIRST	Code for the treatment of marking in the function AUTOMATIC START:	
	No =	Partwise marking.
	Yes =	Platewise marking.
MARK_AUTO_OPPOSITE	Code describing which marking lines to treat in automatic marking:	
	No =	Mark only this side.
	Yes =	Mark all.
MARK_COLOUR	The colour for marking lines.	
MARK_COLOUR_OS	The colour for marking lines on the other side.	
MARK_COLOUR_TS	The colour for marking lines on this side.	
MARK_PART_USED	This code controls if the nested parts should be checked against the part file or not.	
	No =	No check.
	Yes =	Check.
MARK_POSNO_SELECTION	This code controls how to determine if two parts can be considered to be equal.	
	No =	Block and position number are checked..

	Yes = GPS1-GPS4, position number and bevel are checked. If no GPS data are available, only the part names are compared.
MARK_POSNO_TEXT_OS	Text to be added in the burning sketch when • a part is mirrored AND • no starts have been put for the marking lines AND • all marking lines are on OS
MARK_SIZE_WELD_SYMBOL	Limiting part area for adding bevel symbols instead of bevel texts in the Nesting burning sketch. If the circumscribed area of the part is less than the parameter, symbols will be used.
MARK_TEXT	Marking text to be placed on marking lines caused by folded flanges.
MARK_TEXT_CODE	This code controls whether the text in MARK_TEXT shall be placed on marking lines caused by folded flanges. **0 =** Do not place text **1 =** Place text stored in MARK_TEXT
MARK_TEXT_COLOUR	Colour to apply to texts in burning sketch.
MARK_TEXT_NOM_HEIGHT	Code for adding the nominal height of a folded flange to the parameter MARK_TEXT: **No =** Do not add **Yes =** Add
MARK_TEXT_OS	Additional text for side of folded flange (other side).
MARK_TEXT_TS	Additional text for side of folded flange (this side).
MAX_CONTOUR	Maximum number of contours per plate part. For the purpose of dimensioning data structures, the maximum number of contours per plate part (i.e. number of holes + number of marking lines + 1) must be decided in advance. There is a connection between this value and the maximum number of plate parts nested on the same parent plate: the product between the two maximum values must be less than 32768.

MENUE_SCALE	Parts menu scale factor. When a parts menu is created, each part is scaled with its scale factor to make it possible to have room for many parts on the menu. When a part is picked from the menu to be nested, it is automatically scaled with the reciprocal scale factor. The following 11 default parameters controls which types of plate parts that are accepted in the NEST and PARTS MENU functions. The following values are valid: **No =** Not accepted **Yes =** Accepted
NEST_BEND_TEMP	Bending templates.
NEST_BRA_PLA	Bracket plates.
NEST_CHAMFER_PARTS	Nest chamfer parts.
NEST_CLIPS	Nest clips.
NEST_CONV_PROFILES	Nest converted profiles.
NEST_DEV_DOU_CUR_PLA	Developed, double-curved plates.
NEST_DEV_SIN_CUR_PLA	Developed, single-curved plates.
NEST_DOUBLE_PLATES	Nest doubling plates.
NEST_EXISTING_SKETCH	Code controlling the default check box in the VERIFY dialogue: **Whole =** Recreate whole. **Partial =** Recreate partially, only tool path and texts. **Toolpath =** Recreate partially, only tool path.
NEST_JIG_PLA	Jig plates.
NEST_PLANE_PARTS	Plane parts.
NEST_STA_BRA_PLA	Standard bracket plates.
NEST_MAIN_COLOUR	The colour for the main part in parallel nesting.
NEST_PARALLEL_COLOUR	The colour for the parallel part in parallel nesting.

NEST_SINGLE_COLOUR	The colour for single nested plate parts.
NEST_TYPE_ASK	Code for asking for nesting type in New Nest Job: **No =** Do not ask. **Yes =** Ask. **Automatic** Type =1.
NORM_STEEL_QUAL	The normal steel quality. The following parameters are leading texts for the text which is placed in the burning sketch for the normal steel quality restplates. An empty string will result in that this line is omitted. If all leading texts are empty strings no texts will be put in the burning sketch.
NSQ1_TEXT_1	= Ship number.
NSQ1_TEXT_2	= Drawing.
NSQ1_TEXT_3	= Restplate.
NSQ1_TEXT_4	= Rest code. The following parameter is a leading text for the text which is placed in the burning sketch for the normal steel quality workshop rests. An empty string will result in that no text will be put in the burning sketch.
NSQ2_TEXT_1	= Restplate.
OPERATOR_SUP_ADD	Defines if the new operator name, defined by SBB_USER_SIGNATURE, supersede or is added to the old operator name. **supersede e =** Supersede old operator with new **Add =** Add new operator to old, separated with a slash (/) See also CHANGE_OPERATOR
OVERLAP_CHECK	Overlap check code. This code controls the action, if a plate part is overlapping another plate part or outside the standard parent plate. The check can be made in the functions **N**EST, TRANSFORM, OPEN NESTING JOB, EXCHANGE RAW PLATE and EXCHANGE PLATE PART. The code controls the following actions:

	0 =	No overlap check is performed at all
	1 =	The overlap is checked. If there is overlap a warning is given and the plate part must be moved before it is possible to continue
	2 =	The overlap is checked. The operator is prompted to decide whether the part shall be accepted or not
PARTCHECK_HOLE_MINSZ		The size of the largest hole in text bumping (part checking only)
PARTS_MENU_AUTOMATIC		Code for automatic generation of the parts menu:
	No =	No automatic generation.
	Yes =	Automatic generation.
PARTS_MENU_CURR		Current position in the parts menu (NIT only).
PARTS_MENU_EQUAL		Code for the presentation of identical parts in the parts menu:
	No =	Do not present identical parts together.
	Yes =	Present identical parts together.
PARTS_MENU_SELECTION		Code for the selection of plate parts:
	Part =	Select via block/panel name.
	Assembly =	Select via assembly name.
PARTS_MENU_SIZE		The parts menu size (NIT only).
PARTS_MENU_WIDTH		The width of the nesting parts menu when it is automatically generated. It should be given as a factor of the raw plate width.
PARTS_START_POS		Code for how to put the start on the outer contour (automatic starts only):
	0 =	Closest notch.
	1 =	Closest to the previous hole/ marking/outer contour.
PLATE_ALIGNMENT_SYMBOLS		The symbol numbers for the symbols used in the plate alignment functions.

PLATE_ALIGNMENT_SYMBOL_HEIGHT	The height of the plate alignment symbols.
PLOT_WITHOUT_FORM	Plot the burning sketch without for **Yes =** **No =**
POSNO_COLOUR	Colour for position numbers.
POSNO_HEIGHT	The height of the position number symbols.
POSNO_SYMB	Symbol number in symbol font 2 of the symbol used for the position number of a plate part. The symbol numbers available are 60-79. Symbol number 0 indicates that no symbol at all shall be used.
POSNO_SYMB_BAR	Symbol number in symbol font 2 of the symbol used for the position number of a marking line caused by a bar. The symbol numbers available are 60-79. Symbol number 0 indicates that no symbol at all shall be used.
POSNO_SYMB_MARK	Symbol number in symbol font 2 of the symbol used for the position number of a marking line. The symbol numbers available are 60-79. Symbol number 0 indicates that no symbol at all shall be used.
QUAL_CHECK	Quality check code. This code controls the action, if an attempt is made to nest a plate part with one quality code on a parent plate with another. The code controls the following actions: **0 =** No quality check is performed. **1 =** The qualities are checked. If they do not match, the plate part is automatically rejected. **2 =** The qualities are checked. The operator is prompted to decide whether the part shall be allowed or not. **3 =** The qualities are checked according to the rules given in the file assigned to the logical variable SBH_QUALITY_EXCH. If they do not match, the plate part is automatically rejected.

QUICKNEST_ANGLE	The rotation angle increment in quick nesting.
QUICKNEST_CLUSTER	Code for clustering of parts in quick nesting: **No =** No clustering **Yes =** Clustering
QUICKNEST_CLUSTER_AREA	The maximum area in mm^2 for parts to be considered in clustering. (Brackets, clips, converted profiles and doubling plates are always clustered if possible.)
QUICKNEST_CLUSTER_RATIO	Minimum area/perimeter ratio for the two parts to be clustered.
QUICKNEST_CONFIRM	Code for layout confirmation in quick nesting: **No =** No confirmation. **Yes =** Confirmation after each part.
QUICKNEST_IN_HOLE	Code for nesting in holes in quick nesting: **No =** Do not use holes. **Yes =** Use holes.
QUICKNEST_MIRROR	Code for mirroring parts in quick nesting: **0 =** Do not mirror. **1 =** Mirror only if the number of marking lines is equal on both sides **2 =** Mirror always.
QUICKNEST_NEST_SELECTION	Criteria for best nest in quick nesting: **Minimum X =** Minimize the extension in the x-direction. **Scrap =** Minimize the scrap of the circumscribed rectangle.
QUICKNEST_PART_SELECTION	Criteria for part selection in quick nesting: **Area =** Select the parts with decreasing area. **Perimeter =** Select the parts with decreasing perimeter.

QUICKNEST_STARTPOS	The number of different starting positions used in quick nesting.
QUICKNEST_STRIPS	The number of area strips used in quick nesting in the calculation of the remaining available raw plate area.
RAW_PLATE_INFO	A string to be used in multiple nesting describing the contents of the line which is presented below each nesting. The following keywords are available: <NEST_NAME> <QUALITY> <LENGTH> <WIDTH> <THICKNESS> To present the information in 2 lines, the keyword should be given after the last keyword on the first line.
RECT_PART_DPREFIX	The leading text for the diagonal measure of rectangular parts.
RECT_PART_LENGTH	Minimum length of rectangular part to get control measure.
RECT_PART_WIDTH	Minimum width of rectangular part to get control measure.
RECT_PART_LPREFIX	The leading text for the length measure of rectangular parts.
RECT_PART_WPREFIX	The leading text for the width measure of rectangular parts.
REFERENCE_SYMB	Symbol number in symbol font 2 of the symbol used as reference symbol, i.e. for pointing at a marking line in a position number entity. The symbol numbers available are 80-99.
REFLECT_ASSEMBLY_LENGTH	Code used to determine if the length or number of marking lines shall be calculated, REFLECT_PART = 3 only. **No =** Calculate the number of marking lines. **Yes =** Calculate the length of the marking lines.

REFLECT_ON_ASSEMBLY

This code controls the automatic reflection when a plate part is read into the Nesting System. Only the assembly with the largest number of marking lines is treated. Cf. REFLECT_PART.

No = Do not reflect plate part on assembly.

Yes = Reflect plate part on assembly

REFLECT_PART

This code controls whether the plate part should be reflected in the x-axis if "opposite side" contains more marking contours than "this side".

0 = Do not reflect plate part.

1 = Reflect plate part.

2 = Reflect part according to the rules give by the logical variable SBH_MARKING_SIDE (cf. *User´s Guide Hull Prod Information*.

3 = Reflect plate part if the length/ number of the marking lines with the same assembly on the opposite side, is longer/larger.

4 = Reflect plate part according to:

- Number of marking lines + GSD' s except corner GSD's)
- If no marking lines exist, according to REFLECT_PART = 2.

REF_SYMB_PART

Symbol number in symbol font 2 of the symbol used for pointing at the detail in a position number entity. The symbol numbers available are 60-79.

RESTPLATE_NAME_PATTERN

The name format for restplates. Can contain any of substrings:

```
<sclass><quality><thickness><year><month>
```

and user defined strings.

REST_CLASSIFICATION

This code controls if the special steel quality restplates should be classified or not.

No = No classification.

Yes = Classification.

REST_MAXDEV_ANGLE

The maximum angle a line can deviate from horizontal/vertical direction without being considered to be horizontal/vertical

REST_PLATE_NAME

Code for the rest plate naming rule:

1 = Name is selected administrative name + 4 digit running number.

2 = Name auto generated administrative name + 3 digit running number.

REST_PLATE_SEP

Separator used for restplates.

REST_PLATE_SET

Code for rest plate types

1 has types:

1 = Rectangle
2 = Circle
3 = Triangle
4 = Symmetrical T-form
5 = L-form 6 = Right angle trapezi-form

2 has types:

1 = Rectangle
2 = Right angle trapezi-form (“width”)
3 = Triangle
4 = Right angle trapezi-form (“length”)
5 = Truncated rectangle
6 = L-form

Yes = Classification.

RULER_DIST

Code for placing the meter ruler in the burning sketch:

= 0.0 No ruler.

0.0 < f <=1.0 The ruler is placed the distance f*w above the plate, where w is the width of the raw plate.

> 1.0 The distance in mm.

SACHNR_MISS_EXIT

Code to handle the action if the raw plate is missing in the NP-file (NIT only):

No = Let the user decide.

Yes = Exit Nesting.

SELECT_RAWPLATE

Defines if rawplate name should be selected by quality and thickness or by keying in name.

Yes = Select by quality and thickness.

No = Key in name.

SIDEINFO_COLOUR	The colour for the side information in the burning sketch. The following parameters are leading texts for the text which is placed in the burning sketch for the special steel quality restplates. An empty string will result in that this line is omitted. If all leading texts are empty strings no texts will be put in the burning sketch.
SIDE_FOLDED_FLANGE	Code for reflecting parts with folded flanges: **= -1** Folded flange at other side. **= 0** Automatic **= 1** Folded flange at this side.
SIDE_SWEDGE_WALL	Code for reflecting parts with swedging: **TS =** Swedge at this side **OS =** Swedge at other side
SKETCH_SCALE	Predefined scale for burning sketch in form.
SSQ_TEXT_1	= Ship number.
SSQ_TEXT_2	= Drawing.
SSQ_TEXT_3	= Parent plate.
SSQ_TEXT_4	= Restplate.
SSQ_TEXT_5	= Counting number.
STAND_SYMB_HEIGHT	Standard height of symbols.
STAND_TEXT_HEIGHT	Standard height of texts.
START_ANGLE	Min node angle. This value is the preferred minimum angle for a node point where you place a start. It is only valid if closest node point is preferred. The angle has to be found within a given distance from the closest node point, Otherwise, the start will be placed in the closest node point.
START_BURN_AUTO	This is the explicitly given start position (u- and v-coordinates) for the burning. This parameter is only used in the optional programs **AUTONEST** (se016) and **AUTONEST2** (SE021) and have no effect in the Nesting System.
START_CODE	Start code. This is a code controlling the standard way of defining the start and end of a burning start. **0 =** Unknown

	1 = Start in gap, end in next gap
	2 = Start in arbitrary point on a contour, end in the same point or in the next gap whichever comes first in the tool path
	3 = As 2 but end in an arbitrary point on the contour
START_COLOUR	The colour for starts.
START_DIR_HOLE	The angle used to determine the starting point in a hole in the function AUTOMATIC START. If the value is in the open interval [0.360[, the lines with the angles given by the parameter START_DIR_HOLE and START_DIR_HOLE + 90, are used to calculate 4 points in the hole contour. The point closest to a given point is selected. The following values can be used to force the start to be put in a certain direction: + X + Y - X - Y
START_DIR_HOLE_TYPE	The type of holes for which START_DIR_HOLE shall be used: None Circular Elliptical General All
START_END_ACT	Start/end position code. This code controls the action before the first start and after the last start of a plate. The following activities are possible:
	0 = The tool path starts at the first start (no initial idle movement) and ends at the end position at the last start (no ending idle movement).
	1 = The tool path starts at an explicitly defined position with an idle movement to the first start.
	2 = The tool part ends with an idle movement to an explicitly defined position.
	3 = Both start and end are explicitly defined.

	4 = As 0 but also an idle movement between marking and burning to an explicitly defined position.
	5 = As 1 but also an idle movement between marking and burning to an explicitly defined position.
	6 = As 2 but also an idle movement between marking and burning to an explicitly defined position.
	7 = As 3 but also an idle movement between marking and burning to an explicitly defined position.
START_MARK_AUTO	This is the explicitly given start position (u- and v-coordinates) for the marking. This parameter is only used in the optional programs AUTONEST (se016) and AUTONEST2 (se021) and have no effect in the Nesting System.
START_NODE	Node point preference for start creation. This is a default value analogous to the node point preference for bridge creation.
START_POS	Start position. This is the explicitly given start position (u- and v-coordinates) connected to the start/end position code.
STOP_BURN_OUTER	This code controls if a stop should be inserted after cutting an outer contour, and before the next idle movement.
	No = No stop
	Yes = Stop
SWEDGE_HEIGHT	Put swedge nominal height to swedging marking lines in burning sketch.
	= No
	= =Yes
SWEDGE_SIDE_OS	Swedge side orientation string for other side.
SWEDGE_SIDE_TS	Swedge side orientation string for this side.
SWEDGE_TEXT	Text for swedging marking line.
SYMB_HEIGHT	Height of symbols.
SYMB_ROTATION	Rotation of symbols.

TEXT_HEIGHT	Height of texts.
TEXT_HEIGHT_COMPENSATION	Text height for the compensation texts in burning sketch.
TEXT_HEIGHT_EXCESS	Text height for the excess texts in burning sketch.
TEXT_LINE_DIST	The distance between the marking line (folded flange) and the placed marking text (MARK_TEXT) and the distance between the position number line and the position number symbol.
TEXT_ROTATION	Rotation of texts.
TEXT_FONT	The current text font number.
TEXT_TYPE	The text type for labelled texts to be defined: **0 =** Texts to be Printed only. **1 =** Texts to be labelled but not Printed. **2 =** Texts to be labelled and Printed (default).
TEXT_TYPE1_COLOUR	The colour for labelled texts with TEXT_TYPE = 0.
TEXT_TYPE2_COLOUR	The colour for labelled texts with TEXT_TYPE = 1.
TEXT_TYPE3_COLOUR	The colour for labelled texts with TEXT_TYPE = 2.
THICK_CHECK	Thickness check code. This code controls the action, if an attempts is made to nest a plate part with one thickness on a parent plate with another. The code can control the following actions: **0 =** No thickness check is performed. **1 =** The thicknesses are checked. If they do not match, the plate part is automatically rejected. **2 =** The thicknesses are checked. The operator is prompted to decide whether the part shall be allowed or not.

