# Ship Designer

Data: 06/03/2023

# [3D - GENERIC COMMANDS]

## COG

DESCRIPTION: This command calculates the total weight of an object based on its specific weight and volume. It also stores the result in the document user text.

HOW TO: To use this command, first select the object for which you want to calculate the total weight. Then input the specific weight of the material (in kg/m3). The command will then compute the total weight and store it.

## Convslor

[DESCRIPTION] Converts Shipdesigner model to Nupas Cadmatic models

[HOW TO] select the objects you wish to convert and hit enter .

# Creaprog

[DESCRIPTION] Creates a Block Project for ShipDesigner

[HOW TO] provide as input “Costr” and “Block”, hit enter

# DHbeta

DESCRIPTION: It is a function that allows the user to add draining holes to a given object. The user can define the pattern and the type of draining holes they want to add.

HOW TO: To use the DHBetacommand, select an object and then define the type of draining holes you want to add, as well as the pattern. Then select the planes you want to add the draining holes to and the command will add the specified holes to the object.

# IDN

DESCRIPTION: It is a function that allows the user to save an object as a ShipDesigner part

HOW TO: To use the Assign ID command, select an object, fill the Part menu with the data, and then hit enter.

# Incre

DESCRIPTION: It is a function that allows the user to save extra steel on stiffners

HOW TO: To use the INCREID command, select an object, input the desidered extrasteel, and then hit enter.

# Mirrora

DESCRIPTION: It is a function that allows the user to mirror objects SB/PS

HOW TO: To use the Mirror command, select an object, hit enter

# Modelchecker OLD

DESCRIPTION: It is a function that allows the user to check if objects were saved correctly

HOW TO: Simply run the command

## Checker2

Purpose:

The purpose of this Python script is to check if the selected objects have the same layer and "type" and 3d modelled thickness.

User Manual for "check_layer_and_type" function:

Purpose:

The purpose of this Python script is to check if the selected objects have the same layer and "type" user text.

Instructions:

1. e.
2. Select the objects you want to check by clicking on them or using the selection tools in Rhino 3D software.
3. Press "Enter" to execute the script.

Output:

If all selected objects have the same layer and "type" and 3d modelled thickness, the script will print a success message in the command line. If any of the objects have different layers or "type" user text, the script will print an error message in the command line, specifying the name of the object and the error message.

## Quickview

[DESCRIPTION]:

This script sets the current construction plane to the plane associated with a user-selected view. It selects all objects on a layer named "PLANES", prompts the user to enter the name of the desired view, and then sets the construction plane to the plane associated with that view.

[HOW TO]:

To use this script, follow these steps:

1. In Rhino, open a file that contains objects on a layer named "PLANES".
2. When prompted, enter the name of the view whose construction plane you want to use.
3. The script will select the appropriate plane and set the construction plane to it.

## Setview

[DESCRIPTION]:

This script sets the current construction plane to the plane associated with a user-selected view. It prompts the user to specify whether or not to create a section of the plane. If the user selects to create a section, the script intersects the plane with all normal objects.

[HOW TO]:

To use this script, follow these steps:

1. In Rhino, open a file that contains objects on a layer named "PLANES".
2. When prompted, enter the name of the view whose construction plane you want to use.
3. When prompted, enter "1" to create a section of the plane or "2" to skip the sectioning process.

4. If you selected to create a section, the script will draw a section

## Supercut [fresko] 🧊

Description:

"Supercut" is a command-line tool that allows you to split objects in Rhino by extruding curves, either on a given construction plane or along the view direction. This tool can be very useful when working on architectural or product design projects, as it can make it easier to split objects quickly without needing to create additional planes or curves.

*Command-Line Options:*

When you run the "SplitByView" tool, you will be prompted to select objects to split. You can select one or multiple objects by clicking on them or by typing "SelAll" and pressing Enter.

Once you have selected the objects, the tool will prompt you to select cutting curves or draw lines. If you choose to select cutting curves, you will be prompted to select the curves that you want to use for splitting. If you choose to draw lines, you will be prompted to draw lines on the view to use as the cutting curves.

