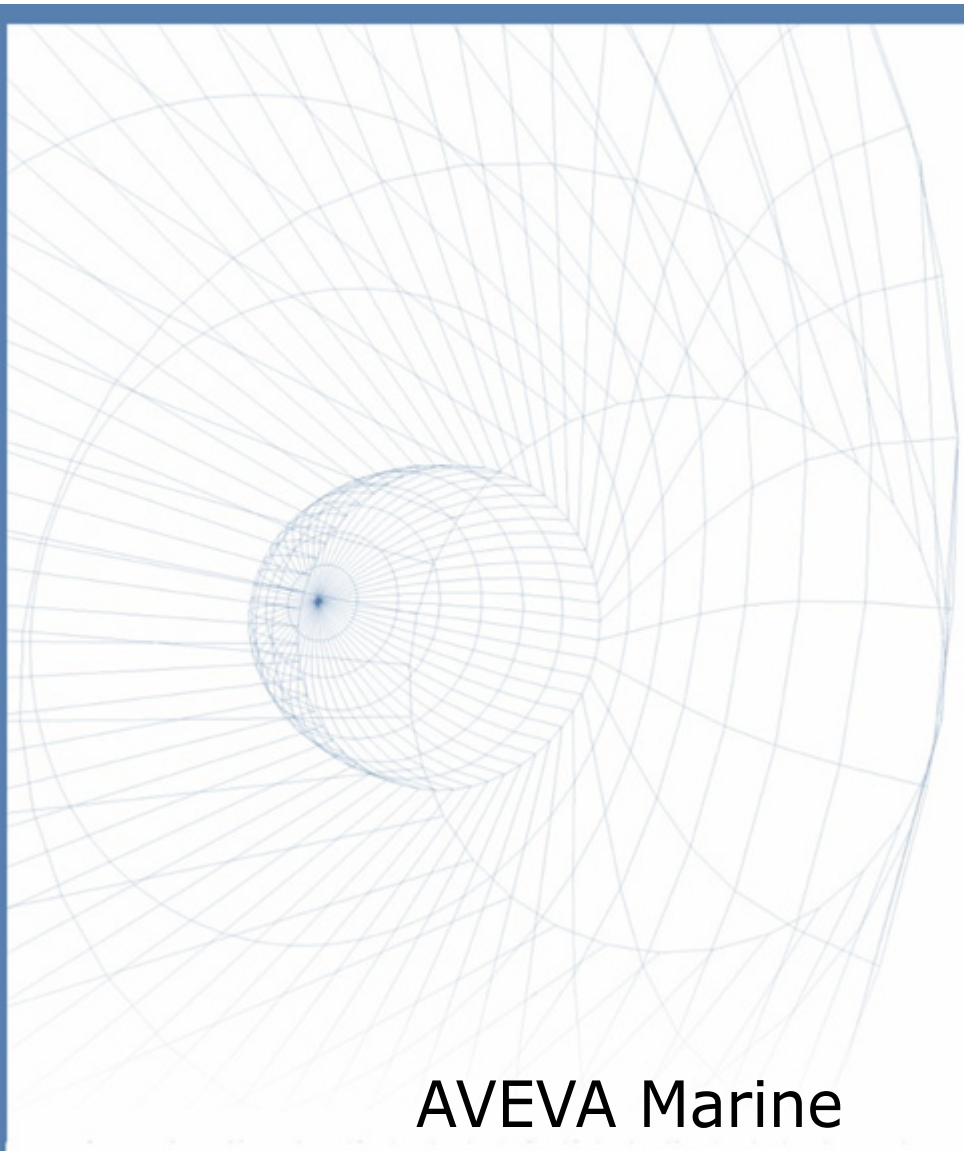

AVEVA Marine

Factory Automation

User Guide

www.aveva.com

AVEVA Solutions Ltd

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AVEVA Marine Factory Automation

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Factory Automation

1 Functions for Factory Automation

1.1 General about Factory Automation

Factory Automation is a collection of options to AVEVA Marine for automation and accuracy control in the ship building process.