THICK_DIFF	Thickness check range. This is the allowed difference between the thickness of a parent plate and a part to be nested and is used in the thickness check (cf thickness check code).
THICK_DIFF_MAX	The upper limit of the thickness check range.
THICK_DIFF_MIN	The lower limit of the thickness check range. The allowed difference between the thickness of a parent plate and a part to be nested was earlier controlled by the parameter THICK_DIFF. If there is a need to have different values of the negative tolerance and the positive tolerance the two parameters above can now be used. These two parameters can of course have the same value.
THICK_MAIN_MAX	The maximum thickness for the main burner.
THICK_MAIN_MIN	The minimum thickness for the main burner.
THICK_PARALLEL_MAX	The maximum thickness for the parallel burner.
THICK_PARALLEL_MIN	The minimum thickness for the parallel burner.
THICK_PRECISION_MAX	The maximum thickness for the precision burner.
THICK_PRECISION_MIN	The minimum thickness for the precision burner.
TREATMENT_CHECK	The code for treatment check: **0 =** No check **1 =** Check **2 =** Check and verify **3 =** Check according to the rules in the file assigned to SBH_TREATMENT_CHECK
USE_BUMP_CONTOUR	Code for using the bump contour in the transformation of parts: **No =** Do not use the bump contour **Yes =** Use the bump contour
VERIFY_ABORT_ON_PATH_ERRO R	Code for aborting VERIFY when the tool path is outside the allowed boundaries: **No =** Continue VERIFY (with warning)

	Yes = Abort verify
VERIFY_COLOUR	The colour used in the VERIFY function.
VERIFY_DELAY_TIME	The delay time in milliseconds between each segment in the tool path verification on the screen.
VERIFY_DIALOGUE_BOX	Code for displaying a question dialog box in VERIFY (starts and auxiliaries): **No =** Do not display box, give ordinary message. **Yes =** Display dialog box.
VERTICAL_SWEDGE_TEXT	Text for vertical swedging line.
VERTICAL_MARK_DIST	The limiting distance for the first vertical marking. If the distance is >=VERT_MARKDIST the measure is set.
WELD_TOLERANCE_CHAMFER	The welding tolerance used in nesting of chamfer parts.
WORKING_SEQUENCE	The working sequence to be used in the VERIFY function. The following keywords are available: Undefined BlastAll MarkAll BurnInner BurnOuter BurnAll LabelText LabelSymbol LabelAll If the working sequence shall be marking, labelling, burning of holes and burning of outer contour the sequence should be: 'MarkAll LabelAll BurnInner BurnOuter'
WORKING_SEQUENCE_SORT	Code for sorting according to the sequence given in WORKING_SEQUENCE when the start order does not follow the given sequence: **N =** Do not sort automatically. **Y =** Sort automatically.

6.1.1 Below follows an example of the default system

```
ACCURACY_OF_RAW_PLATE_LENGTH     = 10
ACCURACY_OF_RAW_PLATE_WIDTH      = 5
ACT_AFTER_LABELLING              = 0
ADDITIONAL_EXCESS                = 30.0  0.0
ANGLE_CHECK                      = 1
ANG_CONSEC_SEG                   = 10.0
AUTO_GRINDING_INFO               = Yes
AUTONEST_CLUSTER                 = No
AUTONEST_FILL_TYPE               = 1
AUTONEST_FORMNAME                = 'NESTFORM'
AUTONEST_IN_HOLE                 = No
AUTONEST_LAYOUT_ONLY             = No
AUTONEST_MAXSCRAP_MIRROR         = 0.20
AUTONEST_MAXSCRAP_MULTI          = 0.20
AUTONEST_MAXSCRAP_SINGLE         = 0.20
AUTONEST_NESTNAME_PREFIX         = 'AUTONEST-'
AUTONEST_OPT_TYPE                = 0
AUTONEST_PART_ANGLE              = 90.0
AUTONEST_PART_SET                = 1
AUTOSCALE_FORM                   = 1
AUTO_BEVEL_INFO                  = Yes
AUTO_EXCESS_INFO                 = No
AUTO_GEN_COUNR                   = No
AUTO_HULL_MARK_INFO              = No
AUTO_NEST_NAME                   = No
AUTO_POSNO                       = Yes
AUTO_POSNO_DIST                  = 0.30
AUTO_POSNO_PART                  = 0
```

```
AUTO_POSNO_POSITION             = 0.0  0.5
AUTO_SIDE_INFO                  = Yes
AUX_COLOUR                      = White
AUX_FUNC_RANGE                  = 100.0
BEVELINFO_COLOUR                = Cyan
BEVEL_LIMIT_TAG_LENGTH          = 100.0
BEV_OS_COLOUR                   = Cyan
BEV_TS_COLOUR                   = White
BRIDGE_COLOUR                   = Cyan
BRIDGE_NODE                     = 25.0
BRIDGE_RADIUS                   = 0.00000
BRIDGE_WIDTH                    = 20.0
BRIDGE_WIDTH_HOLE               = 20.0
BRIDGE_YDIR                     = 0
BURN_BEVEL_I                    = Fixed
BURN_BEVEL_MIN_SEG_LENGTH       = 0.0
BURN_BEVEL_X                    = No
BURN_BEVEL_X_MAX_ANGLE_OS       = 0.0
BURN_BEVEL_X_MAX_ANGLE_TS       = 0.0
BURN_BEVEL_X_MIN_ANGLE_OS       = 0.0
BURN_BEVEL_X_MIN_ANGLE_TS       = 0.0
BURN_BEVEL_Y                    = Yes
BURN_BEVEL_Y_MAX_ANGLE_OS       = 50.0
BURN_BEVEL_Y_MAX_ANGLE_TS       = 50.0
BURN_BEVEL_Y_MIN_ANGLE_OS       = 10.0
BURN_BEVEL_Y_MIN_ANGLE_TS       = 10.0
BURN_NO_BEVEL_TEXT              = 'Manual bevel burning'
CF_RESTART                      = 1
CHANGE_OPERATOR                 = No
CHANGE_SHIP_NR                  = No
```

```
COLOURS_IN_SKETCH              = No
CONT_DIRECTION                 = 1
CORNER_ANGLE                   = 5.0
CORN_LOOP_CTRL                 = 1
CORN_LOOP_LENGTH               = 10.0
CORN_LOOP_MAXNO_AUXFUNC        = -1
CORN_LOOP_MINANGLE             = 10.0
CORN_LOOP_MODIFY_DIST          = 10.0
CORN_LOOP_RADIUS               = 0.0
CUTLINE_COLOUR                 = Magenta
CUTSEQINFO_COLOUR              = Blue
CVBA_INTERVAL                  = 1.0
CVBA_TOLERANCE                 = 0.1
DELETE_IDLE                    = No
DENSITY                        = 7.86E-06
DIRECTION_DEF                  = 1234567
DIRECTION_NAME1                = TOP
DIRECTION_NAME2                = BOTTOM
DIRECTION_NAME3                = FORE
DIRECTION_NAME4                = AFT
DIRECTION_NAME5                = CL
DIRECTION_NAME6                = PS
DIRECTION_NAME7                = SB
DIRECTION_TEXT                 = 80.0
DISPL_AUX                      = Yes
DISPL_CODE                     = 0
DISPL_DIST                     = 0.0
DIST_BLAST_PLATE               = 50.0
DIST_CF_PLATE                  = 20.0
DIST_MAIN_PARALLEL             = 600.0
```

```
DIST_MARK_PLATE                 = 30.0
DIST_PART_PART                  = 20.0
DIST_PART_PLATE                 = 20.0
DRAW_RAW_PLATE_RECT             = No
DWGNO_ASK                       = Yes
EDIT_NEST_FILTER                = Yes
END_MARK_POS                    = 0.0  0.0
END_POS                         = 0.0  0.0
EQ1_MAX_X                       = 16000.0
EQ1_MAX_Y                       = 3600.0
EQ1_MIN_X                       = -100.0
EQ1_MIN_Y                       = 0.0
EQ2_MAX_X                       = 16000.0
EQ2_MAX_Y                       = 3000.0
EQ2_MIN_X                       = -100.0
EQ2_MIN_Y                       = 0.0
EXCESSINFO_COLOUR               = Magenta
EXCHANGE_BURNER                 = Yes
EXPORT_FORMAT                   = Generic
GAP_IN_HOLE_START               = Yes
GAP_RESTART                     = 1
GEN_REST_PLATE                  = 1
GET_STORE_FORM                  = 1
GET_STORE_HOOK                  = 1
GPP_3AX_RESTRICTION_FILE        = ''
GPP_BLAST_COLOUR                = ForestGreen
GPP_BLAST_PARALLEL_COLOUR       = NavyBlue
GPP_BURN_BEVEL_BOTH             = 0
GPP_BURN_BEVEL_CVBA             = 0
GPP_BURN_BEVEL_OS               = 0
```

```
GPP_BURN_BEVEL_TS               = 0
GPP_BURN_COLOUR                 = Orange
GPP_BURN_PARALLEL_COLOUR        = Firebrick
GPP_IDLE_COLOUR                 = MediumAquamarine
GPP_IDLE_PARALLEL_COLOUR        = Blue
GPP_LABEL_SYMBOL_COLOUR         = Wheat
GPP_LABEL_TEXT_COLOUR           = Wheat
GPP_MARK_BLAST_COLOUR           = BlueViolet
GPP_MARK_COLOUR                 = DarkOrchid
GPP_MARK_PARALLEL_COLOUR        = Magenta
GRINDING_BEVEL_CODES            = '400 401 '
GSD_IN_MARK_TRACE               = No
HOLE_DIRECTION                  = 2
HOLE_BURNING_DIRECTION          = None
HOOK_END                        = 100
HOOK_END_GAP                    = 101
HOOK_END_HOLE                   = 100
HOOK_START                      = 100
HOOK_START_GAP                  = 101
HOOK_START_HOLE                 = 100
KEEP_VERIFY_COLOUR              = Yes
KERF_COMP                       = 1.0
LABEL_BACKWARDS                 = No
LABEL_BEVEL_DEGREE              = No
LABEL_EXCESS_SIGN               = No
LABEL_POSNO_SYMBOL              = Yes
LABEL_SYMB_HEIGHT               = 50.0
LABEL_START_REF_SYMBOL          = Yes
LABEL_SYMB_ROTATION             = 0.0
LABEL_TEXT_GEOMETRY             = No
```

```
LABEL_TEXT_HEIGHT               = 50.0
LABEL_TEXT_HGT_CODE             = 0
LABEL_TEXT_LINE_DIST_FACT       = 10
LABEL_TEXT_ROTATION             = 0.0
LOCATION_CL_DIST                = 0.0
LOCATION_CL_TEXT                = 'Across CL'
LOCATION_PS_TEXT                = 'Portside'
LOCATION_SB_TEXT                = 'Starboard'
MARKING_ALL_FIRST               = Yes
MARK_AUTO_OPPOSITE              = No
MARK_COLOUR                     = Green
MARK_COLOUR_OS                  = Blue
MARK_COLOUR_TS                  = Red
MARK_PART_USED                  = Yes
MARK_TEXT                       = 'FOLDED FLANGE'
MARK_SIZE_WELD_SYMBOL           = 100000
MARK_TEXT_CODE                  = 1
MARK_TEXT_NOM_HEIGHT            = No
MARK_TEXT_OS                    = OS
MARK_TEXT_TS                    = TS
MAX_CONTOUR                     = 100
MENUE_SCALE                     = 0.25
NEST_BEND_TEMP                  = Yes
NEST_BRA_PLA                    = Yes
NEST_CHAMFER_PARTS              = No
NEST_CLIPS                      = Yes
NEST_CONV_PROFILES              = Yes
NEST_DEV_DOU_CUR_PLA            = No
NEST_DEV_SIN_CUR_PLA            = Yes
NEST_EXISTING_TEXT              = Whole
```

```
NEST_JIG_PLA                    = No
NEST_MAIN_COLOUR                = White
NEST_PARALLEL_COLOUR            = Cyan
NEST_PLANE_PARTS                = Yes
NEST_SINGLE_COLOUR              = Wheat
NEST_STA_BRA_PLA                = No
NEST_TYPE_ASK                   = No
NORM_STEEL_QUAL                 = A
NSQ1_TEXT_1                     = 'Ship number'
NSQ1_TEXT_2                     = 'Drawing'
NSQ1_TEXT_3                     = 'Rest plate'
NSQ1_TEXT_4                     = 'Rest code'
NSQ2_TEXT_1                     = 'Workshop rest'
OPERATOR_SUP_ADD                = Supersede
OVERLAP_CHECK                   = 2
PARTCHECK_HOLE_MINSZ            = 10000
PARTS_MENU_AUTOMATIC            = No
PARTS_MENU_CURR                 = 1
PARTS_MENU_EQUAL                = No
PARTS_MENU_SELECTION            = Part
PARTS_MENU_SIZE                 = 50
PARTS_MENU_WIDTH                = 0.50
PLATE_ALIGNMENT_SYMBOLS         = 0 0
PLATE_ALIGNMENT_SYMBOLS_HEIGHT  = 50.00000
POSNO_HEIGHT                    = 300.0
POSNO_SYMB                      = 61
POSNO_SYMB_BAR                  = 62
POSNO_SYMB_MARK                 = 60
QUAL_CHECK                      = 3
QUICKNEST_ANGLE                 = 90.0
```

```
QUICKNEST_CLUSTER               = Yes
QUICKNEST_CLUSTER_AREA          = 1000000
QUICKNEST_CLUSTER_RATIO         = 0.90
QUICKNEST_CONFIRM               = No
QUICKNEST_IN_HOLE               = Yes
QUICKNEST_MIRROR                = 2
QUICKNEST_NEST_SELECTION        = MinimumX
QUICKNEST_PART_SELECTION        = Area
QUICKNEST_STARTPOS              = 2
QUICKNEST_STRIPS                = 50
RAW_PLATE_INFO                  '<NEST_NAME><QUALITY><BR><L
                                ENGTH><WIDTH><THICKNESS>'
REFERENCE_SYMB                  = 83
REFLECT_ASSEMBLY_LENGTH         = Yes
REFLECT_ON_ASSEMBLY             = No
REFLECT_PART                    = 1
REF_SYMB_PART                   = 83
REST_CLASSIFICATION             = Yes
REST_PLATE_SEP                  = -
RULER_DIST                      = 0.0
SACHNR_MISS_EXIT                = Yes
SELECT_RAWPLATE                 = Yes
SIDEINFO_COLOUR                 = White
SSQ_TEXT_1                      = 'Ship number'
SSQ_TEXT_2                      = 'Drawing'
SSQ_TEXT_3                      = 'Parent plate'
SSQ_TEXT_4                      = 'Rest plate'
SSQ_TEXT_5                      = 'Counting number'
STAND_SYMB_HEIGHT               = 20.0
STAND_TEXT_HEIGHT               = 20.0
START_ANGLE                     = 90.0
```

```
START_BURN_AUTO                 = 100000.0  100000.0
START_CODE                      = 1
START_COLOUR                    = Red
START_DIR_HOLE                  = +X
START_DIR_HOLE_TYPE             = 'All'
START_END_ACT                   = 3
START_MARK_AUTO                 = 0.0  0.0
START_NODE                      = 150.0
START_POS                       = 0.0  0.0
STOP_BURN_OUTER                 = No
SYMB_HEIGHT                     = 300.0
SYMB_ROTATION                   = 0.0
TEXT_FONT                       = 0
TEXT_HEIGHT                     = 50.0
TEXT_LINE_DIST                  = 20.0
TEXT_ROTATION                   = 0.0
TEXT_TYPE                       = 2
TEXT_TYPE1_COLOUR               = Green
TEXT_TYPE2_COLOUR               = Yellow
TEXT_TYPE3_COLOUR               = White
THICK_CHECK                     = 2
THICK_DIFF                      = 0.0
THICK_DIFF_MAX                  = 0.0
THICK_DIFF_MIN                  = 3.0
THICK_MAIN_MAX                  = 100.0
THICK_MAIN_MIN                  = 0.0
THICK_PARALLEL_MAX              = 40.0
THICK_PARALLEL_MIN              = 10.0
THICK_PRECISION_MAX             = 10.0
THICK_PRECISION_MIN             = 0.0
```

```
TREATMENT_CHECK                 = 0
USE_BUMP_CONTOUR                = Yes
VERIFY_ABORT_ON_PATH_ERROR      = No
VERIFY_COLOUR                   = Red
VERIFY_DELAY_TIME               = 0
VERIFY_DIALOGUE_BOX             = Yes
VERTICAL_SWEDGE_TEXT            = 'SWEDGING ACCURACY'
WELD_TOLERANCE_CHAMFER          = 150.0
WORKING_SEQUENCE                = 'MarkAll BurnAll LabelAll
                                  '
WORKING_SEQUENCE_SORT           = Yes
```

6.2 Developed Plates

When developed plates are nested they are automatically converted to the plate format for nesting.

The conversion means that the marking information created by the program sf831d - ADD MARKING TO DEVELOPED PLATE will be 'preprocessed' for the nesting system. Depending on the information stored in the developed plates this may result in identifications for marking curves if demanded via an ip to the function.

- **Control Information**

  Below follows a list of all the ip's that can be given. The complete file name should be assigned to the environment variable SBH_DEVOBJ_IP.

```
[CURVEIDENT]
[MARKGAP = <gap length>]
[TEXTHEIGHT = <height of the text>]
[MATERIALSYMBOL [LENGTH = <length> [PLACE = <place>] ] ]
[TEMPLATE [LENGTH = <length> [PLACE = <place>] ] ]
```

  The ip's have the following meaning:

  CURVEIDENT

  If given, the identification for the curves will be given as follows:

frame	FR <no>
waterline	WL <no>
vertical	BT <no>
longitudinal	L <no>
transversal	T <no>

Necessary information must be stored in the plate.

`MARKGAP = <gap length>`

If given, marking contours will start and stop at the distance `<gap length>` from the outer contour. The distance is measured in the direction of the marking line.

`TEXTHEIGHT = <height of the text>`

Tells height of the text of the description names. If the ip is missing, then `<height of the text>` = 100.

`[MATERIALSYMBOL [LENGTH = <length> [PLACE = <place>] ] ]`

If this ip is given, material symbols will indicate on which side of the marking line the material will be placed. The corresponding information must be stored in the plate. For material side marking using an angle the addition LENGTH, `<length>` it is possible to change the length of the legs of the angle from the default value 30 mm to what is given. `<place>` denotes the distance from any of the end points. It is given as a fraction of the contour length, where a positive value denotes distance from the starting point and a negative value denotes distance from the ending point. The distance will never be less than twice the segment length.

`TEMPLATE [LENGTH = <length> [PLACE = <place>] ]`

This ip gives the same information for the templates as the above gives for marking curves except that the sign of `<place>` is irrelevant. If the ip is given, the template names will be substituted by the string "TPL." followed by the template number.

Below is an example of an ip-file:

Example:

```
CURVEIDENT
MARKGAP = 15
TEXHEIGHT = 200
MATERIALSYMBOL
LENGTH = 30
PLACE = 0.5
TEMPLATE
LENGTH = 50
PLACE = 0.2
```

6.3 Bevel Information in the Burning Sketch

In the generation of the hull model it is possible to define the bevelling along the outer contour of the parts. In the splitting process this information is transferred to the resulting plate parts.

In the Nesting System, the bevel information is available for inspection (in SHOW PRODUCTION INFORMATION) and for insertion into the burning sketch (in PRODUCTION INFORMATION).

If the logical variable SBH_BEVEL_CTRL is assigned the extended bevel handling is used to retrieve the bevel texts. For more information, see *User's Guide Hull / Setup and Customisation / Bevel Excess and weld / Bevel Handling*.