Next, you will be prompted to choose whether to split the objects using the construction plane or the view direction. If you choose to use the construction plane, you will be prompted to select a plane to use. If you choose to use the view direction, the tool will automatically use the current view's camera plane.

Finally, you will be prompted to choose whether to keep the original object or create a new solid object after splitting. If you choose to keep the original object, the tool will split the object and delete the portion that is cut away. If you choose to create a new solid object, the tool will split the object and keep both portions.

Examples:

1. Splitting objects using a construction plane:

- Select the objects that you want to split.

- Choose "Select cutting curves" and select the curves that you want to use for splitting.
- Choose "Use CPlane" to split the objects using the construction plane.
- Choose "No" to keep the original object and delete the portion that is cut away.

2. Splitting objects using the view direction:

- Select the objects that you want to split.
- Choose "Draw lines" to draw the lines that you want to use for splitting.
- Choose "Use view direction" to split the objects using the view direction.
- Choose "Yes" to create a new solid object and keep both portions after splitting.

Notes:

- If you select "Draw lines" instead of "Select cutting curves", the tool will prompt you to draw lines on the view to use as the cutting curves. You can draw as many lines as you like, and the tool will use them all for splitting.
- If you choose to split the objects using the view direction and the active viewport has a perspective projection, the tool will warn you that perspective view-based trimming may be unpredictable.

## SuperMoveEdge [fresko] 🧊

Introduction:

The SuperMoveEdge function is a command that moves the selected edges of a Brep object in Rhino. The function works by prompting the user to select the edges they want to move, and then specifying the starting and ending points of the movement.

Instructions:

1. Select the edges you want to move by clicking on them in the Rhino viewport. You can select multiple edges at once by holding down the shift key while clicking.

2. Once you have selected the edges, press enter to confirm your selection.
3. You will be prompted to specify the starting point of the movement. Click on the starting point in the Rhino viewport.
4. You will then be prompted to specify the ending point of the movement. Click on the ending point in the Rhino viewport.
5. The edges you selected will be moved from their original position to the new position specified by the starting and ending points.
6. The new Brep object with the moved edges will replace the original object in the Rhino viewport.
7. You can repeat the process to move additional edges or select a new set of edges to move.

Additional Notes:

- The SampleMoveBrepEdge function only works with Brep objects that contain edges. If the object you select does not contain edges, the function will not work.
- The movement of the edges is relative to their original position. If you want to move the edges to an absolute position, you will need to calculate the transformation matrix manually.
- The function uses the Rhino command line to prompt the user for input. Make sure the command line is visible and active when using the function.
- The function does not have any undo functionality, so make sure to save your work before using it.
- If you encounter any issues or errors while using the function, check the Rhino command line for error messages or consult the Rhino documentation.

# [WORKSHOP DOCS.]

## ExportDB

Introduction:

The B4 Exporter is a comman that automates the export of data from Rhino to Microsoft Excel. The script exports data from selected objects in Rhino to a new worksheet in an Excel workbook.

Instructions:

1. Run Rhinoceros and open the file containing the objects you want to export.
2. Run EXPORTDB command
3. Select the objects you want to export by using the Rhino command "SelObjects".
4. Press enter
5. The EXPORTDB  will create a new worksheet in an Excel workbook and export the data from the selected objects.
6. The exported data includes the following information: IDN, Type, Material, Quantity, Side, Mass, Description, Nested.

Troubleshooting:

If you experience any problems while using the B4 Exporter, try the following steps:

1. Make sure you are using Rhinoceros 5 or 6.
2. Make sure you have Microsoft Excel installed on your computer.
3. If you are still having issues, contact the developer for assistance.

## Annotation

Description: The Annotation function is a command that allows the user to add annotations to the Rhino viewport. The user can select from three options: Annotation, Titles, or Subtitles, and then select a starting point for the annotation. The user is then prompted to enter the text for the annotation, and the script creates the text object at the specified location with the specified text and font size.