The options can normally be utilized one by one or several together. They are typically integrated in the various design functions and detailed documentation about them can normally be found there.

In order for the Factory Automation functions to work properly according to a customer's specific needs, conventions, equipment, etc., a proper setup must normally be done. All necessary tools to do this set up are documented.

1.2 Profile Cutting Interface

The profile cutting interface option is used to basic data for cutting robots or for external profile manufacturing software. The robot controllers and/or robot programming languages must have the possibility to read the generic interface format, which is output from this interface. The receiving system must have a preprocessor.

The Profile Cutting Interface contains one program, which performs the transfer, and some belonging utility files. The result of a run is an ASCII file, in a specific format, describing the nested profiles or an individual profile to be cut. This ASCII file is called the "generic file". Input to the profile cutting option are nestings from the database for nested profiles or individual profiles and user given control records in a separate input file.

The generic file is an ASCII file, fully readable, containing all necessary information to control a profile cutting line. There will be one generic file for each nesting, called an "order". Each order contains all required data for each profile to be cut. The main data are the position of the zero point of the profile and the macro description for each end-cut. All holes, notches and cutouts are presented by full geometry in relation to a zero point.

This generic file may be used to create the specific input to control the robot itself and the means of transportation of the raw bar through the profile cutting line. A template for how to use the generic file to write preprocessors is available. This preprocessor should be developed by the customer or by the robot vendor. The option does not contain any translation to receiving robot programming languages or controllers.

Further detailed information about this option is available in:

- *Hull / Manufacturing / Profile Manufacturing*
- *Hull / Manufacturing / Profile Interface*
- *Hull / Manufacturing / Profile Cutting Interface*

1.3 Dotori (Variable Bevelling of Plates and Profiles)

The Dotori option for calculation of continuously varying bevel angle is "seamless", i.e. very closely integrated with the ordinary Bevel Handling of Hull. Its purpose is to support the definition and use of bevelling for fillet welding which in one way or other depend on the connection angle between the elements involved.

Further detailed information about this option is available in:

- *Hull / Setup and Customisation / Bevel, Excess and Weld / Bevel Handling*
- *Hull / Setup and Customisation / Cutout and Clips / Profile Cutout Standards / Cutouts and Inclined Profile Penetrations*
- *Hull/ Planar Hull / Design Language of Hull / Weld Statement*

1.3.1 Special Bevel Types

The bevel types of the Bevel Handling always have a fixed shape (even if they may have variants, e.g. with chamfer). A number of special basic bevel types have been added for fillet welding which may have varying bevel angles. These angles can be made dependent on the connection angle in different ways, e.g. be set equal to the connection angle, optionally with a fixed difference.

1.3.2 Automatic Selection of Bevel Type

It is possible to specify rules for the automatic selection of bevel types depending on the connection angle and the plate thickness. The rules for selection are specified in the bevel definition file and they are activated by the user when selecting the bevel type in the modelling. The selection may include bevel types with fixed or varying bevel angles.

1.3.3 Sets of Selection Rules

Different production lines and/or different types of parts may have different rules for the automatic selection of bevel types. Correspondingly, the system offers the possibility to define an unrestricted number of "sets", i.e. each production line (etc.) may have individual rules for selection of bevel type.

1.3.4 Extended User Control of Bevelling

The Dotori option has been combined with extended possibilities for end user control of the bevelling, with the following features:

- Additional options for the partition into bevel intervals have been implemented.
- The user may choose between automatic picking of bevel type and manuals election.
- The user may select between automatic calculation of the bevel angle and manually given angles.
- The user may select the set of bevel rules that should be used in the current situation.

1.3.5 Bevel in Webs of Built Shell Profiles

Built profiles may under certain circumstances be split into web and flange as separate parts. If the web in such a split is converted into a plate part then the system supports an automatic calculation of the bevel angle along its trace. However, currently the bevel type cannot be chosen - the system will currently generate a bevel with tight connection.

1.3.6 Bevel Information in Cutout Lugs

When parts of profile cutouts are welded against the profile section the system supports the calculation of the bevel angle of the contour against the profile section. Simultaneously the geometry of the cutout will be changed accordingly if required.