6.3.1 Bevel Information in Small Parts

For small parts the bevel text can be exchanged with a symbol. This is controlled by a default MAX_SIZE_WELD_SYMBOL that is set to the maximum part area where bevel symbols are to be placed.

The bevel symbols to be used are specified in a text file and the full file name must be defined in the environment variable SBH_BEVSYMB_EXCH. Different symbols can be define for positive and negative bevel codes.

Format of bevel symbol exchange file:

```
<symbol font>
<bevel code, symbol id>
```

Example:

```
130
210,1
211,2
212,3
-212,4
```

6.4 Quality Code Exchange

A plate part is always created to be of a certain quality (grade). The natural choice is to nest a part on a raw plate with the same quality as the one specified for the part. However, if there is available space on a plate with a different (normally higher) quality it may be allowed to nest the part on that plate.

In AVEVA Marine it is possible to specify the qualities that are allowable to be used for a plate part of a given quality. This is done via a text file in a simple format.

The allowable exchange must be specified individually for each quality. Each quality is corresponded by a "record", starting with the current quality, followed by all the qualities it can be exchanged for, separated by commas (,) and terminated by a semicolon (;). The order within the list is arbitrary.

The complete name of the "exchange file" should be assigned to the environment variable SBH_QUALITY_EXCH. This file may be checked for correctness via the inithull utility.

An example of the layout of the file is given below

.

Example:

```
A27,
      A27,A32,A36,
      D27,D32,D36,
      E27,E32,E36,
      A27Z,A32Z,A36Z,
      D27Z,D32Z,D36Z,
      E27Z,E32Z,E36Z,
      ST44-2,ST44-3,ST52-3,STE285,STE315,STE355;
A32,
      A32,A36,
      D32,D36,
      E32,E36,
      A32Z,A36Z,
      D32Z,D36Z,
      E32Z,E36Z,
      ST52-3,STE315,STE355;
A36,
      A36,D36,E36,A36Z,D36Z,E36Z,ST52-3,STE355;
D36
      D36,E36,D36Z,E36Z,ST52-3,STE355;
```

In general the qualities to be used must be defined in a text file specifying also the density. This file should be assigned to the environment variable SBH_QUALITY_LIST. See the on line documentation *Hull Setup and Customisation, Miscellaneous, Customer Setup of Material Qualities*.

6.5 The Burner Machine Data

The input data of the burning machines used is given in a special language (cf. AVEVA Hull *Basic "Interpretative Language")*. Any number of burning machines can be handled. The complete file name should be given in the environment variable SBH_BURNER_DATA.

The syntax of the BURNER statement is described below.

```
BURNER,<burner name>,<burner no>,<burning type>
     /BURNER_ID=<burner id>
     /BURNER_REF_ID=<burner refid>
     /PREP_MAIN=<prep main>
     /PREP_ADD=<prep add>
     /MARKING=<mark time>
     /MARKING_START=<mark start time>
     /GSD=<gsd time>
     /GSD_START=<gsd start time>
     /BLASTING=<blast time>
     /BLASTING START=<blast start time>
     /LABELLING=<label time>
     /LABELLING START=<label start time>
     /PARALLEL=<direction>
     /IDLE_FAST=<idle time fast>
     /IDLE_SLOW=<idle time slow>
     /BEVEL=(<bevel_type>,<bevel_typ1>,<bevel_typ2>,...,<bevel_typ10>
     /BURNING=(<thick 1>,<thick 2>,<burn time>[,<bevel_type>)
     /STARTING=(<thick 1>,<thick 2>,<start time>)
     /RATIONALIZATION=(<pretext>,<fixed factor>,<posttext>)
     /TIME_FACTOR=(<pretext>,<factor>,<posttext>)
     /PREP_CONST=(<add const>,<side 1>,<side 2>);
     /PREP_WATER=<prep_wtime>
     /PUNCHING=<punch_time>
     /SIGNING=<sign_time>
     /REMOVING=<remove_time>
     /SCRAP_CONST=<scrap_time>
     /COMMENT=<comment>
     /PROCESS_ID=<process id>
     /DEFAULT_FILE1=(<file 1>[,<thick 1>, <thick 2>])
     /DEFAULT_FILE2=(<file 2>[,<thick 1>, <thick 2>]);
```

<burner name>	is the name of the burning machine. It has the type string.
<burner no>	is an arbitrary number of type integers.
<burning type>	is an additional text specifying the type of burning, e g. vertical. It has the type string and can be omitted.
BURNER_ID=<burner id>	
<burner id>	identifies which of the nested plates that shall be cut by the burner <burner name>. It has the type string. The <burner_id> is given when a new nest job is initiated and it is stored on the plate.
It is also possible to give BURNER_ID any number of times, e.g. once for each nested plate to be cut with the burner in question.	
BURNER_REF_ID=<burner refid>	
<burner refid>	is used in some functions instead of <burner id>. It has the type string. The same <burner refid> can be given for different <burner id>. This can be necessary when one burning machine is used in several modes (single burning, parallel burning, bevel burning) and different burning data is required in each case.
PREP_MAIN=<prep main>	

	<prep main>	is a decimal giving the main preparation time in minutes. It shall only be used once for each burner.
PREP_ADD=<prep add>		
	<prep add>	is a decimal giving the additional preparation time in minutes. It shall only be used once for each burner.
PARALLEL=<direction>		
	<direction>	is a string giving the parallel direction.
MARKING=<mark time>		
	<mark time>	is a decimal giving the marking time in minutes/m.
MARKING_START=<mark start time>		
	<mark start time>	is the marking time in minutes.
GSD=<gsd time>		
	<gsd time>	is a decimal giving the gsd marking time in minutes/m.
GSD START=<gsd start time>		
	<gsd start time>	is the gsd start time in minutes.
BLASTING=<blast time>		
	<blast time>	is a decimal giving the blasting time in minutes/m.
BLASTING_START=<blast start time>		
	<blast start time>	is the blasting start time in minutes. It is applied twice, one time for the starting and one time for the end of the blasting
LABELLING=<label time>		
	<label time>	is a decimal giving the labelling time in minutes/ character.
LABELLING START=<label start time>		
	<label start time>	is a decimal giving the label start time in minutes.
IDLE_FAST=<idle time fast>		
	<idle time fast>	is a decimal giving the fast idle time in minutes/m.
IDLE_SLOW=<idle time slow>		
	<idle time slow>	is a decimal giving the slow idle time in minutes/m.
BEVEL= (<bevel type>,<bevel typ 1>, <bevel typ2>,...,<bevel typ10>)		
	<bevel type>	is a string identifier common for the bevel types (<bevel typ 1>, <bevel typ2>,...,<bevel typ10>). The bevel types can be given either as an enumerated list or as a range. At most 10 bevel types can be given. It is possible to mix the enumerated list and the range.
BURNING=(<thick 1>,<thick 2>, <burn time>[,<bevel_type>])		
	<thick 1>	is a decimal giving the beginning of the thickness interval in which the burning time <burn time> is valid.
	<thick 2>	is a decimal giving the corresponding ending of the thickness interval.
	<burn time>	is the burning time in minutes/m for the thickness interval

	[<thick 1>,<thick 2>]	
	<bevel_type>	defines the bevel type(s) for which the BURNING attribute data are valid.
BURNING is repeated as many times as required to cover the capacity of the burning machine.		
If BEVEL has been defined, one set of BURNING attributes must be given for each <bevel type>.		
STARTING=(<thick 1>,<thick 2>, <start time>)		
	<thick 1>	is a decimal giving the beginning of the thickness interval the starting time <start time> is valid.
	<thick 2>	is a decimal giving the corresponding ending of the thickness interval.
	<start time>	is the starting time in minutes for the thickness interval
	[<thick 1>,<thick 2>]	
	STARTING is repeated as many times as required to cover the capacity of the burning machine.	
RATIONALIZATION=(<pretext>,<fixed factor>,<posttext>)		
RATIONALIZATION is used to present a summary of the path times using different rationalization factors.		
	<pretext>	is a string to be written before the summary.
	<fixed factor>	is a decimal factor which is always applied to the path time.
	<posttext>	is a string to be written after the summary.
TIME_FACTOR=(<pretext>,<factor>, <posttext>)		
TIME_FACTOR is used to present additional summaries of the path times using different rationalization factors. It can be used with at most three different factors.		
	<pretext>	is a string to be written before the summary.
	<factor>	is a decimal factor which is applied to the path time resulting from RATIONALIZATION.
	<posttext>	is a string to be written after the summary.
PREP_CONST=(<add const>,<side 1>,<side 2>);		
	<add const>	is an additional constant due to some surface treatment that must be given to one or both sides. It is given in min/nested plate.
	<side 1>	is a string code for the surface on "this side".
	<side 2>	is a string code for the surface on "the other side".
PREP_WATER=<prep_wtime>		
	<prep_wtime>	is a decimal giving the water cleaning time in minutes. It shall only be used once for each burner.
PUNCHING=<punch_time>		
	<punch_time>	is a decimal giving the punching time in points/minute.
SIGNING=<sign_time>		
	<sign_time>	is a decimal giving the sign factor.
REMOVING=<remove_time>		
	<remove_time>	is a decimal giving the remove factor.

SCRAP_CONST=<scrap_time>		
	<scrap_time>	is a decimal giving the scrap factor.
COMMENT=<comment>		
	<comment>	is a comment displayed in the burner id selection window.
PROCESS_ID=<process id>		
	<process id>	is the process id which is needed for some types of burning machines.
DEFAULT_FILE1=(<file 1>[,<thick 1>, <thick 1>])		
	<file 1>	is the name of the superior default file without file path.
	<thick 1>	is a decimal giving the beginning of the thickness interval the default file <file 1> is valid.
	<thick 2>	is a decimal giving the corresponding ending of the thickness interval.
DEFAULT_FILE2=(<file 2>[,<thick 1>, <thick 2>])		
	<file 1>	is the name of the inferior default file without file path.
	<thick 1>	is a decimal giving the beginning of the thickness interval the default file <file 2> is valid.
	<thick 2>	is a decimal giving the corresponding ending of the thickness interval.

6.5.1 Examples

Below are examples of input of burning machine data.

EXAMPLE 1:

Example:

```
BURNER , 'TELEREX' , 1 , 'VERTICAL'
/BURNER_ID='C'
/BURNER_ID='J'
/BURNER_ID='SB242'
/PREP_MAIN=21.0
/PREP_ADD=9.5
/MARKING=0.094
/IDLE_FAST=0.094
/BURNING=(  3.0 , 15.0 , 2.04)
/BURNING=( 15.1 , 20.0 , 2.23)
/BURNING=( 20.1 , 25.0 , 2.50)
/BURNING=( 25.1 , 30.0 , 2.84)
/BURNING=( 30.1 , 35.0 , 3.23)
/BURNING=( 35.1 , 40.0 , 3.82)
/BURNING=( 40.1 , 45.0 , 4.15)
/BURNING=( 45.1 , 50.0 , 4.68)
/BURNING=( 50.1 , 55.0 , 5.25)
/BURNING=( 55.1 , 60.0 , 5.87)
/STARTING=(  3.0 , 15.0 , 1.15)
/STARTING=( 15.1 , 20.0 , 1.43)
/STARTING=( 20.1 , 25.0 , 1.72)
/STARTING=( 25.1 , 30.0 , 2.01)
/STARTING=( 30.1 , 35.0 , 2.24)
/STARTING=( 35.1 , 40.0 , 2.53)
/STARTING=( 40.1 , 45.0 , 2.82)
/STARTING=( 45.1 , 50.0 , 3.10)
/STARTING=( 50.1 , 55.0 , 3.39)
/STARTING=( 55.1 , 60.0 , 3.68)
/RATIONALIZATION=(  , 0.967 , ' hours')
/TIME_FACTOR=('1 plate part:' ,1.075, ' hours')
/TIME_FACTOR=('N plate parts:' ,1.220, ' hours')
/PREP_CONST=( 0.133 , 'B' , 'D')
/COMMENT='TELEREX VERTICAL only'
/PROCESS_ID='not relevant'
/DEFAULT_FILE1=('sbh_nest1_def1.def'. 0, 14.9)
/DEFAULT_FILE2=('sbh_nest1_def2.def'. 0, 14.9)
/DEFAULT_FILE1=('sbh_nest2_def1.def'. 15.0, 25.0)
/DEFAULT_FILE2=('sbh_nest2_def2.def'. 15.0, 25.0);
```

EXAMPLE 2:

Example:

```
BURNER, 'TELEREX, 2, 'BEVEL'
/BURNER_ID='BEV'
/PREP_MAIN=21.0
/PREP_ADD=9.5
/PARALLEL = -Y
/MARKING=0.09
/MARKING_START=0.11
/GSD=0.010
/GSD_START=0.11
/BLASTING=0.09
/BLASTING_START=0.11
/LABELLING=0.02
/LABELING_START=0.11
/BEVEL=( 'I', 0, 10, 15)
/BEVEL=( 'Y', 20:35)
/BEVEL=( 'X', 40:55::3, 41)
/BURNING=( 3.0,15.0,2.04, 'I')
/BURNING=(15.1,20.0,2.23, 'I')
/BURNING=(20.1,25.0,2.50, 'I')
/BURNING=(25.1,30.0,2.84, 'I')
/BURNING=(30.1,35.0,3.23, 'I')
/BURNING=(35.1,40.0,3.82, 'I')
/BURNING=(40.1,45.0,4.15, 'I')
/BURNING=(45.1,50.0,4.68, 'I')
/BURNING=(50.1,55.0,5.25, 'I')
/BURNING=(55.1,60.0,5.87, 'I')
/BURNING=( 3.0,15.0,2.50, 'Y')
/BURNING=(15.1,20.0,2.84, 'Y')
/BURNING=(20.1,25.0,3.23, 'Y')
/BURNING=(25.1,30.0,3.82, 'Y')
/BURNING=(30.1,35.0,4.15, 'Y')
/BURNING=(35.1,40.0,4.68, 'Y')
/BURNING=(40.1,45.0,5.25, 'Y')
/BURNING=(45.1,50.0,5.87, 'Y')
/BURNING=(50.1,55.0,6.12, 'Y')
/BURNING=(55.1,60.0,6.70, 'Y')
/BURNING=( 3.0,15.0,3.23, 'X')
/BURNING=(15.1,20.0,3.82, 'X')
/BURNING=(20.1,25.0,4.14, 'X')
/BURNING=(25.1,30.0,4.75, 'X')
/BURNING=(30.1,35.0,5.33, 'X')
/BURNING=(35.1,40.0,5.92, 'X')
/BURNING=(40.1,45.0,6.19, 'X')
/BURNING=(45.1,50.0,7.00, 'X')
/BURNING=(50.1,55.0,9.21, 'X')
/BURNING=(55.1,60.0,10.0, 'X');
```

6.6 The Nesting Filter File

In addition to the standard filter used when plate parts are selected for nesting (name, thickness and quality) a number of additional criteria can be defined in a file using a special language (cf. Tribon M3 Basic "Tribon Interpretative Language"). The file name should be given in the environment variable SBH_NEST_FILTER. The file is supposed to be available in the directory SBH_PRODUCTION_SETTINGS or, if this is not defined, in the directory SB_SHIP.

All INCLUDE/EXCLUDE statements below are mutually exclusive.

```
INCLUDE_ASSEMBLY, <assembly>;
EXCLUDE_ASSEMBLY, <assembly>;
```

These statements are used to include/exclude parts belonging to a specific assembly. <assembly> is the name of the assembly at the lowest level. The statements can be repeated to include/exclude several assemblies.

```
INCLUDE_BEVEL,<bev_typ1>,<bev_typ2>, ..., <bev_typ5>;
EXCLUDE_BEVEL,<bev_typ1>,<bev_typ2>, ..., <bev_typ5>;
```

These statements are used to include/exclude parts with specific bevel types. 'None', 'I', 'V', 'Y', and 'X' are supported. At most 5 bevel types can be given.

```
INCLUDE_PARTS/PLANAR
             /DEVELOPED
             /BRACKET
             /STANDARD_BRACKET
             /JIG
             /BENDING_TEMPLATE
             /CLIP
             /CHAMFER_PART
             /CONVERTED_PROFILE
             /DOUBLING_PLATE;
EXCLUDE_PARTS/PLANAR
             /DEVELOPED
             /BRACKET
             /STANDARD_BRACKET
             /JIG
             /BENDING_TEMPLATE
             /CLIP
             /CHAMFER_PART
             /CONVERTED_PROFILE
             /DOUBLING_PLATE;
```

These statements are used to include/exclude parts of a specific type.

```
INCLUDE_POSNO,<part_type>, <posno1>,<posno2>, ..., posno12>
              /MIN_POSNO=<min_posno>
              /MAX_POSNO=<max_posno>;
EXCLUDE_POSNO,<part_type>, <posno1>,<posno2>, ..., posno12>
              /MIN_POSNO=<min_posno>
              /MAX_POSNO=<max_posno>;
```

These statements are used to include/exclude parts with specific position numbers. The part types PLANAR, DEVELOPED, BRACKET, STANDARD_BRACKET, JIG, BENDING_TEMPLATE, CLIP, CHAMFER_PART, CONVERTED_PROFILE and DOUBLING_PLATE are supported. At most 12 position numbers can be given.

The attributes MIN_POSNO and MAX_POSNO can be used to specify an interval of position numbers. Only one of them needs to be given.

```
INCLUDE_QUALITY,<qual1>,<qual2>, ..., qual12>;
EXCLUDE_QUALITY,<qual1>,<qual2>, ..., qual12>;
```

These statements are used to include/exclude parts with specific quality. At most 12 qualities can be given.

```
INCLUDE_THICKNESS,<thick1>,<thick2>, ..., thick12>
              /MIN_THICK=<min_thick>
              /MAX_THICK=<max_thick>;
EXCLUDE_THICKNESS,<thick1>,<thick2>, ..., thick12>
              /MIN_THICK=<min_thick>
              /MAX_THICK=<max_thick>;
```

These statements are used to include/exclude parts with specific thickness. At most 12 thicknesses can be given.

The attributes MIN_THICK and MAX_THICK can be used to specify an interval of thicknesses. Only one of them needs to be given.

```
FILE/NAME=<file_name>
              /BROWSE=<file_path>
              /LINES=(<name_pos>,<no_lines>)
              /FORMAT=(<position>,<length>;
```

This statement can be used if an external file with plate part names exist. <file_name> is the name of the file. It is also possible to only specify the file path using the attribute /BROWSE. The file can then be selected from a file open dialogue.

In the file, the data can extend over several lines. <name_pos> is the line number where the part name can be found and <no_lines> is the number of lines for each part. <position> is the position of the first part name character in <name_pos> and <length> is the length of the part name.

```
MAXIMUM_SIZE,<max_size>;
```

This statement can be used to set the minimum area of the selected parts. The area should be given in m^2.

```
MINIMUM_DIM,<min_x>,<min_y>);
```

This statement can be used to set the minimum dimensions of the selected parts. The true circumscribed rectangle is used. The dimensions should be given in mm.