How to use:

1. When prompted, select the option for the type of annotation you want to create: Annotation, Titles, or Subtitles.
2. When prompted, select the starting point for the annotation.
3. Enter the text for the annotation when prompted.
4. Press Enter to create the annotation.

## Draining Hole Export (DHEXPORT)

Description:

The DHEXPORT command creates a DH (Draining Holes) table for selected objects in Rhino. It opens an Excel file, sets the sheet name to "DH", and populates the DH table with the object ID, DH type, and draining holes positions.

How to Use:

1. Open Rhino and load the command.
2. Select the objects that you want to create the DH table for.
3. Hit enter

4. An Excel file will open, and the DH table will be populated with the selected objects' information.

Example:

1. Open Rhino and load the Python script.
2. Create a few objects and assign them with DH types and object types.
3. Select the objects.
4. "Run"
5. An Excel file will open, and the DH table will be populated with the selected objects' information.

Troubleshooting:

1. The script will only work if the Excel file path and sheet name are correct. If the Excel file cannot be found or the sheet name is incorrect, the script will not work.
2. If an object does not have a DH type or object type assigned to it, the script will skip that object and continue processing the rest of the selected objects.
3. If there are any errors during the script execution, the error message will be displayed in the command line.

## B2EXPORT (Fabbisogno profili)

Description:

The ExportDB function is a commanddesigned to export data from a Rhino model to a Microsoft Excel spreadsheet. Specifically, this function exports the lengths of objects in the Rhino model and writes them to an Excel file. The data is then organized by object type and displayed in a user-friendly format.

How to Use:

1. Run ExportLunghezzeB2
2. Select the objects

3. A new Excel file will be opened and populated with the length data from the Rhino model.

Example:

Let's say you have a Rhino model of a ship and you want to calculate the lengths of stiffners. You can use the ExportDB function to export this data to an Excel spreadsheet for further analysis.

Troubleshooting:

- Make sure that the objects in the Rhino model have lengths assigned to them before running the function.

## ExportPesoTagliati

Description:

This Python script exports weight data from Rhino and writes it to an Excel spreadsheet. The script uses the Rhino library and the Microsoft.Office.Interop.Excel library to access Rhino and Excel respectively. The script extracts the weight data from the Rhino document and then writes it to a specific Excel spreadsheet.

How to use:

1. Open a Rhino document that contains weight data that you want to export to an Excel spreadsheet.
2. Select the objects
3. Hit enter

Use case:

Let's say you have a Rhino document that contains weight data for several objects. You want to export this weight data to an Excel spreadsheet named "weights.xlsx" located in your Documents folder.

Troubleshooting:

- If the script doesn't run, make sure you have the necessary libraries installed and imported.
- If the script doesn't write to the Excel spreadsheet, make sure the Excel spreadsheet is not open in another program.
- If the weight data is not exported correctly, make sure the Rhino document contains weight data and that the weight data is labeled correctly

## AutoIDN

The "ID" command adds an ID to an object in Rhino. The ID can be either the name of the object or a combination of the name and type. The ID is added as a leader with an arrow pointing to the object, and it can also be accompanied by a text with the object's type.

Description:

The "ID" command allows you to easily add identification to objects in Rhino. This can be useful when you need to reference or label specific objects, especially in complex models.

How to use:

1. Type "ID" in the Rhino command line and press enter.
2. Choose the type of ID you want to add: "a" for name only, "b1" for ID without type, "b2" for ID with type, or leave it empty to add a leader with no text.
3. Select the object you want to add the ID to.
4. Click to place the leader and text in the desired location.

Example:

1. Choose "a" for name only.
2. Select the object you want to add the ID to.

3. Click to place the leader and text in the desired location.
4. The name of the object will now be displayed in a leader with an arrow pointing to the object.

Troubleshooting:

- If the ID is not displaying correctly, make sure that the layer for annotations is visible.
- If the text is too small or too big, adjust the height parameter in the command.
- If the leader is not pointing to the correct location, adjust the points of the leader until it is in the desired position.

## Height

User manual:

This script is used to calculate the height and draw a line between two selected points in Rhino. It also adds a text annotation indicating the calculated height.