1.3.7 Modification of Part Geometries

When an edge is connected to a plate surface in a fillet weld connection then the geometry of the edge, i.e. the nominal geometry, may have to be modified because of the combination of the bevel type, the connection angle and the orientation of the material relative to the mould plane. An automatic adjustment of the part contours is an important feature of this option. It is effective for both fixed angle bevel types and for bevel types with varying bevel angles.

1.3.8 Bevel Information in Nesting and Burning

The bevel information defined in the modelling is automatically available in the parts extracted from the model. The bevel information can be presented in the nesting system and also output in the generic format for nested plates. Here the internal representation of the bevels has been replaced by explicit dimensions, angles, etc. Based on this format, post processors can be developed which transform the bevel information into direct burner control

1.4 Plate Cutting Interface

This option is used to extract any plate parts stored in the data banks into a generic format, suitable to be processed by programs outside the system.

The option is a stand-alone part of the software. The extraction can be applied for both plate parts from the internal structure and plate parts related to the shell structure The selection of plate parts can be done in the following ways:

- all belonging to an assembly
- all belonging to a block
- all belonging to a panel
- an individual plate part
- all changed/created after a given date

Further detailed information about this option is available in:

Hull / Manufacturing / Plate Manufacturing / Plate Interface

1.5 Genauigkeit (GSD Marking Triangles)

One of the major shipyard requirements is to reduce the need for additional fitting in the assembly process and thus reduce the production time. One way of doing this is to increase the accuracy in the assembly process, preferably at each step so that the accumulated errors are reduced. An important means to facilitate improved accuracy in both fabrication and assembly is the use of assembly is the use of assembly triangles (GSD Marking Triangles). These are used mainly for alignment in the assembly process, but may sometimes be used also for checking of accuracy after assembly.

The option is an integrated part of the software for marking of hull parts. Profile marking of GSD Marking Triangles is only handled in the Profile Cutting Interface. Customisation and control of the marking of GSD Marking Triangles is done via logical variables specific to this option. GSD Marking Triangles may be generated both for plates and for profiles and both for parts from the internal structure and parts related to the shell structure. GSD Marking Triangles are possible to mark in the following positions:

- at plate corners
- along profile traces
- along traces of abutting panels.
- along bracket traces
- along ship centreline
- along shell boundary
- at profile ends
- at manually defined positions on plane plate parts
- along the nominal plate edge on developed shell plates

In cases where both the GSD Marking Triangles and the marking trace are available for a marked item, the Nesting System will automatically combine them into one geometry with only one marking start necessary.

Further detailed information about this option is available in:

Hull / Setup and Customisation / Alignment Marking / Marks for Assembly (GSD)

1.6 Reference Planes

This option allows the shipyard to add reference planes for alignment into the product model. The reference planes can be marked on plates and profiles (both planar and curved), increasing the accuracy in the assembly process. The option allows the customer to define and set-up any plane, optionally restricted by a box and to use this to generate marking information on the plates and profiles. A plane can be defined as a principal plane, as a plane defined by three points or as a plane defined by a point and a normal vector. Planes to be marked are selected interactively. Hull has a function to display all planes intersecting the current panel and allows the user to pick among these.

Further detailed information about this option is available in:

Hull / Setup and Customisation / Alignment Marking / Reference Planes

1.7 Extended Shrinkage Handling

Hull has a basic function for automatic or semiautomatic compensation for shrinkage induced by heating, e.g. welding. This feature has certain limitations that have been removed by the option Advanced Shrinkage handling. In the old method the amount of shrinkage compensation is evaluated considering the marking traces (the number of them, their direction and the partition between them, the plate thickness, etc.). But then it is always applied as resizing factors (in two directions), i.e. not considering the position of each individual weld trace.

The extended shrinkage compensation is an automatic function - with options for manual overriding - that is applied in the automatic extraction of parts from the hull model, stored in

nominal dimensions (both profiles and plates). Condition for the evaluation in the Advanced Shrinkage handling is that the parts have been supplied with marking traces.