6.6.1 Example

Example:

```
INCLUDE_PARTS /PLANAR
              /DEVELOPED
              /BRACKET
              /CLIP;
INCLUDE_THICKNESS /MIN_THICK=5.0
                  /MAX_THICK=99;
EXCLUDE_POSNO, DEVELOPED /MAX_POSNO=10;
INCLUDE_POSNO, PLANAR, 10, 20, 30;
```

6.7 Forms in Nesting

The burning sketch in Nesting is a production drawing and thus a drawing form can be applied. In addition to Drafting standard features, Nesting forms have a set of replace texts that can be applied. In addition to that there are some information that is controlled by default settings, such a drawing can contain bevel and excess information.

When inserting form the burning sketch will be scaled to fit the form, this is done automatically depending on the setting of the default AUTOSCALE_SKETCH.

6.7.1 Automatic Production Information added to Burning Sketch

When adding a form to the burning sketch a number of production information items can be added automatically to the sketch as texts. By setting appropriate defaults the following can be added:

- Bevel texts
- Excess information
- Side information
- Cutting sequence text
- Swedging texts, knuckle line texts and folded flange texts

These are controlled by the following default values:

- AUTO_BEVEL_INFO
- AUTO_EXCESS_INFO
- AUTO_SIDE_INFO

- AUTO_CUTSEQ_INFO

For text at swedging and knuckle lines the following defaults apply:

- SWEDGE_TEXT
- KNUCKLE_ANGLE
- KNUCKLE_TEXT
- MARK_TEXT

6.7.2 Nesting Replace Texts

For nesting a set of replace texts has been assigned in the interval 3000-3999. These can be added to the form according to Drafting rules for replace texts. Below they are preceded by a $-sign which means that they will be replaced by one text string. @ or & indicates that the text is a table item and subsequently will be replaced by several values.

For general information on rules and dynamic texts, see *Drafting_Model Viewing and General Drafting_Drawing Forms and Rule*.

The replace texts for nesting are:

Interval 10001-10999, to be defined via vitesse. See *Developer's Toolkit - Vitesse - User Defined Simple Rules in Forms*.

$3000 - Name of nesting job.

$3001 - Name of used standard plate.

$3002 - Name of superior default file.

$3003 - Name of inferior default file.

$3004 - Purchase information.

$3005 - Nesting date.

$3006 - Production date.

$3007 - Postprocessing date.

$3008 - Start position (x,y).

$3009 - Plate quality code.

$3010 - Plate density.

$3011 - Raw plate thickness.

$3012 - Raw plate length.

$3013 - Raw plate width.

$3014 - Raw plate area.

$3015 - Raw plate weight.

$3016 - Used plate length.

$3017 - Used plate width.

$3018 - Used plate area.

$3019 - Used plate weight.

$3020 - Rest plate length.

$3021 - Rest plate width.

$3022 - Rest plate area.

$3023 - Rest plate weight.

$3024 - Classification code.

$3025 - Classification length.

$3026 - Classification width.

$3027 - Classification area.

$3028 - Classification weight.

$3029 - Number of plate parts.

$3030 - Total area of plate parts.

$3031 - Total weight of plate parts.

$3032 - Brutto scrap.

$3033 - Scrap on used plate.

$3034 - Waste area.

$3035 - Waste weight.

$3036 - Number of burning starts.

$3037 - Total burning length.

$3038 - Total burning idle length.

$3039 - Number of marking starts.

$3040 - Total marking length.

$3041 - Total marking idle length.

$3042 - Plate charge.

$3043 - Plate preparation, this side.

$3044 - Plate preparation, other side.

$3045 - Total number of plates to be burned.

$3046 - Number of plates, normal burning.

$3047 - Number of plates, mirrored burning.

$3048 - Scale of drawing form.

$3049 - Drawing number.

$3050 - Ship number.

$3051 - Operator.

$3052 - Restplate name.

$3053 - Counting number.

$3054 - Min position number.

$3055 - Total idle length.

$3056 - Burner id.

$3057 - Side table in 3-axis.

$3058 - Assembly id lowest level, used in part table. If the logical variable TB_USE_PART_ASSEMBLIES is set to an arbitrary value the assembly information is retrieved from the plate part object. Otherwise it is retrieved from the panel object. / Assembly on lowest level in 3 axis.

$3059 - Burned as shown, used in the part table.

$3060 - Burned mirrored, used in the part table.

$3061 - Page (sketch) nr/total nr of pages, used in the part table.

$3062 - AVEVA Marine name, used in the part table. / Part name in 3-axis.

$3063 - Part position number, used in the part table. / Position number in 3-axis.

$3064 - Block number, used in the part table. / Block number in 3-axis.

$3065 - Number of blasting starts.

$3066 - Total blasting length.

$3067 - Total blasting idle length.

$3072 - Total burning length for I-bevel.

$3073 - Total burning length for other bevels.

$3074 - Number of GSD-starts.

$3075 - Number of signing-starts.

$3076 - Length of I-bevel burning. Individual plates.

$3077 - Length of I-bevel burning. Individual plates.

$3078 - Length of I-bevel burning. Individual plates.

$3079 - Initial distance between burner 1 and 2, only 3 axis

$3080 - Total burning length for Y-bevel, only 3 axis

$3081 - Total burning length for X-bevel, only 3 axis

$3082 - Same as $3057 but mirrored, only 3-axis.

$3083 - Same as $3058 but mirrored.

$3084 - Same as $3062 but mirrored, only 3-axis.

$3085 - Rolling information, this side or other side. Table value for portside parts. Only used in 3-axis.

$3086 - Same as $3085, but for starboard.

$3087 - The process name.

$3088 - A flag indicating if parallel nesting is possible.

$3089 - The total path time in hours.

$3090 - Nesting mode (2-axis or 3-axis), only 3-axis.

$3091 - Number of starts I-bevel.

$3092 - Number of starts Y-bevel.

$3093 - Number of starts X-bevel.

$3094 - Length of manual bevel cutting.

$3095 - Number of manual bevel cutting intervals.

$3096 - Length of chamfer cutting.

$3097 - Number of chamfer cutting intervals.

$3098 - Length of excess 1 cutting.

$3099 - Number of excess 1 cutting intervals.

$3100 - Length of excess 2 cutting.

$3101 - Number of excess 2 cutting intervals.

$3102 - Length of excess 3 cutting.

$3103 - Number of excess 3 cutting intervals.

$3104 - Length of edge grinding.

$3105 - Number of edge grinding intervals.

$3106 - Number of starts I-bevel for parallel burning.

$3107 - Number of starts Y-bevel for parallel burning.

$3108 - Number of starts X-bevel for parallel burning.

$3109 - Part id, used in the part table.

$3110 - Cutting sequence, used in the part table.

$3111 - Total length, I-bevel.

$3112 - Total length, Y-bevel TS.

$3113 - Total length, Y-bevel OS.

$3114 - Total length, X-bevel TS.

$3115 - Total length, X-bevel OS.

$3116 - C/M code (consequent/mirrored), used in the part table.

$3117 - Number of identical parts, used in the part table.

$3118 - Part with Folded flange/Curved stiffener, used in the part table.

$3119 - Steel quality code, used in the part table.

$3120 - Plate thickness, used in the part table.

$3121 - Minimum needed raw plate length.

$3122 - Minimum needed raw plate width.

$3123 - Hole data (type, dimension, possible curve name).

$3126 - Same as $3109 but mirrored.

Furthermore, two $-values are used to position a nesting sketch in form:

$3998 - Positioned in lower left corner of available drawing area. All other values ignored.

$3999 - Ditto upper right corner.

Below is an example of a drawing form with some of the $-texts mentioned above.

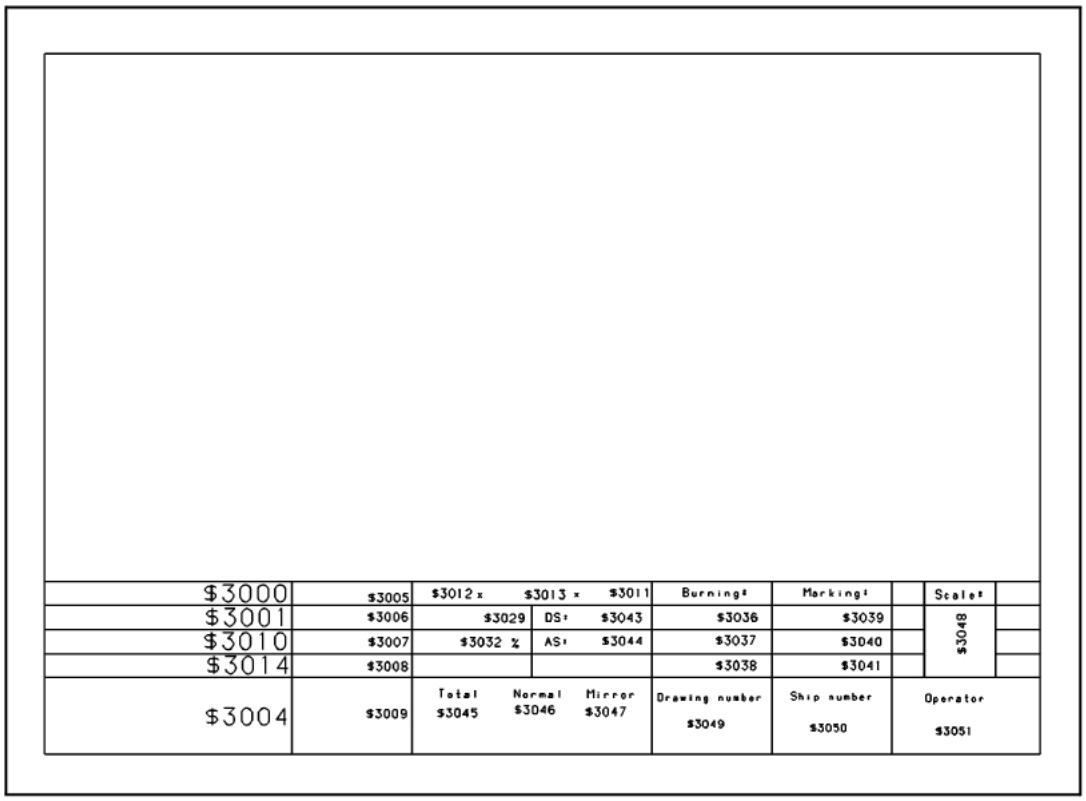


Figure 6:1. Figure Showing an Example of a Drawing form with some of the $-texts mentioned above.

7 Nesting System - Working Routines

In this section, the start-up and close-down of the system will be described and some notes on a normal working session, where a plate is nested, will be given.

7.1 Start-up of the System

The first function to be actuated should be either OPEN NESTING JOB or NEW NESTING JOB.

7.2 Close-down of the System

The session at the workstation is terminated by choosing the EXIT SYSTEM.

7.3 Nesting a Plate

The operations needed when a plate is nested may differ a lot at different shipyards. The following operation sequence is just a small example of how it may be done.

1. Choose the NEW NESTING JOB function and define the name of the resulting nested plate and the dimensions of the raw plate.
2. Choose the NEST function and give the name of a plate part to nest.
3. Position the plate part using the mouse. Press the MB1 order to release the part from the cursor.
4. Use the various transformation functions to position the plate part in its final position.
5. Choose OPERATION COMPLETE when the part is properly positioned. The system will then ask for the name of the next plate part. An empty name will finish the NEST function, If a new name is given, continue at 3.
6. Now all the plate parts should have been properly nested. If burning bridges should be used, choose the BRIDGE function, the proper bridge types and position the bridges. If gaps are used for burning starts, position the gaps as well.
7. If the plate parts contain marking lines, define the marking sequence with the function START MARKING.
8. Choose the START BURNING function to define the burning sequence.
9. Finally, the tool path can be verified using the VERIFY function. Now the nesting is ready and the postprocessor can be run to produce NC control information. The plate can be stored using the STORE PLATE function.
10. To produce a burning sketch, use the appropriate sketch functions (DISPLAY SKETCH, DISPLAY NESTED PLATE, ADD FORM, TEXT, PRODUCTION INFO etc.) This should be done before the plate is stored.

When an existing plate is to be modified, the function OPEN NESTING JOB should be used. This will make the plate appear on the display with the latest versions of all plate parts. It is, in fact, renested automatically.

Manufacturing Data for 3-axis Burning Equipment

1 General

This document describes the module in which one or several developed plates are laid out on a raw plate to become a nested plate. It also describes how these nested objects are used to produce manufacturing data for 3-axis burning equipment.

The module can be divided into a number of processes as described below.

1.1 Input

Input to the program is based on the general selection tool and this input is normally generated automatically when activating this function through the interactive nesting application via a production program interface. The selection possibilities are described in *Manufacturing, Hull Production Program Interface, General Selection Tool*.

1.2 Nesting (or Combining) Developed Plates

Nesting is a task where a number of developed plates (optionally only one) are placed together on a specified raw plate. This process is dependent on parameters given by the user in a restriction file (see below). The result of the nesting process is a nested object.

1.3 Processing Nested Objects

A process carries out work related to the presentation of manufacturing data such as burning sketches, roll-axis information, etc. Additionally it produces a ‘non-AVEVA Marine’ formatted output file (generic file) containing all information necessary to produce a 3-axis N.C. file.

1.4 3-Axis Post-processor

The system contains a module that generates N.C. data from the generic file in a suitable format for 3-axis burning equipment.

2 Environment

2.1 Set-up of Program

The name of the executable of this program is sf623d. It communicates via an input file and resulting files. The program is normally activated through the Job Launcher (JL) where the following set-up is required:

Name recognised by JL: **Nesting 3-Axis**

AVEVA Marine logical	JL Set-up and Explanation
SB_INPUT1	Input file to be set-up with extension.dat in JL.
SB_OUTPUT1	Output file with run-time information. To be set-up in JL as first output file with extension.log.
SB_OUTPUT2	Output file with names of created nestings.
SB_OUTPUT3	Output file with names of created nestings

Name	Description
SB_PLDB	Data bank containing developed plates
SBH_3AX_SKETCHDB	Data bank for storage of burning sketches and drawings with roll axis information. For more information about drawings stored in PADD data base see below.
SB_DEFSTD	Data bank containing drawing forms used for burning sketches and standard plates.
SB_DEFNPL	Data bank for storage of nested objects.
SBH_3AX_RESTRICT	File name of restriction file.
SBB_USER_SIGNATURE	Signature to be added to burning sketches.
SBH_BURNER_DATA	Burner data file.
SBH_BURNER_ID	Burner identification.
SBH_HOOK_SELECTION	Hook selection file for holes.

3-axis nesting drawings are stored in the PADD data base pointed out by SBH_3AX_SKETCHDB and SBH_3AX_SKETCHDB_PADD. The logical name SBH_3AX_SKETCHDB should point to it self and for the SBH_3AX_SKECTDB_PADD pointing out where these drawings are stored in PADD. See the *2D Drawing / Hull Drafting /*

Model Viewing and General Drafting / Concepts User Guide for setting up the department and registry field of SBH_3AX_SKETCHDB_PADD reference.

If the logical name SBH_3AX_SKETCHDB_PADD is not assigned the software will try to store the drawings in PADD where the standard drawings are stored (SB_PDB_PADD).

In addition, standard logical names such as SB_SHIPPRINT, SB_TAPE etc. must be assigned. They are normally defined within a project and are not further dealt with here.

3 Creating Nestings and Producing Manufacturing Data

3.1 The Restriction File

Default and restriction parameters are defined by the customer in an ASCII-file assigned to the logical variable SBH_3AX_RESTRICT. The file is keyword oriented and can be prepared using any available text editor.

The contents of the file are divided into a general information section, valid for plates of all thicknesses, and one or more sections that are thickness dependent. If there is no requirement for different parameter values for different plate thicknesses, then the thickness variable in the restriction file should be set to a very high value. A value of i.e. 9999.0 will cover all thicknesses.

Keywords in the restriction file are:

FORM_BURN=<Name of burning sketch form to be used>

Burning sketch forms are explained in *Drawing Forms* below. There is no default value for this keyword.

FORM_ROLL=<Name of roll axis sketch form to be used>

Roll axis sketch forms are explained in *Drawing Forms* below. There is no default value for this keyword.

MARK_CHANGE=NONE/VMARK/HOOKS

By specifying VMARK, a V-shaped jag showing material side will be added to all marking lines.

By specifying HOOKS, all marking lines will be converted to a number of hooks, composed of a short line and arc (similar to a lying question mark).

By specifying NONE, marking lines will not be modified.

If the keyword is omitted, the value NONE is used by default.

NAME_SUFFIX=NONE/`<suffix>`

This keyword is used when generating names of resulting sketches, as explained in *Results* below. The value NONE is interpreted as no suffix and is the default.

TOTAL_NUMBER=YES/NO

Specifies whether the total number of sketches is to be added to the sketch names as shown in *Results* below. The default value is NO.

PARTID_FONT=`<font number>`

Number of font from which the symbol used for part identification is fetched. Default value is 2.

PARTID_SYMBOL=<symbol number>

Symbol number to be used for part identification. Default value is 61.

PARTID_TYPE=FULL/SHORT

Type of part identification to be presented. FULL refers to the part identification defined by the logical TB_PARTNAME_FULL whereas SHORT refers to TB_PARTNAME_SHORT. See *Hull / Setup and Customisation / General / Customer Control of Part Names* for further explanation. SHORT is the default value if the keyword is not given.

MARK_PANEL_PREFIX=<text>

A text to be added as prefix to all marked panel names on the produced sketch. Default is an empty string.

LONG_PREFIX=<text>

A text to be added as prefix to all longitudinal identifications on the produced sketch. Default is an empty string.

LONG_FONT=<font number>

Number of font from which the symbol used for longitudinal part identification is fetched. Default value is 2 and value 0 is interpreted as no symbol wanted.

LONG_SYMBOL=<symbol number>

Symbol number to be used for longitudinal part identification. Default value is 1 and value 0 is interpreted as no symbol wanted.

DISPLAY_REST=YES/NO/YES_AND_STORE

This keyword specifies whether rest plate information should be displayed on the burning sketch and also if special steel rest plates should be stored as rest plates.

NO is the default value.

If YES is used, information about rest material will be displayed on the sketch, but no special steel rest plates will be stored.

If YES_AND_STORE is used, special steel rest plates will also be stored in case a rest plate for the actual quality and thickness is available. If special steel rest plates have been created, the recreation of a nest will be handled in the following way:

If the involved parts occupy equal or less space, the recreation will be performed and the resulting sketch will contain information about rest material.

If the involved parts occupy larger space on the plate, a message will be given to the user and no recreation of the nesting is done. In this case the nesting must be deleted and the parts nested once again.

MIN_SST_AREA=<m2>

Minimum area of special steel quality rest material that should be used when creating rest plates. Default value is 2.

THICKNESS=<thickness of plate>

All subsequent keywords until the next THICKNESS keyword are valid only for parts with thickness less or equal to the given thickness. THICKNESS keywords and the associated sections of the restriction file should be arranged in increasing order of thickness. If there are no thickness dependent parameters then the thickness should be set to a very high value e.g. 9999 mm (default).

AREA_DIFF=<mm2>

The maximum allowable difference in area of mirrored parts. The default is 100 mm^2.

NUMBER_OF_PARTS=ONE/TWO/MANY

Maximum number of parts allowed to be nested on any one plate. MANY is the default value.

HEAD1_ORIGIN=1/2

Definition of head one's reference position at start of processing.

Value 1 is lower left corner (default).

Value 2 is upper left corner.

HEAD_DX=<mm>

HEAD_DY=<mm>

HEAD1_ORIGIN is modified by these values to locate the exact initial position of head one. The default value for these keywords is 0.

INITDIST_1_2=<mm>

Initial distance between head 1 and 2 as set by the operator before the start of processing. The default value is 1000 mm.

MINDIST_1_2=<mm>

Minimum allowable distance between head 1 and head 2. This value is also used to locate head 2 whenever it is idle. The default value is 700 mm.

IF_2AX=IGNORE/2AX

Instruction what to do when a part cannot be burnt in 3-axis mode, i.e. the width is smaller than the minimum distance or too big slope or length difference between the sides to be burnt in parallel. Two possibilities exist:

By using IGNORE which is the default value the nesting is not performed. The user is instructed in the log to move the part to an NP-file and nest it in normal nesting. Parts belonging to the nesting that are possible to burn in 3-axis mode must be re-nested.

By using 2AX, the part will be burnt with one torch only. This option is, however, not available in the post processor until further development.

SEQUENCE=PARALLEL_LEFT_RIGHT/LEFT_PARALLEL_RIGHT

Keyword used to define the burning sequence. The default sequence is PARALLEL_LEFT_RIGHT i.e. the parallel edges are cut first, followed by the left edge of the plate and then the right.

OVERRIDE_LENGTH=<mm>

For the last part on the plate the burning sequence is always PARALLEL_RIGHT_LEFT unless the length of the remaining plate is greater than this distance. The default value is 1000 mm.

CORNER_LOOP=ROUND/SQUARE

Keyword defines the type of corner loops to be used. The default is SQUARE.

MIN_DIST_CUT=<mm>

Lead in and lead out over plate edge (see the figure below).

The default value is 10 mm.

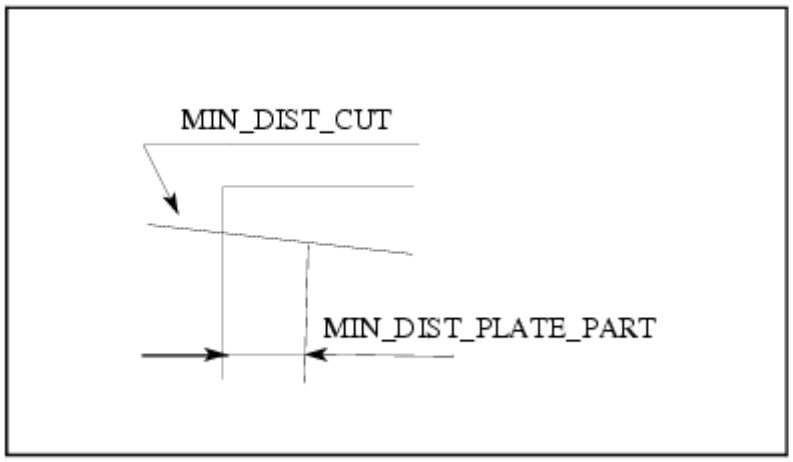


Figure 3:1. Figure Showing the Keywords MIN_DIST_CUT.

MIN_DIST_PLATE_PART=<mm>
Minimum distance from edge of plate to plate part. The default value is 15 mm.

MIN_DIST_BURN_OUT=<mm>
Minimum distance to burn out from a contour.
The default value is 15 mm.

MIN_DIST_PART_HOLE=<mm>
Minimum distance between part and slot created between two parts.
The default value is 30 mm.

HOLE_X_DIST=<mm>
Width of the slot between two parts. The default value is 100 mm.

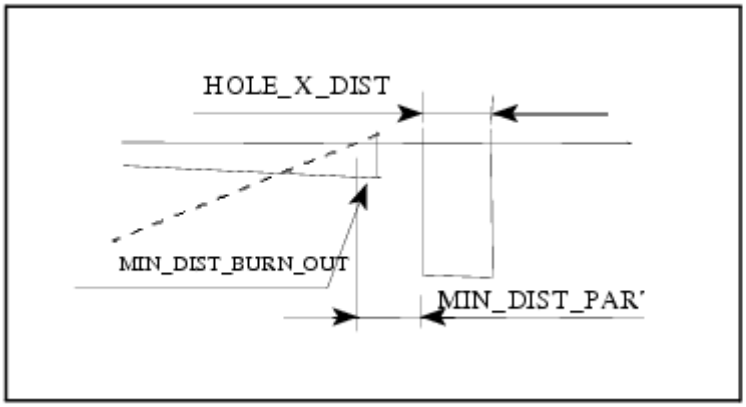


Figure 3:2. Figure Showing the Keywords HOLE_X_DIST, MIN_DIST_BURN_OUT and MIN_DIST_PAR.

HOLE_Y_DIST=<mm>
Minimum depth of the slot between two parts. The default value is 100 mm.

MAX_DIST_LAST_PART=<mm>
Maximum allowed distance between last part and plate edge when burning parallel to plate edge in the X-direction. If the actual distance is greater than this value, then burning out to plate edge is done in the Y-direction. Default value is 200 mm.

In the figure below the bold line with arrows is the value assigned to MAX_DIST_LAST_PART. The upper case shows the burning path when the distance is larger than the limit value, the lower when it is shorter.

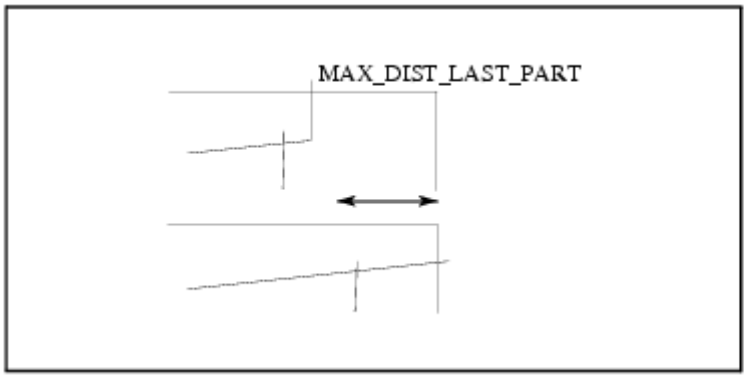


Figure 3:3. Figure Showing the Keyword MAX_DIST_LAST_PART.

MIN_DIST_CORNER=<mm>

The minimum distance between the tangential lead in or lead out of the contour and the plate edge. If this distance can not be maintained, a knuckle is generated on the lead in or lead out. The default value is 15 mm.

RADIUS_CORNER=<mm>

The radius to be used if corner loops have been specified as ROUND (see above). The default radius is 30 mm.

MAX_SLOPE_DIFF=<degrees>

The maximum allowable difference in slope between contours to be burnt in parallel. This is to prevent an overly large difference in cutting speeds between the contours. The default value is 15 degrees.

MAX_LENGTH_DIFF=<%>

The maximum allowable difference in length between contours to be burnt in parallel.

This is to prevent parallel burning of contours with extreme difference in length. The default value is 60% i.e. the difference in length may not be more than 60% of the length of the longest contour.

DISPLAY_RULER=YES/NO

Specifies whether a scale ruler is to be added to the burning sketch. The default is NO.

RULER_DIST=<mm>

Distance from top of raw plate to where ruler is drawn. The default is 15 mm.

TS=<text>

A text string to represent "This Side" on drawings. The default is "TS".

OS=<text>

A text string to represent "Other Side" on drawings. The default is "OS".

TOP=<text>

A text string to represent "Top" on drawings. The default is "TOP".

BOTTOM=<text>

A text string to represent "Bottom" on drawings. The default is "BOTTOM".

AFT=<text>
A text string to represent "Aft" on drawings. The default is "AFT".

FWD=<text>
A text string to represent "Forward" on drawings. The default is "FWD".

PS=<text>
A text string to represent "Port Side" on drawings. The default is "PS".

SB=<text>
A text string to represent "Starboard Side" on drawings. The default is "SB".

CL=<text>
A text string to represent “Centreline” on drawings. The default is "CL".

SHL=<text>
A text string to represent "Shell" on drawings. The default is "SHL".

EXC=<text>
A text string to represent "Excess" on drawings. The default is "EXC.".

DISPLAY_IDLE=YES/NO
Controls if idle movements should be displayed on the burning sketch. Default is YES.

3.2 Hook Selection File

The customer has a possibility to set-up the system to select burning hooks in holes from the ones defined in the nesting system. This set-up is done by a "hook selection file", which is an ordinary ASCII file with the following contents:

```
Hole type   Thickness   Start hook   End hook
```

The selection is done by searching for the correct hole type, e.g. 'D'. If this is found, an exact match of the hole size, e.g. '50' is searched. If the hole size is not matched, the closest smaller hole size, e.g. '30' is used. The thickness of the current plate is then compared to the thickness for the hole type in the hook selection file. If the plate thickness is not matched then the definition with the closest smaller thickness is selected.

Example of a Hook Selection File:

Example:

```
D30      12.5      101      101
D30      18.0      102      102
D30      40.0      103      103
```

If the current plate thickness is 15 mm, the hook 101 is selected. If the current plate thickness is 10 mm, no hook will be selected and the warning:

"No hook found for this hole <type> <size>"

"Check hook selection file (SBH_HOOK_SELECTION)"

A special hook type "-2" is used to for holes that shall not be burnt, but instead marked with a special marking (see below).

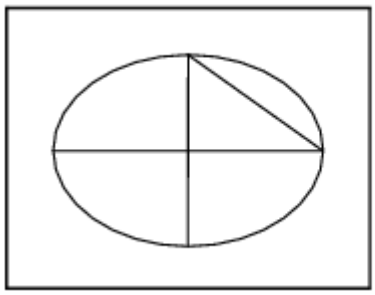


Figure 3:4. Figure Showing Special Marking for Small Holes.

Alternatively such holes will be marked with a cross (see below) if the environment variable SBH_MARK_HOLE_CROSS is set to a real value. If the assigned value is greater than zero both arms of the cross will be drawn with the indicated length. Otherwise the arms will be extended to the hole contour.

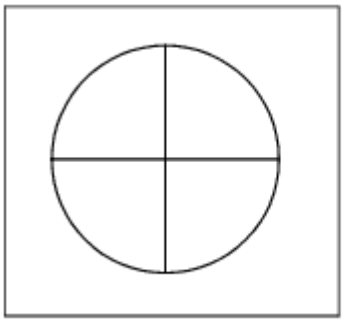


Figure 3:5. Figure Showing Special Marking for Small Holes when SBH_MARK_HOLE_CROSS is defined.

The Hook Selection File is read already in the automatic part generation and the marking is created by this program. The produced sketch will also contain designation of the hole together with a text string defined in ppanparts.

3.3 Burner Data

Burner data and burner identification have to be assigned if path time calculations shall be performed. The logical SBH_BURNER_DATA should be assigned to a burner data file (see *The Burner Machine Data in Chapter Initialisations for Nesting*.

The logical SBH_BURNER_ID should be assigned to an appropriate burner identification code.

3.4 Drawing Forms

The drawing forms to be used should contain the same type of $-values as drawing forms used in the Nesting System for plane parts. Additionally, there are two more values required for the positioning of the nested developed plates on the drawing form. These $-values are 3998 and 3999. The area free for drafting is defined by their position on the drawing form. $-value 3998 shall be placed in the lower left corner of this and $-value 3999 in the upper right corner.

Note: The forms must, however, not be defined in the Nesting System, but in Drafting. When created, they have to be moved manually to the standard data bank used in Nesting.

3.5 Nesting

Nesting is carried out using the job code nest3ax. The nest3ax job code obtains the input information from a user-written input file which is described below.

3.5.1 Input

Input to the system is written by the user, using a input language based on Interactive Language. The actual nesting is performed by the user via this input. The data to be given are the name of the nesting (DNC-ID), the name of raw plate and the developed parts to be nested.

The plates to be nested (developed or planar) are selected via the standard selection tool of Hull. The possibilities of this tool are described in *Manufacturing, Hull Production Program Interface, General Selection Tool*.

The names of the produced nestings are created automatically based on the input from the user. This is further explained in *Nest PPI in Chapter Nesting System - Application Functions*.

The part statement can have an attribute attached, with values /PS, /SB or /BOTH. If not given, /BOTH is assumed. Below is given an example:

Example:

```
CREATE_NEST, '521399';

PLATE, 'XXX';

PART, 'ES123-500-1SP';
```

A nesting can also be recreated. The input file is used to select a previously generated nest. The developed part names and raw plate name are fetched from the nest and the nesting process is re-activated with this data. All result files are produced in the same way as for normal nesting. The following input is needed to recreate a nesting with name 521399:

```
RECREATE_NEST, '521399';
```

The input syntax is defined in the file D0320147.SBD which must be available on SB_SYSTEM.

3.5.2 Results

The log file written by nest3ax contains information about all produced nested plates, including the name of nesting and all produced drawings, the raw plates used, whether a mirror image has been created, etc.

- **Drawings**

  One burning sketch for the nesting and one bending sketch for each part are created for each nesting.

The sketches have the same name as the DNC identity and are differentiated by a running number. The burning sketch is number 1, followed by the bending sketches for the parts in order on the plate. The running number may be separated from the DNC identity by a suffix specified by NAME_SUFFIX in the restriction file. The total number of sketches will be appended to the running numbers if TOTAL_NUMBER is set to YES in the restriction file.

i.e.

```
<DNC ID><suffix from restriction file><running number> <total number of sketches>
```

e.g.

With NAME_SUFFIX = '-' and TOTAL_NUMBER = YES in the restriction file; and with DNC identity equal to 521399, the following names will be assigned to the sketches for a nest with two plate parts:

521399-13	Burning sketch
521399-23	Bending sketch for first part on the plate.
521399-33	Bending sketch for second part on the plate.

- **Generic Data**

The purpose of the generic data is to describe the nesting in a way that is sufficient for production of burner specific processor code. The program producing this code is normally called a post-processor.

A generic file is produced for each nesting. The names of the generic files are written into a separate file, making it possible for the post-processor to interact with the results of the nesting module via this single file.

The format of the generic file is described in a separate document.

3.6 Post-processing

Post-processing is carried out on the produced generic data.

Hull Profile Nesting

1 Profile Nesting

1.1 General

This document describes the process where profiles from the production data banks are combined into nested objects. Furthermore, the handling of raw material for profiles and the restrictions applied when manufacturing the profiles are described in this document.

The processing of the nested objects into results such as robot information, manufacturing lists, etc. is not a part of this product. Information of this type can be found together with the product TBR 10515 - Profile Cutting Interface.

The automatic nesting is performed with given profiles, so that the amount of scrap will be as small as possible.

The systems starts by sorting all profiles on possible way of manufacturing (robot or manual), on type, on dimensions, on quality and finally on length.

After this sorting, the nesting is performed, either on user selected or automatically selected material.

Profiles added to a nested profile will be marked as "nested" and will be disqualified when selecting new profiles with the filter for non-nested profiles is set. The "nested" mark will be removed if the nested profile is deleted. The delete operation must then be done via the menu function "Delete on DB" when running Hull Planar Modelling or Hull Curved Modelling

1.2 Set-up of Program

1.2.1 Set-up in the Job Launcher

The name of the executable of this program is sf605d. It communicates via an input file and resulting files. The program is normally activated through the Job Launcher (JL) where the following set-up is required:

Name recognised by JL: Hull Profile Nesting

AVEVA Marine logical	JL set-up and explanation
SB_INPUT1	Input file to be set up with extension .dat in JL
SB_OUTPUT1	Output file with run-time information. To be set up in JL as first output file with extension .log
SB_OUTPUT2	Output file showing information on nestings and a summary of used material. To be set-up in JL as second output file with extension.lst
SB_OUTPUT3	Output file with comma-separated data about the nested profile

1.2.2 Set-up of the Restriction File

Defaults and restrictions are defined by the user in a normal ASCII-file assigned to the logical variable SBH_PROF_RESTRICT. The file is keyword oriented and can by handled by any editor. A complete description of the restriction file can be found in *Create Profiles in Chapter Nesting System - Application Functions* .

1.2.3 Set-up of Raw Material Data

Raw material data is defined in an ordinary ASCII-file, created by any text editor. The file shall be assigned to SBH_RAW_PROFILES and its syntax is described below:

Item Contents/Declaration

1. The raw profile designation. The identification of the raw profile. It has to be unique within the current run. It is a string of characters with the maximum length of 26 characters.
2. The raw profile type. A code number defining the profile type according to the VIS standards. It is an integer.
3. The number of profile parameters. It depends on the profile type. It is an integer.
4. The profile parameters according to the VIS standards. The number of parameters is given by the previous item (3). They are floating point numbers.
5. The raw profile steel quality. A code number designating the profile steel quality. It is an integer.
6. The length of the raw profile. It is a floating point number.
7. The buying mark of the raw profile. It is a string of characters with the maximum length of 26 characters.
8. The number of available raw profiles in the current run. It is not updated during the run as yet. It is an integer.

Example:

```
RSB09012 10 2 90 12 10 20000 'RAW BUY SB1' 25
RSB12012 10 2 120 12 10 20000 'RAW BUY SB2' 50
```

Materials of the same type, dimension and quality are handled to some extent. At present, up to five materials of different lengths are used when selecting raw material for each profile to be nested.

1.2.4 Use of Quality Exchange

The quality exchange functionality is activated in cases where raw material is specified and the automatic selection of raw material is not used.

1.2.5 Allowing for Profile Rotation.

Default in Profile Nesting is to allow no rotation of the individual profiles to be nested. This can be changed using the environment variable SBH_PROFILE_ROTATION. The following values can be assigned:

NO Default, no rotation allowed.

UD Allows for *Up-Down* rotation.

EE Allows for *End-End* rotation.

UDEE Allows for both *Up-Down* and *End-End* rotation.

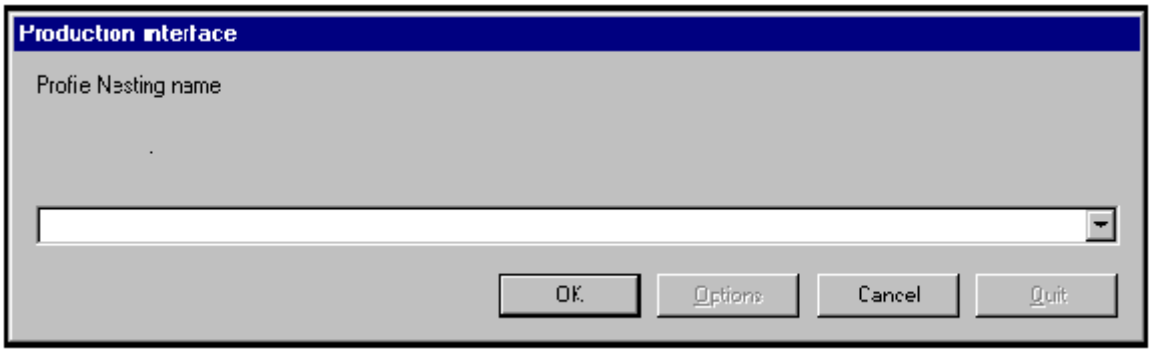
1.3 Input

Input to the program is based on the general selection tool and this input is normally generated automatically when activating this function through any of the interactive hull applications via a production program interface. This interface and the selection possibilities are described in *Manufacturing, General About the Production Program Interface*.