Customisation of the shrinkage compensation takes place via a shrinkage object that contains the amount of shrinkage as a function of plate thickness, partition between traces, etc. This object is currently common to the basic shrinkage compensation. The condition for when a trace should be considered to contribute to the shrinkage is controlled by a large number of default parameters of the parts extracting programs.

The extended shrinkage handling has the following main features:

- Shrinkage compensation consists of two parts: shrinkage compensation
- perpendicular to weld traces and compensation in the direction along weld traces
- The shrinkage perpendicular to the traces is applied locally in the position of the trace.
- The shrinkage compensation along the trace is applied as a scaling factor, the
- magnitude of which is depending on the partition between parallel traces.
- Shrinkage compensation may be applied for both planar and curved parts.
- The length of a trace relative to the size of a plate part will be considered when deciding if a trace should contribute to the shrinkage.
- The shape of a plate part will be considered: Somewhat different evaluation algorithms will be applied for "rectangular" compared to "triangular" parts.
- The shrinkage compensation in stiffeners from planar panels will be made so that the length of the profile becomes equal to the length of the corresponding weld trace on the plate (even if the trace should extend over several plates with different plate thickness).
- Additionally, it is possible to get a local shrinkage compensation at the ends of butt welded profiles.

Further detailed information about this option is available in:

Hull / Hull Setup and Customisation / Shrinkage Compensation

1.8 Extended Bevel Handling

The Hull application has in its basic version possibility to define bevelling along seams and outer contours of panels. This information has mainly been used to produce appropriate bevel symbols in drawings. It can also be used to set bevel information in burning sketches.

This option is a powerful extension to the basic bevel handling. It has also been extended to profiles and to brackets (those generated according to the extended bracket handling).

The extended bevel handling is available in modules handling the planar parts of the hull.

1.9 Robot Interface

The Robot Interface option is used to allow transfer of production information from the product model to welding robot facilities. The robot controllers and or robot programming languages must have the possibility to read the Robot Interface format. The receiving system must have a preprocessor. The production parts are transferred as volumes, e.g. general cylinders. In addition, curves of miscellaneous types can be transferred. Certain production information and contours of plates are also available.

The option contains one program together with utility files, which produce the transfer file. The result of a run is an ASCII file in a specific format describing the production parts and information in a specific format. The data to be transferred can be selected in various ways,

e.g. by names, by date and by a box in space. The accuracy of the resulting volumes can be controlled in different ways, e.g. excluding some components or details. Identities, e.g. panel names, are transferred. Layers for different types of information can be defined.

Please note, that the option does not contain any translation to receiving robot programming languages or controllers.

The option can achieve the following:

- Transfer of plane panels, including:
  - plate contours with cutouts and notches
  - thickness
  - holes
  - flanges
  - bracket plates
  - plane stiffeners
  - pillars
  - clips
- Transfer of bracket panels
- Transfer of plane plates
- Transfer of profiles, including:
  - holes
  - notches
  - cutouts
  - endcuts

    Note: Endcuts of folded and planar flanges can be treated. Endcuts of curved flanges are not treated in this release.
- Transfer of longitudinals/transversals either
  - converted into plane profiles (default) or
  - approximated with straight segments (optional)
- Transfer of limited production information, like:
  - material quality
  - surface treatment designations
  - position numbers
  - destination.

Further detailed information about this option is available in:

Hull / Manufacturing / Plate Manufacturing / Robot Interface

1.10 Panel Line Control

This module supports the definition of information to control production panel lines.

The Panel Line Control Module (PLCM) is used to nest assembly parts onto large raw plates and produce NC information for blasting, marking, burning and text labelling. The nesting of the individual piece parts and the creation of the large raw plate is also supported. The parts to be nested in this module are supposed to be generated in other modules of the Hull application. The main functions of this module are the interactive assembly nesting program and the post-processor to convert the cutting path into NC information. In addition, a number

of the utilities can be used. In the actual nesting program the function is roughly as described next. The individual piece parts are nested first, one part per raw plate. NC-information for each part and one burning sketch for all parts are automatically created. This first step can be omitted. The assembly part is then automatically nested on the raw plate from step one.