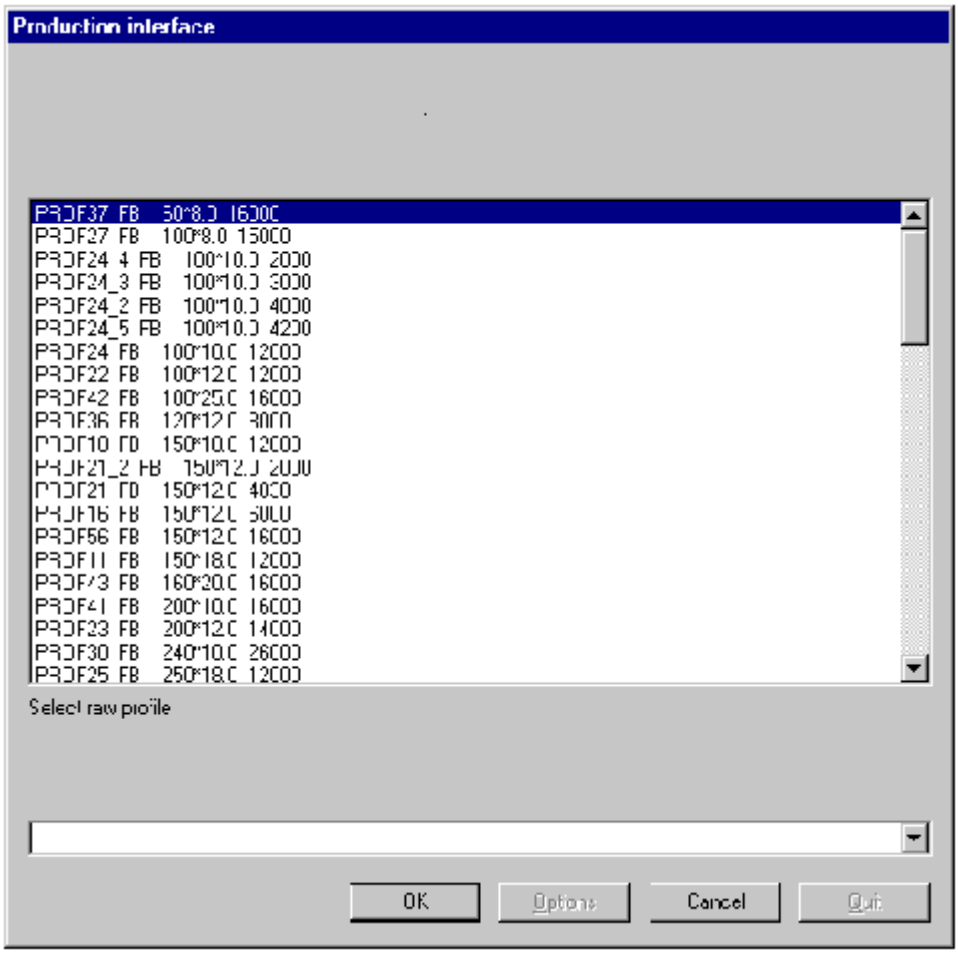
Two different types of selections can be done, either hull objects to be used to extract individual profiles (via panels, assemblies, etc.) or already existing nested profiles. In the latter case, it is assumed that a re-nest is wanted and a control question will be presented asking if this is really the case. The re-nest option is useful when nestings have to be updated due to design changes.

The selection of individual profiles aiming at the creation of new nestings will be followed by a user interaction where data specific to the profile nesting can be added as specified below.

First information to be given by the user is the base name of the nesting results. The nesting objects will be named with this base name concatenated with a two digit running number.



Next, information about raw material shall be given. In case no material is selected (pressing OK directly), an automatic selection of material in the list will be made during the nesting process.



1.4 Output

The result of the nesting process is a nested profile, stored on the data bank assigned to the logical SBH_NEST_PROFDB. These objects can be further processed by the Profile Cutting Interface, producing robot files, sketches and manufacturing lists. The nesting process as such produces the following output:

1.4.1 List File

A list file is produced showing information on nestings and a summary of used material.

- **Example of list file**

  Comes later....

1.4.2 Comma-separated File

The comma-separated files will contain data rows as specified below.

- **Layout of the File**

Term	Term type
Name of the nested profile	String
Burning mark	String
Profile type	Integer
Steel quality code	Integer
Not used	
Profile parameter A	Real
Profile parameter B	Real
Profile parameter S	Real
Profile parameter T	Real
Profile parameter C	Real
Profile parameter U	Real
Total length of raw profile	Real
Used length of raw profile	Real
Rest length of raw profileReal	Real
Running number of profile on the nested profileInteger	Integer
Name of profile	String
Position number of profile	Integer
Marking identification of profile	String
Location of profile (PS/SB,TS/OS)	String
Starting point of profile on the nested profile	Real
Length of profile	Real
End cut type of left endInteger	Integer
End cut type of right end	Integer
Turning/rotation of the profile on the nested profile	String

1.5 Output Drawings

A picture of the nested profile will be created and stored on the data bank assigned to the logical SBH_RECEIPT. The name of the picture will be the same as the nesting name

1.5.1 Example of Drawing

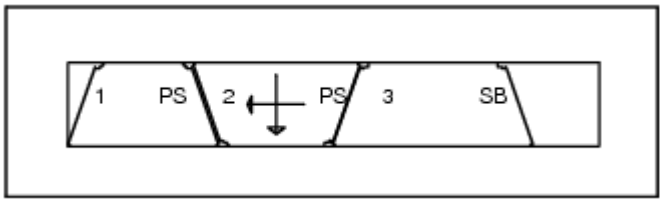


Figure 1:1. Example of Drawing.

Hull Panel Line Control

1 Background

1.1 General

In large ships a considerable portion of the main structure, both internally and in the deck(s) and shell, consists of big flat plate areas, stiffened with profiles. Examples are the flat bottom and side, flat decks, platforms, bulkheads, etc. In normal shipbuilding terminology these structures are called **panels**. Such panels are in major yards produced in specialized production facilities, so called **panel lines**. The purpose of this document is to describe, partly on an overall level, a module developed to support the operation of panel lines.

The use of the word panel in this context is not quite in line with the standard concept "panel". Even if a panel may be identical to a production panel for the panel line it may also be larger, e.g. be built up by several production panels. To keep the two concepts apart this document will use the name **assembly part** for the production unit and thus the concept "panel" will be reserved for the model objects of the Hull application. The only exception is the concept "panel line" in this document and in references to it.

In order not to create confusion and to differentiate them from assembly parts the ordinary individual plate parts are in this document called **piece parts**.

1.1.1 Scope of Panel Line Control Module

An assembly part consists normally of several plates, welded together to a big plate area, and a number of profiles that are used to stiffen the resulting assembly. The most important of these stiffeners are mostly parallel to each other. In addition assembly parts, especially in the internal structure, may contain holes, minor stiffeners around the holes, etc.

A panel line for production of assembly parts consists of several "stations", typically those listed below. However, this may differ between yards depending on the techniques used, the facilities available, etc.

1. A station for preparation of the piece parts that will be welded together to form the assembly part.
2. A station for welding of the assembly part butt joints.
3. A station for cutting to final dimensions and marking up of the assembly part.
4. A station for mounting and welding of the main parallel stiffeners on the assembly part.
5. A station for assembly of the remaining parts (stiffeners, smaller brackets, etc.) and other assemblies.

(Station 2 may comprise of equipment for welding and turning of the assembly part.)

The Panel Line Control Module (in this document called PLCM) contains modules supporting certain of the activities in stations 1, 3 and 4 in this list. These modules can be used rather independently depending on the facilities available in a certain yard.

The software modules involved in PLCM consist of both new programs and of existing modules which have been changed. New programs are:

- the actual PLCM module (which is an interactive program),
- a new post processor.

Existing modules which have been given extended functionality are:

- Hull Modelling,
- PPANPARTS (for extraction of parts from the model).

This document contains a survey of the modules involved in PLCM and notes on their specific use in this context.

PLCM is normally dependant on assembly names, set in the Assembly Planning application.

1.1.2 Modelling for Panel Line

The objects to be passed through the panel line (i.e. the assembly parts) are derived from panels, generated in Hull using the ordinary modelling tools (i.e. Hull Modelling for panels in the internal structure and Curved Panel Generation for shell structures). It is the model in combination with its break down into assemblies that forms the basis for the extraction of the assembly parts for the panel line. Thus, the definition of the assembly parts is a side effect of the assembly definition in Assembly Planning. A panel may be divided into one or several assembly parts, or a panel may be divided into assembly parts and piece parts, the latter to be fabricated as normal individual parts.

An assembly part is composed of all those piece parts within a panel which belong to the same assembly on the lowest level. These piece parts should be contiguous, i.e. possible to combine into one big part.

When generating a panel it is divided into its piece parts by the seams of the panel. Those seams which form the limits between assembly parts are called assembly seams and will automatically be identified as such in PPANPARTS (see below). The only major restriction in generation of panels for the panel line is that **an assembly seam must be an assembly seam along its full extension**. Thus it must not along part of its length be a joint between different assembly parts and along another part be an internal seam in another assembly part. If such a situation should occur the seam must be split into two. (See the figure below showing a panel consisting of three assembly parts "A", "B" and "C". The seam "a", dividing assembly parts "B" and "C" must not continue into assembly part "A". It must be divided into two separate seams "b" and "c".)

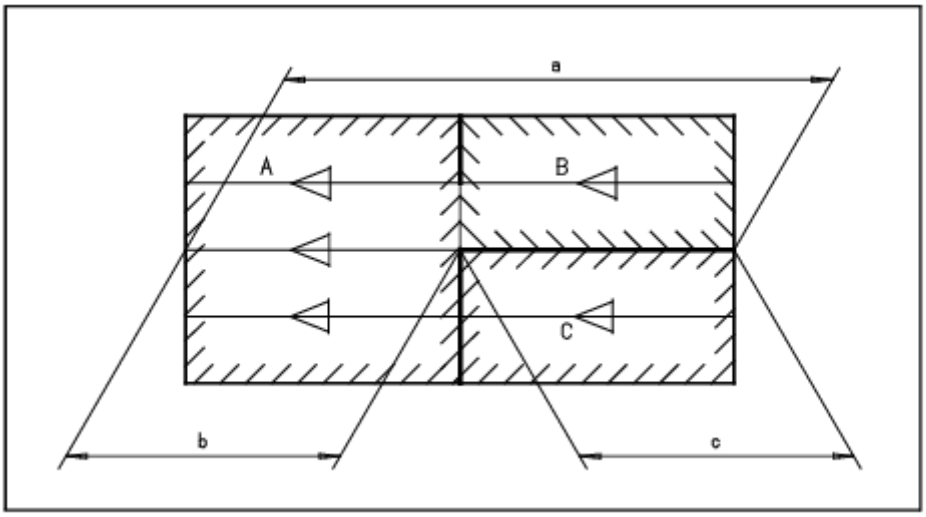


Figure 1:1. Panel.

All piece parts to be included in an assembly part must be generated so that the surface to be turned upwards in the panel line are located in the same plane, i.e. the side of an assembly part turned upwards must be planar.

For use in the panel line supposes that the excess may be defined in three different levels (see *User's Guide Setup and Customisation, Bevel Excess and Weld, Excess and Excess Symbol*s.

The excess on level 3 (specified by the Hull Modelling default parameter `EXC_TYPE_3`) is supposed to be used on piece part level and will be cut off in the panel line, i.e. this excess will not be included in the geometry of the resulting assembly parts (cf. below).

In general, the shape, size and main characteristics of the assembly parts are supposed to be such that they are possible to process in the panel line. This is the responsibility of the designer. The system will only check for violation of the formal conditions of assembly parts.

1.1.3 Extraction of Assembly Parts from Hull Model

Piece parts are extracted from the hull model (the panels) by the automatic part generation module PPANPARTS. The same module is used to extract also the assembly parts from the mode if switched into "panel line mode" by the assignment of a logical name SBH_PANEL_LINE. To this variable should be assigned the full file specification of a file which containing the names of the assemblies (on the lowest level) to be processed in the panel line.

- **Assembly Name File**

  The Assembly Name file is used to specify the names of those assemblies on the lowest level which are supposed to be processed in the panel line.

  The file is an ordinary ASCII (text) file with the following characteristics:

  - It must contain one assembly per text line.
  - Assembly names can be given with the "wild card" symbol replacing one or several groups of characters, and with the %-sign replacing a single character.

Example:

U2*

B2%CA

In this case all parts will be approved with assembly names (on lowest level) starting in "U2" as well as all those which are five characters long, starting in "B2" and ending in "CA".

1.1.4 Ppanparts

PPANPARTS is the module that normally extracts parts from the hull model (both plate and profile parts). A variant of this module has been established to be executed in "panel line mode" (cf. above). When executed in this way the result consists of both **piece parts** and **assembly parts**. In panel line mode PPANPARTS can be used to extract assembly parts also from (almost) planar shell panels, generated in the module for shell panels (CPANGEN).

When determining the shrinkage additions for assembly parts, the first plate thickness (the thickness from the individual plate with lowest attribute number within the part) will by default be used. However, the user has the possibility, via a logical name, to control the thickness to be used. The name of the logical name is:

SBH_PLCM_SHRINK_PLATE

The following three different values are valid:

1. FIRST
   Like today
2. FREQUENT
   The most frequent thickness will be used
3. MEAN
   Take the mean thickness of all individual plates

Curved panels (almost planar ones) are temporarily converted to plane panels to be able to create the assembly parts. When defining the panel plane of this temporary plane panel, PPANPARTS acts in the following way:

1. If the logical name SBH_CPAN_PLANE_PLATE has been defined and points to a plate number, use this plate to defined the nominal panel plane.
2. If not item 1 is fulfilled, get the first "absolute plane" plate with assembly defined.
3. If neither item 1 nor item 2 are fulfilled, create the nominal panel plane via the first three corner points of the panel.

To accept a curved panel for PLCM, there is a check for maximum deviation of the corner points of the panel. The distances are measured from the created panel plane. The maximum allowed deviation is by default 50 mm. It might be overridden by the logical name SBH_CPAN_MAXDEV.

- **Piece Parts**

The piece parts are extracted in the same way as in a normal run of PPANPARTS and get their names according to the same conventions as parts for the normal production flow. However, parts with a "hit" on the assembly names in the Assembly Name file (cf. above) should be combined to an assembly part, and are thus supposed to get their marking added in the panel line. Therefore the marking is added to the assembly part only and the piece parts are stored without any marking. However, it is essential for the function of the Panel

Line module (see below) that these rudimentary piece parts are available in the plate data bank.

- **Assembly Parts**

  The assembly parts are the parts that would result if only the assembly seams (cf. above) had been available in the panel. Their names are formed in the following way:

  `<assembly_part_name> ::= <panel_name>-<assembly_name>`

  Example:

  An assembly part with assembly name U2B derived from panel AB123-4 will get the name AB123-4-U2B.

  Assembly parts will be stored in the plate data bank together with normal piece parts.

  A symmetric panel may contain the model information for two sets of assembly parts, one on port and one on starboard side. Both these assembly parts will be generated provided their names are available in the Assembly Name file.

  The following types of information will be available in an assembly part:

  1. The outer geometry of the assembly part with information about excess and bevelling, including continuously varying bevel angles.

     Excess defined according to `EXC_TYPE_3` (cf. above) will be removed from the assembly parts.

     Bevel information is taken away from edges with remaining excess (i.e. where modification of the edge geometry are supposed to take place in later assembly stages).

     An assembly part is welded together from several piece parts which may have different thicknesses. Thus the bevel type may change along an edge, e.g. when passing into a piece part with thicker plate. The bevel type is normally associated with a gap. Thus the gap might change along an edge, creating small "steps" in the geometry of the edge. However, these are not acceptable when cutting the edges (with bevels) in the panel line. Therefore, PPANPARTS in a situation as described above replaces all gaps with the **smallest** gap along the current edge, see the figure below which shows a very much magnified view of an edge in the transition from one thickness (and bevel type) to another.

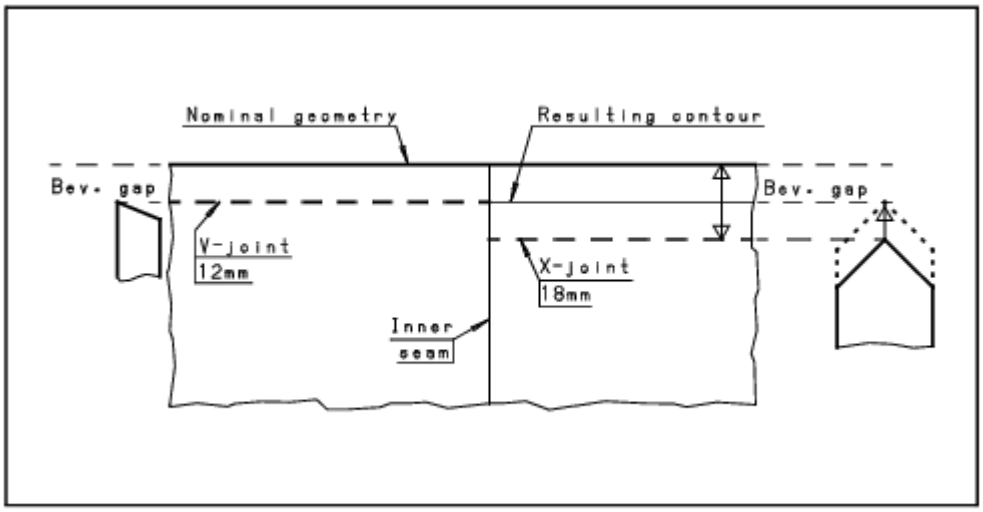


Figure 1:2. Figure showing a very much magnified view of an edge in the transition from one thickness (and bevel type) to another.

2. Information about holes and marking traces are derived in a similar way as for ordinary piece part cutting.
3. Those stiffeners on the assembly part which have the assembly common with the assembly part (supposed to be the parallel stiffeners mounted in station 4 of the panel line) will result in both a marking trace and a trace for sand blasting. The latter will be restricted to within a certain distance from the edges to prevent the blasting tool to be damaged. This is made with the IP MARKGAP in the program sf834d.
4. The inner (piece part) seams of the assembly parts are available as geometry (in order to be able to lock the height sensor when passing them).
5. Information about the piece parts is available as attribute information in the assembly part (extension, thickness).

1.1.5 Panel Line Control Module

The ACTUAL PLCM module is an interactive program (similar to the ordinary plate nesting module) used to create numerical control information for the panel line. Its main function is to control a burning and blasting/marking gantry in station 3 of a panel line. However, it has also the optional function to control a certain type of preparation (parallel cutting) of the piece parts in station 1 which are to be welded together to form the assembly part in station 2.

This module is supposed to be described in detail in a document of its own. However, its main characteristics are summed up below.

- **Piece Part Cutting Option**

  For a given assembly part this option has the following characteristics:

  - it finds out which piece parts it consists of,
  - sorts them in the correct order depending on the orientation of the assembly part in the panel line,
  - produces control information for the parallel cutting (including bevel cutting) of them,
  - produces a workshop sketch for station 1 of the panel line.

At the present time this option supports only a special equipment in station 1 delivered by TSB GmbH.

Result of this option is control information for parallel burning (ASCII file in a certain format) and workshop drawings.

- **Control of Panel Line Gantry**

  This is the main function of PLCM, used to produce control information of a gantry for cutting and blasting/marking of assembly parts. It has the following main characteristics.

  - Support for burning, both one torch burners for vertical cutting of holes and multiple torch burners for burning of complicated bevels.
  - Automatic translation of user defined bevel types into burner selection and control.
  - Support for blasting.
  - Support for normal marking.
  - Support for labelling of text and symbols.
  - Support for gantry equipped with one or two tool "wagons". In the latter case the use of the two sets of tools are optimized (by parallel operation) to minimize processing time.
  - Semi-automatic selection of orientation of the assembly part in the panel line depending on orientation of main stiffeners (one of the two possible orientations suggested).
  - Automatic to semi-automatic generation of burning loops and selection of auxiliary functions.
  - Semi-automatic generation of workshop drawing for the panel line.
  - Addition of special reference marks for mounting of the big stiffeners in station 4.

  The operation sequence supposed in PLCM may to some extent be adapted to the operation sequences in certain installations. Therefore, minor modifications may have to be implemented for each installation.

  Result of the function is, in addition to the workshop drawing, a file containing all necessary information for the numerical control of the gantry.

  The information is available in basically two different formats:

  1. A CL-file in a general XML-format, output as an ordinary ASCII file.
  2. A CL-file in a general so called generic format, output as an ordinary ASCII file.

  This will allow any customer to develop a post processor of his own. (However, these formats currently do not cover all the control information output from the assembly nesting).

  The generic format is described in a separate document, *Generic Data for 2-axis and 3-axis Post-processing (preliminary version)*.

- **Exceptional Handling of Piece Parts**

  The normal flow through the panel line has been described above, i.e. when the piece parts are parallel cut in the first station and then welded together to form the assembly part. However, in certain cases the piece part has got a chamfer along one of its edges or it has such a complicated geometry that it is necessary to process it in a burning machine for contour cutting. Such plates do not pass station 1 of the panel line but arrive directly to station 2.

  The conditions for this and the complete routines to be followed in such a case are described in *Assembly Nesting* .

- **Plate Alignment**

  For some Panel Line burning machines it is necessary to define how the plate shall be aligned at the machine. This is done by adding two plate alignment crosses on one of the individual parts. These crosses will be marked in Station 1 and also added to the assembly part. (For chamfer parts the definition must be done in the Nesting system and the marking will be made at the burning machine where the chamfer part is produced).

  When the tool path is verified, the user must first indicate one of the plate alignment crosses and an extra idle movement to the cross will be added.

1.1.6 Post Processors

A post processor is required to translate the CL file information output from PLCM into control information for the gantry.

Currently available is a post processor, based on a CL file in format, for a gantry with the control unit NCE 510.

For gantries with another control format post processors may have to be developed/ modified either by AVEVA or by the customer (in that case based on the generic format).

2 Assembly Nesting

2.1 General

This chapter is a user-oriented description of the Panel Line Control Module (PLCM). It concentrates on the functional characteristics of the system and on the interfaces to other subsystems of AVEVA Marine. The graphical part is based on the General Design System, see *User's Guide Drafting*. In this document are only described the PLCM specific functions. For the documentation of the general nesting functions, see *Nesting System - Application Functions*.

The Panel Line Module is a tool for automatic nesting of assembly parts as well as the piece parts of which an assembly part consists.

All parts to be nested are extracted from the model with the automatic part generation module PPANPARTS.

With the system the tool paths are defined creating CL-descriptions to be used as input to a suitable postprocessors. Shop drawings (burning sketches) can be created and plotted.

2.1.1 Coordinate Systems

A nested assembly part is described in a coordinate system with the origin in the lower left-hand corner of the parent plate. The nested piece parts however, are described in a coordinate system with the origin in the lower right-hand corner of the parent plate.

2.1.2 Defaults

The keywords of the PLCM specific default variables are given below in alphabetic order.

The following default parameters have been added in the Hull Panel Line Control Module:

ASSEMBLY_PROFILE_VERTICAL	Code for the presentation of assembly profiles. No = Horizontal Yes = Vertical
ASS_START_NO_GAP	Code for the option to put a start without gap: No = Do not allow starts without gap Yes = Allow starts without gap
BLAST_COLOUR	Colour for blasting contours.

BLAST_PARALLEL	This code controls the parallel blasting support for the burning machine: No = Parallel blasting is not supported Yes = Parallel blasting is supported
BLASTING_ADDITINAL	Code for defining single blasting starts: No = Single blasting starts not allowed Yes = Single blasting starts allowed
BURN_ALLOWED	Code for the allowed types of burning contours (only if S1_MARK_INDPARTS_CHAMFER = Yes and all individual parts are marked as chamferparts) None = Starts cannot be set on any burning contour BurnInner = Starts can be set on inner contours BurnOuter = Starts can be set on outer contours BurnAll = Starts can be set on all burning contours
CLAMPING_EDGE	The text displayed at the clamping edge.
COMBI_PROCESS_DIRECTION	Code for the directionof the combined blasting/ marking process No =Horizontal Yes = Vertical
COMBI_PROCESS_GEOMETRY	Code for the geometry to be used in the combined process blasting-marking: • Marking Use the marking geometry • Blasting Use the blasting geometry
COMBI_PROCESS_VMARK	Code for using V-mark in the combined blasting/ marking process No = Do not use V-mark Yes = Use V-mark
CORN_LOOP_CTRL	More information about this keyword in the Plate Nesting default information. See *Defaults in Chapter Initialisations for Nesting*.
CORNER_TOL	The tolerance used when searching for an individual part.
DIST_BURNERS	The distance between the main burner and a side burner in a tool set.
DIST_EDGE_STOP	The distance from the edge of parent material to the point of end burning. A negative value indicates that the burning will end before the edge.
DIST_PLATE_PLATE	The distance between the raw plates (station 1 only).
ENTRANCE_LINE	The text displayed at the entrance line.

EQUIPMENT_1	Tool set 1 activity: No = Not active Yes = Active
EQUIPMENT_2	Tool set 2 activity: No = Not active Yes = Active
GSDMARK_SEPARATE	This code controls the order in which the GSD's are treated when starts are created for all marking lines: No = No separate treatment of GSD's Yes = The marking lines are treated first, then the GSD's
HEIGHT_CONTROL_ALWAYS	This code controls if the height control at inner seams should be made even if the upper side is not flat: No = The height control is made only if the upper side is flat Yes = The height control is always made
HEIGHT_CTRL_OFF_DIST	The distance before an inner seam where the height control is switched off.
HEIGHT_CTRL_ON_DIST	The distance after an inner seam where the height control is switched on.
IND_NEST_MIRROR	Code for mirroring the individual parts: No = Do not mirror Yes = Mirror
IND_PART_ALL_STARTS	Code for the option to put both starts on the top and bottom of individual parts: No = Do not put start on the extreme contours Yes = Put all starts
IND_PART_DIMENSION	Code for the option to key in the raw plate dimensions: No = Only name allowed Yes = Name and dimension allowed
IND_PART_DIRECTION	Code for placing the first individual part: 0 = Against entrance line 1 = Towards the entrance line
IND_PART_NESTNAME	Code for creating the names of the individual parts: 0 = <assnest><partno><noparts> 1 = <assnest><partno>, <partno> always 2 digits

IND_PART_NESTING	This code controls the usage of the individual part nesting. No = No individual part nesting Yes = Individual part nesting

The following texts are leading texts for the texts connected to the individual parts. The texts are placed in the burning sketch for the individual parts.

IND_PART_TEXT_1	'Part'
IND_PART_TEXT_2	'Length:'
IND_PART_TEXT_3	'Prefab length:'
IND_PART_TEXT_4	'Width:'
IND_PART_TEXT_5	'Prefab width:'

The following texts are leading texts for the texts connected to the nesting of the individual parts. The texts are placed in the burning sketch for the individual parts.

IND_PLATE_TEXT_1	'Raw plate'
IND_PLATE_TEXT_2	'DNC number'
IND_PLATE_TEXT_3	'Burner id'
IND_PLATE_TEXT_4	'Position number'
IND_PLATE_TEXT_5	'Sequence'

The following texts are leading texts for the texts connected to the raw plates used in the nesting of the individual parts. The texts are placed in the burning sketch for the individual parts.

IND_RAWPLATE_TEXT_1	'Raw plate'
IND_RAWPLATE_TEXT_2	'Length:'
IND_RAWPLATE_TEXT_3	'Width:'
IND_RAWPLATE_TEXT_4	'Quality:'
IND_RAWPLATE_TEXT_7	'Thickness:'

The following texts are side indication texts in the nesting of the individual parts. The texts are placed in the burning sketch for the individual parts.

IND_RAWPLATE_TEXT_5	'L'
IND_RAWPLATE_TEXT_6	'R'

LABEL_AUTO_POSNO	This code controls the automatic labelling of position numbers: None = No automatic labelling Blast = Automatic labelling of the marking lines connected to blasting lines Mark = Automatic labelling of all other marking lines All = Automatic labelling of all marking lines
LABEL_AUTO_POSNO_DIST	The relative distance from the marking line start at which the labelled text is positioned.
LIST_ASSPROF_EXT	File extension for the file with assembly profile data (empty means no file).
LIST_PARTSEQ_EXT	File extension for the file with individual part sequence (empty means no file).
LOCATION_CL_REPLACEMENT	Code for CL-replacement of direction text: No = Do not replace SB/PS-text by CL text Yes = Replace SB/PS-text by CL text
MARK_BLAST_COLOUR	The colour used for simultaneous marking-blasting lines.
MARK_BLAST_MIN_LENGTH	The minimum length of a contour to be simultaneously marked and blasted.
MARK_CTRL_OFF_DIST	The distance before an inner seam where the marking height control is switched off.
MARK_CTRL_ON_DIST	The distance after an inner seam where the marking height control is switched on.
MARK_PARALLEL	This code controls the parallel marking support for the burning machine: No = Parallel marking is not supported Yes = Parallel marking is supported
MARK_PARAL_DIFF	Factor used to decide if two marking contours can be marked in parallel. The length of the shortest contour divided by the length of the longest contour, is compared with this factor. If the result of the division is less than the factor, contours can not be marked in parallel. (Decreasing the factor allows more contours to be marked in parallel.)
NEST_PARTS_IN_PLCM	Code for adding parts/assembly parts in an assembly nest: No = Not allowed Yes = Allowed
ONLY_FLAT_UPPER_SIDE	Code for parts with/without flat upper side: No = The shape of the upper side is irrelevant Yes = The upper side must be flat

PLATE_ALIGNMENT_GET_MODE	Code for retrieving th plate alignment: Manual = Manual Automatic = Automatic
PLATE_MAX_X	Maximum length for assembly part.
PLATE_MAX_Y	Maximum width for assembly part.
PLATE_POS_CTRL	This code controls the display of the nestings of the individual parts. The next nesting will be displayed in the direction given by the default. The following values are currently possible: +1 = positive y-direction
POSITION_ASSPART_U	Code for automatic translation of the assembly part in the u-direction (after individual parts nesting): No = No automatic translation Yes = Automatic translation
POSITION_ASSPART_V	Code for automatic translation of the assembly part in the v-direction (after individual parts nesting): No = No automatic translation Yes = Automatic translation
RASTER_MARKING	This code controls the usage of raster marking of profile marking lines: No = No raster marking Yes = Raster marking
RASTER_MARK_DIST	The successive distances between the light sources.
RASTER_MARK_LENGTH	The length of a raster marking.
REF_DIST1 REF_DIST2 REF_DIST3 REF_DIST4 REF_DIST5	The distances from the left edge of the area to each reference line.
REF_HEIGHT	The height of the station area for profile assembly.
REF_NO_FIXED	The number of the reference line around which the assembly part should be centred. The parameter should be 0 for interactive definition.
REF_NO_LINES	The number of reference lines in the profile assembly station. The maximum number is 5.
REF_WIDTH	The width of the station area for profile assembly.
SCRAP_OPPOSITE_REFSIDE	Code for the position of the scrap: No = Scrap opposite clamping edge Yes = Scrap opposite reference side

SIDE_LOOPS	Code controlling the addition of side loops: No = Do not add side loops Yes = Add side loops
THICKNESS_CTRL	Code for thickness check in the height control at inner seams: No = Do not control thickness Yes = Control thickness
TRANSFORM_ASSPART	Code for using the TRANSFORMATION function transformation. Only a restricted number of transformation types are available (translation and rotation). No = Do not allow the user to transform Yes = Allow the user to transform
UPDATE_PROF_SEQUENCE	Code for updating the assembly profiles with sequence number: No = Do not update Yes = Update
UPDATE_PROF_SEQ_SORT	Code for sorting of assembly profiles with the same y-coordinate in the definition of Reference Lines: -X = Sort in the negative x-direction +X = Sort in the positive x-direction
VERIFY_PARAL_COLOUR	Colour used to verify a contour to be handled in parallel.
WELD_TOLERANCE	Welding tolerance added for some individual parts.

Below follows an example of the default parameters:

ASS_START_NO_GAP	= No
BLAST_COLOUR	= Cyan
BLAST_PARALLEL	= Yes
BLASTING_ADDITIONAL	= Yes
BURN_ALLOWED	= BurnAll
CLAMPING_EDGE	= 'Clamping edge'
COMBI_PROCESS_DIRECTION	= Horizontal
COMBI_PROCESS_VMARK	= No
COMBI_PROCESS_GEOMETRY	= Marking
CORNER_TOL	= 20.000000
DIST_BURNERS	= 20.000000
DIST_EDGE_STOP	= 50.000000

```
DIST_PLATE_PLATE          = 0.0
ENTRANCE_LINE             = 'Entrance line'
EQUIPMENT_1               = Yes
EQUIPMENT_2               = Yes
GSDMARK_SEPARATE          = Yes
HEIGHT_CONTROL_ALWAYS     = No
HEIGHT_CTRL_OFF_DIST      = 50.000000
HEIGHT_CTRL_ON_DIST       = 20.000000
IND_NEST_MIRROR           = No
IND_PART_ALL_STARTS       = Yes
IND_PART_DIMENSION        = No
IND_PART_DIRECTION        = 1
IND_PART_NESTNAME         = 0
IND_PART_NESTING          = Yes
IND_PART_TEXT_1           = 'Part'
IND_PART_TEXT_2           = 'Length:'
IND_PART_TEXT_3           = 'Prefab length:'
IND_PART_TEXT_4           = 'Width:'
IND_PART_TEXT_5           = 'Prefab width:'
IND_PLATE_TEXT_1          = 'Raw plate'
IND_PLATE_TEXT_2          = 'DNC number'
IND_PLATE_TEXT_3          = 'Burner id'
IND_PLATE_TEXT_4          = 'Position number'
IND_PLATE_TEXT_5          = 'Sequence'
IND_RAWPLATE_TEXT_1       = 'Raw plate'
IND_RAWPLATE_TEXT_2       = 'Length:'
IND_RAWPLATE_TEXT_3       = 'Width:'
IND_RAWPLATE_TEXT_4       = 'Quality:'
IND_RAWPLATE_TEXT_5       = 'L'
IND_RAWPLATE_TEXT_6       = 'R'
```

IND_RAWPLATE_TEXT_7	= 'Thickness'
LABEL_AUTO_POSNO	= Blast
LABEL_AUTO_POSNO_DIST	= 0.05
LIST_ASSPROF_EXT	= 'pam'
LIST_PARTSEQ_EXT	= 'psq'
LOCATION_CL_REPLACEMENT	= Yes
MARK_BLAST_COLOUR	= BlueViolet
MARK_BLAST_MIN_LENGTH	= 2000.000000
MARK_CTRL_OFF_DIST	= 50.000000
MARK_CTRL_ON_DIST	= 50.000000
MARK_PARALLEL	= Yes
MARK_PARAL_DIFF	= 0.950000
NEST_PARTS_IN_PLCM	= No
ONLY_FLAT_UPPER_SIDE	= Yes
POSITION_ASSPART_U	= Yes
POSITION_ASSPART_V	= No
PLATE_ALIGNMENT_GET_MODE	= Automatic
PLATE_MAX_X	= 16000.00
PLATE_MAX_Y	= 18000.00
PLATE_POS_CTRL	= 1
RASTER_MARKING	= Yes
RASTER_MARK_DIST	= '500 3000 4000 4000 3100 '
RASTER_MARK_LENGTH	= 300.000000
REF_DIST1	= 3000.00000
REF_DIST2	= 6000.00000
REF_DIST3	= 9000.00000
REF_DIST4	= 12000.00000
REF_DIST5	= 15000.00000
REF_HEIGHT	= 12000.000000
REF_NO_FIXED	= 0

REF_NO_LINES	= 3
REF_WIDTH	= 12000.000000
SCRAP_OPPOSITE_REFSIDE	= No
SIDE_LOOPS	= No
THICKNESS_CTRL	= No
TRANSFORM_ASSPART	= No
UPDATE_PROF_SEQUENCE	= No
UPDATE_PROF_SEQ_SORT	= +X
VERIFY_PARAL_COLOUR	= Magenta
WELD_TOLERANCE	= 500.0

The keywords of the default variables specific for the nesting of the piece parts are given below.