Parallel blasting, marking and burning are supported. Raster marking can be performed. All bevel burning is automatically controlled, including continuously varying bevel angle. Text labelling is supported. The texts for the position numbers can be generated automatically. A complete burning sketch for the assembly part can be created to be sent to the workshop.

A post processor is available to produce NC paper tape file in ESSI format and code for the NCE510 controller from ESAB. Other post processors are not available. If a new post processor is needed a customized development must be ordered from AVEVA.

This option will be available on request only.

1.11 Accuracy Control Measurement Points

The Accuracy Control Measurement Points option allows you to define and transfer measurement information to and from advanced measurement equipment for the purpose of accuracy control.

Production tolerances and reference points for steel assemblies can be defined in the system for later export to the measurement equipment.

Advanced measurement equipment, e.g. of a theodolite type of instrument, is frequently used for accuracy control in shipyards. The Accuracy Control Measurement Points option is an interface to this type of equipment and allows you to utilize the Product Model for measurement purposes.

The functions support the following work-flow for measurement operations:

- definition of production tolerances.
- planning and definition of measurement operation for assemblies.
- output of simplified measurement sketches in DXF format.
- output of measurement data in a neutral ASCII file.
- transfer of the neutral file and the DXF sketch to the measurement equipment

The functions of the option are provided as a combination of utility programs and interactive dialogs within applications.

For further detailed information about this option, please follow the following link:

1.11.1 Tolerance Definition

Production tolerances may be defined in different boxes in space. Separate tolerances will be handled in all principal axes directions. The tolerance definitions are available for the export to the measurement system.

1.11.2 Measurement Data Definition

The measurement data are defined on a per assembly basis and consist of a local coordinate system and measurement points. Interactive functions are provided to define the data in 3D views of the model. Measurement points are given an unique identification.

1.11.3 Export of Measurement Data

The export function generates a neutral ASCII file that contains the defined measurement data. The file can be imported into the measurement system.

1.11.4 Export of Measurement Sketches

A simplified measurement sketch in DXF format can be generated. Depending on the functionality of the measurement system, the sketch may also be utilized for the measurement operation.

1.12 Robot Interface 2 (STEP based format)

Robot Interface 2 is a variant of the Robot Interface. Robot Interface 2 allows you to export parts geometry and welding data of assemblies to a neutral file. The files can be used for further processing for robot welding purpose (or other use of geometry and welding trace data).

The data exchange model of Robot Interface 2 is defined along the lines of (but not fully compliant to) ISO 10303-218 for weld data. The language EXPRESS (ISO 10303-11) is used for the description of this data model.

The neutral file format is defined along the lines of ISO 10303-21 (STEP).

The Robot Interface 2 application reads assembly welding information, previously defined by the Weld Planning application. Assembly structure, parts geometry and welding information is exported to a neutral file, one for each assembly. The application provides a graphical tree browser to navigate in the assembly structure and to select assemblies for neutral file generation.

Please note that Robot Interface 2 does not contain any translation to receiving robot programming languages or controllers.

Further detailed information about this option can be found in:

Weld Planning / Robot Interface 2

1.12.1 Navigation in the Assembly Structure

Robot Interface 2 allows you to navigate in the assembly structure by means of a tree browser with drag & drop functionality and interactive query functionality for assembly information

1.13 Neutral File Generation

Neutral files in Robot Interface 2 format can be created on a per assembly basis from within a tree browser. The data exchange model of Robot Interface 2 is defined along the lines of (but not fully compliant to) ISO 10303-218 for weld data. It is composed of information elements, so called entities, describing the assembly structure, assembly welding information and parts geometry. The language EXPRESS (ISO 10303-11) is used for the description of this data model. The neutral file format is defined along the lines of ISO 10303-21 (STEP).

- The file generation has the following features:
- Neutral file generation on a per assembly basis.

- Full assembly structure definition (previously defined by Assembly Planning) included in output file.
- Full welding information including precise weld geometry (previously defined by Weld Planning) included in output file.
- Parts geometry for hull steel parts included in output file.