S1_CVBA_IN_INDNEST_GENFILE	Code for adding bevel code for CVBA in the generic files for the individual parts: No = Do not add CVBA bevel code Yes = Add CVBA bevel code
S1_EQIPMENT_1	Tool set 1 activity: No = Not active Yes = Active
S1_FORM_NAME	The name of the nesting form to be used automatically when a form is inserted. The user will be prompted to give the formname if no name has been given.
S2_EQUIPMENT_2	Tool set 2 activity: No = Not active Yes = Active
S1_MARK_INDPARTS_CHAMFER	Code for handling of chamfer parts: No = Only those individual part that do not fulfil the Station 1 restrictions will be marked as chamfer parts. Yes = All individual parts will be marked as chamfer parts If at least one individual part does not fulfil the Station 1 restrictions.
S1_MARK_CHAMFER_PARTS	Code for handling marking on chamfer parts when all individual parts have been marked as chamfer parts: No = Do not add marking Yes = Add marking
S1_MIN_BURNPL_LE	Min length of burned plate

S1_MIN_BURNPL_WI	Min width of burned plate
S1_MAX_BURNPL_LE	Max length of burned plate
S1_MAX_BURNPL_WI	Max width of burned plate
S1_MIN_EDGE_I_TH	Min thickness for I bevel
S1_MIN_EDGE_X_TH	Min thickness for X bevel
S1_MIN_EDGE_Y_TH	Min thickness for Y bevel
S1_MAX_EDGE_I_TH	Max thickness for I bevel
S1_MAX_EDGE_X_TH	Max thickness for X bevel
S1_MAX_EDGE_Y_TH	Max thickness for Y bevel
S1_MIN_RAWPL_LE	Min length of raw plate
S1_MIN_RAWPL_TH	Min thickness of raw plate
S1_MIN_RAWPL_WI	Min width of raw plate
S1_MAX_RAWPL_LE	Max length of raw plate
S1_MAX_RAWPL_TH	Max thickness of raw plate
S1_MAX_RAWPL_WI	Max width of raw plate
S1_MIN_RESTPL_LE	Min length of rest plate
S1_MIN_RESTPL_WI	Min width of rest plate
S1_MAX_RESTPL_LE	Max length of rest plate
S1_MAX_RESTPL_WI	Max width of rest plate
S1_MIN_VERT_REST_LE	Min length of scrap after vertical cut
S1_MIN_VERT_REST_WI	Min width of scrap after vertical cut
S1_MAX_SCRAP_LE	Max length of scrap
S1_MAX_SCRAP_WI	Max width of scrap
S1_MAX_BEVEL_ANGLE	Max bevel angle
S1_MIN_BEVEL_ANGLE	Min bevel angle

Below follows an example of the default parameters:

S1_CVBA_IN_INDNEST_GENFILE	= No
S1_EQUIPMENT_1	= Yes
S1_EQUIPMENT_2	= Yes
S1_FORM_NAME	= ‘INDNEST_FORM’

S1_MARK_IND_PARTS_CHAMFER	= No
S1_MARK_CHAMFER_PARTS	= No
S1_MAX_BURNPL_LE	= 16100.000000
S1_MAX_BURNPL_WI	= 3050.000000
S1_MAX_EDGE_I_TH	= 10.000000
S1_MAX_EDGE_X_TH	= 50.000000
S1_MAX_EDGE_Y_TH	= 14.500000
S1_MAX_RAWPL_LE	= 16100.000000
S1_MAX_RAWPL_TH	= 50.000000
S1_MAX_RAWPL_WI	= 3100.000000
S1_MAX_RESTPL_LE	= 16100.000000
S1_MAX_RESTPL_WI	= 2050.000000
S1_MAX_SCRAP_LE	= 16100.000000
S1_MAX_SCRAP_WI	= 500.000000
S1_MIN_BURNPL_LE	= 5000.000000
S1_MIN_BURNPL_WI	= 1000.000000
S1_MIN_EDGE_I_TH	= 4.000000
S1_MIN_EDGE_X_TH	= 15.000000
S1_MIN_EDGE_Y_TH	= 10.500000
S1_MIN_RAWPL_LE	= 5000.000000
S1_MIN_RAWPL_TH	= 4.000000
S1_MIN_RAWPL_WI	= 1000.000000
S1_MIN_RESTPL_LE	= 5000.000000
S1_MIN_RESTPL_WI	= 500.000000
S1_MIN_VERT_REST_LE	= 2000.000000
S1_MIN_VERT_REST_WI	= 1050.000000
S1_MAX_BEVEL_ANGLE	= 50.0
S1_MIN_BEVEL_ANGLE	= 20.0

2.1.3 Functions, Overview

Below, a survey of the functions of the system will be given. In the overview, all function names are printed in block letters.

- **Nesting Functions**

- **Initiate Nesting Job**

Assembly Nesting, First Part

This function is used to start a nesting of an assembly part. The name of the nest job is given and if **OPERATION COMPLETE** is given the name of the nest job will be the same as the name of the assembly part. The user can then select the assembly part to be nested.

The assembly part is presented on the screen with the side turned upwards which contains most blasting contours. In this process the assembly part may have been reflected in the x-axis. The user has then the option to rotate the assembly part 180 degrees before it is automatically nested.

The circumscribed rectangle of the assembly part is used as a temporary raw plate. To this raw plate is added the excess of type `EXC_TYPE_3` for the piece parts. If no such excess has been defined default excess values are used, given by the default parameter `ADDITIONAL_EXCESS`.

If a station for preparation of the piece parts is available (controlled with the default parameter `IND_PART_NESTING`) the PLMC now enters the individual parts nesting mode. Otherwise the temporary raw plate is used as the final raw plate.

Individual Part Nesting

To facilitate the welding of the piece parts to form the raw plate to the nesting of the assembly part, it is necessary to define one side as a reference side. This side must always be straight. To do this the user simply points on the outer contour. If the x-coordinate of the closest segment on the outer contour is smaller than the x-coordinate of the mid-point of the circumscribed rectangle of the outer contour, the left side is chosen as reference side.

The system will now sort all piece parts belonging to the assembly part in ascending y-coordinate order. In this way the piece parts will be treated in the correct order.

For each piece part the user now has to defined the raw plate to be used. The selected raw plate is automatically positioned around the piece part. The default part to plate distance is also taken into account. If the raw plates are selected from a list, only the raw plate which are big enough are listed.

The least size is determined by the size of the piece part itself, the default part to plate distance, the welding tolerance for 'small' piece parts. It must be possible to have room for the welding tool, so the raw plate for small piece parts which not are rectangular in shape need to be somewhat larger then the piece part. The welding tolerance is given by the default parameter `WELD_TOLERANCE`.

The size of the raw plate is also dependent on the reference side. If one of the piece parts has half the length of the assembly part and is placed to the left on the assembly part, the necessary length of the raw plate will be the length of the assembly part if the reference side is to the right.

When the raw plates for all piece parts have been defined, the user has an option to let the system create rest plates automatically. This is only done if the size of the rest is large enough. The size is controlled with the default parameters for this station. A vertical cut will

be created at a distance from the piece part. This distance is dependent on the presence of EXC_TYPE_3, cf above. For small piece parts the welding tolerance is also used.

Before the final raw plate is created the system adjusts the individual raw plates so that the reference line will be straight. The temporary raw plate is then replaced by the final one.

Each piece part is nested on one raw plate, so the individual nesting ends up with a number of nested plates. The name of an individual nesting is

<assembly nest><order><ind number>

where <assembly nest> is the name of the assembly nesting, <order> is the order number for the current individual nesting (starting at 1) and <ind number> is the total number of piece parts for the current assembly part. The assembly nesting ABC with 5 piece parts will result in the individual nestings ABC15, ABC25, ABC35, ABC45 and ABC55.

There is however only created one burning sketch for all the individual parts. The sketch is automatically updated with the following data:

- individual part geometry
- the geometry of the rawplates
- texts with individual part dimensions
- texts with individual nesting data
- texts with raw plate dimensions
- side indication

The PLMC now resumes the assembly nesting mode.

Assembly Nesting, Second Part

The following items will be added automatically to the nested assembly part:

- corner loops with bevel info
- height control at inner seams
- height control before corner loops
- side loops

The corner loops are created automatically in all corners. Two consecutive segments defines a corner if the angle between them is larger than the default parameter CORN_LOOP_MINANGLE. It is also possible to get the loops in only those corners where there is a bevel transition. This is controlled with the default parameter CORN_LOOP_CTRL. The loops are unsymmetric (CORN_LOOP_RADIUS must be 0) and consists of 3 line segments. The segments to be burned are always extended the distance DIST_EDGE_STOP outside the raw plate. If DIST_EDGE_STOP < 0 the burning will stop before the edge.

In the corner loops are automatically added all auxiliary functions necessary to control advanced bevel burning. Note that these auxiliary functions are some kind of pseudo functions.

On some burning machines the height control device is in contact with the plate. To get a good result from the burning, especially bevel burning, it is necessary to temporarily switch the height control off when the burner passes an inner seam (the weld extends some millimetres above the plate) and then on again (flying). This is done automatically if both the default parameters HEIGHT_CTRL_OFF_DIST and HEIGHT_CTRL_ON_DIST are > 0.

The height control is always switched off before a corner loop. The same default parameter as above (HEIGHT_CTRL_OFF_DIST) is used.

In some cases it is necessary to change bevel along one or both of the vertical sides of an assembly parts. This can only be done at an inner seam. It is seldom possible to change the bevel parameters on the fly. To solve this problem the **side loop** has been introduced.

In the general side loop, the right burner is switched off at a default distance (`DIST_BURNERS`) before the seam. The burning continues the same distance after the seam where the left and main burners are switched off. The tool is then moved (idle) to a position 40 mm before the inner seam and 70 mm outside the raw plate. In this position new bevel parameters are given to the burners and the burning continues with a 30 mm straight line after which the height control is switched off. The burning then continues along a quarter of a circle with 40 mm radius.

The side loops are created automatically. If side loops are present, parallel burning is not possible.

On some burning machines it is also necessary to have a height control at the inner seams for the marking. This is made if both the default parameters `MARK_CTRL_OFF_DIST` and `MARK_CTRL_ON_DIST` are > 0. When the starts are created for the marking, all marking lines passing one or several inner seams are treated.

For horizontal inner seams the marking ends at a default distance before the inner seam and starts again at a default distance after the seam. This means that one start is replaced by at least two starts. All starts connected to one marking line will however be treated in sequence to minimize extra idle movement. Parallel start will be kept.

For vertical inner seams the treatment is the same as for the horizontal ones, but parallel starts will be replaced by single starts.

- **Old Plate**

  This function is used when an old assembly nesting is to be brought up for modification. If individual nestings are have been defined they are also read from the data bank. When all nestings have been read the system is in the individual parts nesting mode.

  If an assembly part has been changed after it was nested the user has the option to automatically delete all corner loops, side loops and auxiliary function and re-create them.

- **Store Plate**

  This function is used to store the current assembly nesting. If individual nestings are have been defined they are also stored on the data bank.

  One file is also created for each individual parts nesting. This file contains control data for the burning machine at the first station. The name of these files are

  `<ind nesting name>.tsb.`

  For chamfer parts, see below, the special chamfer file is updated with the name of the assembly nesting.

- **Exchange Raw Plate**

  This function is used to exchange the raw plate of an individual part nesting. The final raw plate of the assembly part will be re-created. Rest plates can be created.

- **Rename Plate**

  This function is used to rename the current assembly nesting. If individual nestings are have been defined they are also renamed. The information in the individual nesting burning sketch are updated.

- **Delete on Data Bank**

  This function is used to delete assembly nestings from the data bank. If individual nestings are have been defined they are also deleted.

- **Corner Loop**

  In some cases the length of one of the segments in an unsymmetrical corner loop can be several metres. In such a case it can be necessary to modify the existing corner loop. The system asks for the new values of the lengths of the line segments to be burned.

  If corner loops have been created only at the corners where bevel transitions occurs, this function can be used to create an unsymmetric corner loop in the corners where no loops have been created. Note however that the created loop will not contain any information for bevel burning.

- **Start Blasting**

  This function is used to create all blasting starts automatically. Parallel blasting is supported (only horizontally).

- **Start Marking**

  This function is used to indicate a single marking start, in which case the operator will be prompted to indicate the start position or to create marking starts on all marking lines automatically.

  Parallel marking is supported (both vertically and horizontally) but only when all marking starts are created automatically.

  GSD's are treated together with marking lines in this function if the default parameter `GSDMARK_SEPARATE` is set to No. Otherwise the GSD's will be treated after the marking lines.

  The PLCM contains an option to only mark a part of those marking lines that are connected to blasting lines (controlled by `RASTER_MARKING`. The raster marking will be made parallel if possible, and the length of each raster mark is given by the default parameter `RASTER_MARK_LENGTH`. The position of the mid point of the raster markings are controlled with `RASTER_MARK_DIST`.

- **Start Burning**

  This function is used to indicate a single burning start, in which case the operator will be prompted to indicate the start position and possibly the tool direction or to create burning starts for all the holes on the plate automatically. Parallel burning is supported, vertically for the outer contour and horizontally for the holes.

  The switch between single and parallel starts is made with **OPTIONS**.

  To create a parallel start for the outer contour the user has to indicate the following points (Y1 refer to the left tool set and Y2 to the right tool set):

  1. Start position Y1 parallel
  2. End position Y1 parallel
  3. End position Y1 single (**OPERATION COMPLETE** means the same point as end Y1 parallel)
  4. Start position Y2 parallel
  5. End position Y2 parallel

The corner loops at the positions 1) and 5) will be split into starting and ending hooks. These hooks must have been created in PLCM in order to extend outside the raw plate automatically, cf below.

The system checks that parallel burning is possible (Y1 and Y2 cannot be too close, the lengths of the sides to be burned must be equal).

To create a parallel start for a hole the user must first create a gap in the hole to be burnt by Y1. Then the user indicates the starting point for Y1 (in the gap) and the starting point on the second hole for Y2. If the holes are identical and are created on the same y-coordinate, the system automatically creates a gap and a start on the second hole in the correct position.

Normal nesting hooks should be used for the holes.

- **Automatic Labelling of Position Numbers**

  It is possible to let the system automatically generate the texts to signed for position numbers. This is controlled with the default parameters `LABEL_AUTO_POSNO`. If `LABEL_AUTO_POSNO` = Blast, the system automatically creates the position number text to be labelled for all marking lines connected to blasting lines. The text is placed at the relative distance `LABEL_AUTO_POSNO_DIST` from the starting point of the marking line. `LABEL_AUTO_POSNO` = Mark gives the position numbers for all other marking lines. Possible values are also No and All.

- **Reference Lines on Assembly Parts and Profiles**

  This function is a tool that provides alignment marking on the plates and profiles on the assembly part. The default parameters `REF_HEIGHT`, `REF_WIDTH`, `REF_NO_LINES`, `REF_DIST1`, `REF_DIST2`, `REF_DIST3`, `REF_DIST4` and `REF_DIST5` specifies the layout of the reference lines in the welding station.

  When the function is used in the PLCM, the default height and width of the welding station is shown on the screen and the user can place the rectangle around the welding unit using normal positioning tools. If the assembly part is too larger for the welding station, a message is given and no rectangle appears.

  The reference lines will be shown on the assembly nesting, for checking purposes. The assembly part is updated with the reference lines and stored in the assembly nesting. If the function is repeated, the old reference data are deleted and overwritten. The reference lines are intersected with every stiffener in the assembly part belonging to the same assembly on the lowest level, and for each intersection the information about the line is stored with the stiffener. It is however required that the system is used in a way that all parts are individually stored, otherwise no marking of profiles can be done. The information is stored in such a way that it can be written to the generic file. The profile line will then mark the reference line on the profile. Thus, in the profile welding station there will be alignment marks on the assembly part to position the unit in the station, and marks on the stiffeners so that they can be aligned on the assembly part.

- **Store Hook**

  This function is used to store a drawing as a hook in the hook object D003HOOKS. The hook contour will be marked with contour code 1 = 1. (In the nesting system contour code 1 = 0). These hooks will automatically extend outside the raw plate. This is essential in bevel burning where the start has to be at an edge.

- **Verify**

  This function is used to create a generic file which is input to the post processor. Both incremental and absolute modes are supported. In the absolute mode, all parallel idle movement is performed in 3-axis mode.

  Parallel verification is supported. A collision control of the two tool sets as well as a check that the two tool sets are within the limits of the machine, is performed. If the second tool set is at a position outside the limits, the second tool set is sent to its home position and locked.

  The verification is aborted when the marking or blasting geometry extends outside the raw plate.

- **Production information / Show Production Information**

  The functionality of these functions are exactly the same in the PLCM, both for the assembly nesting and for the individual parts nesting, as in the Nesting system.

• Miscellaneous Functions

- **Plot**

  When a plot is made of the burning sketch for the individual parts nesting, the name of the plot will be the same as the name of the first individual nesting.

• Chamfer Parts

If the station for preparing the piece parts cannot handle part with chamfer, it is possible in the PLCM to automatically skip these piece parts (hereafter called chamfer parts). Piece parts which violates the restriction of the burning machine at this station are also treated in the same way.

The chamfer parts are marked with a special object code in PPANPARTS, so they will not be treated as piece parts in the assembly nesting. The chamfer parts have to be nested in the Nesting system.

The nesting of the chamfer part is not made with the original part geometry. The circumscribed rectangle, optionally modified with `EXC_TYPE_3`, is used instead. The reason for this is that the resulting plate must be rectangular in shape since it shall be welded together with the other piece parts.

When a chamfer nesting is stored a file is created with the following contents:

- DNC ID
- Burner ID
- Paint TS
- Paint OS
- Length
- Width
- Thickness
- Quality

The name of the file is `<chamfer part>.chamfer`.

When an assembly part includes chamfer parts, the raw plate information is taken from the file above. The DNC id and the burner id is stored in the burning sketch.

It is not possible to make the nesting of an assembly part containing chamfer parts, until all chamfer parts have been nested. The system checks if the file mentioned above exists.

If a chamfer nesting is deleted the chamfer file is deleted if the chamfer part has not been used in the assembly nesting. Otherwise the chamfer file is renamed to `<chamfer part>.del` and the assembly nesting is deleted as well.

There is also an option to PLCM to accept that plate parts have been burnt to shape already before being brought to the panel line. The panel line is then used for welding, marking and blasting only. It is controlled by the default parameter S1_MARK_INDPARTS _CHAMFER. If this is set to Yes and at least one individual part has characteristics as indicated above, then all the remaining individual parts belonging to the same assembly part are supposed to be treated in the same way.

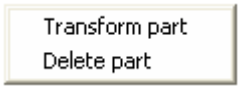
2.1.4 Right Click

A quick way to perform certain actions on objects in the nesting or in the burning sketch is by indicating a nesting component and right-click. The context menu alternative depends on which object type the user has clicked on.

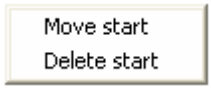
- **Panel Line Nesting**

The context menus are available for nested parts, bridges, starts, auxiliary functions and labelled texts. By right-clicking on an object in the nest a pop up menu is displayed showing the alternatives that can be performed.

Menu viewed for click on the raw plate.



Menu viewed for click on a Part.

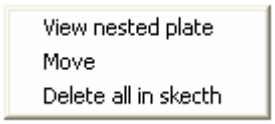


Menu viewed for click on a start.

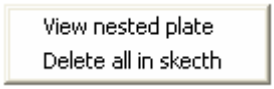
- **Burning sketch**

The context menus are available for text/symbol and production information. By right-clicking on an object in the burning sketch a pop up menu is displayed showing the alternatives that can be performed.

Menu viewed for click on the drawing form.



Menu viewed for click on a production information.



Menu viewed for click on the raw plate.