Description:

The script gets two points from the user and calculate the height between them. Then, it adds a line between these points and an annotation text displaying the height.

How to use the command:

1. Open Rhino.
2. Open the script editor by typing "EditPythonScript" in the command line.
3. Copy and paste the script into the editor.
4. Run the script by clicking the "Run" button or typing "RunPythonScript" in the Rhino command line.
5. Select the first point to calculate the distance from.

6. Select the second point to calculate the distance to.
7. The script will draw a line between the two points and add a text annotation indicating the calculated height.

Example:

1. Select the first point at the bottom of the object.
2. Select the second point at the top of the object.
3. The script will draw a line between the two points and add a text annotation indicating the calculated height.

## Intavolatore

Description:

Intavolatore is a command for Rhino 3D modeling software. This script is designed to section a 3D object or group of objects and provide an output in the form of a technical drawing with detailed annotations and notes. The script is particularly useful for engineering and architectural design applications. The script works by selecting the surface of the plane where the section is to be created and then selecting the objects that need to be sectioned.

Example:

1. Open Rhino 3D modeling software
2. In the Rhino workspace, select the plane where the section is to be created by clicking on it
3. Select the objects that need to be sectioned by clicking on them
4. Follow the prompts in the command line to provide additional input
5. The output will be displayed in the Rhino workspace

Troubleshooting:

If the script does not produce the desired output, check the following:

- Check that the correct input has been provided for all prompts
- Check that the correct objects have been selected for sectioning
- Check that the correct surface has been selected for creating the section
- Check that the correct type of section has been selected (FR, LONG, or H)
- Check that the script is compatible with the version of Rhino being used. The script was written for Rhino version 6+ and may not be compatible with other versions.

## Length Type

User Manual: Calculate the total length of profiles of a specific type and calculate It's Length

Description: This command is designed to calculate the total length of selected profiles and assign it as a weight to the current document. It also prints the total length of the profiles in the command prompt.

How to Use the Command:

1. Open Rhinoceros 3D and create a new document.
2. Select the profiles for which you want to calculate the total length.
3. Run the script the command

Example:

Suppose you have created a few profiles in Rhinoceros 3D, and you want to calculate their total length and assign it as a weight to the current document. Follow these steps:

1. Select the profiles for which you want to calculate the total length.
2. Run the command
3. The script will print the total length of the selected profiles in the command prompt and assign it as a weight to the current document.

Troubleshooting:

- If the selected profiles do not have any length data, the script may not produce an accurate result.

## MarkLines

Description:

This script generates marks on edges of selected surfaces/polysurfaces.

How to use the command:

1. Open Rhino software.
2. Select the surfaces or polysurfaces on which you want to generate marks.
3. The script will generate marks on the edges of the selected parts.

Example:

Suppose you have a 3D model of a house in Rhino, and you want to generate marks on the edges of the roof surface. Follow these steps:

1. Open the Rhino software.
2. Open the file that contains the 3D model of the Yacht.
3. Select the deck surface of the yacht.
4. The script will generate marks on the edges of the roof surface.

Troubleshooting:

1. If the script is not generating marks on the edges, make sure you have selected the correct surfaces/polysurfaces.
2. If you are getting any error messages, read the message carefully to understand the issue and try to resolve it accordingly.

## MarkTexts

Command name: MarkText

Description: The MarkText command adds text to the midpoint of the selected curves with their corresponding object name.

How to use the command:

1. Type MarkText in the Rhino command line or use the script editor to run the code.
2. Select the curves for which you want to add text to the midpoint.
3. Press Enter.

Example:

1. Type MarkText in the Rhino command line.
2. Select the curves for which you want to add text to the midpoint.
3. Press Enter.

Troubleshooting:

If the command is not working, try the following troubleshooting steps:

1. Make sure you have selected curves before running the command.
2. Ensure that the text is not too big or too small to be visible on the curve.

## Normalizzatavole

Description:

This script is used to create traces from curves selected by the user, then add text on the centroid of the curve and scale and color the traces and directions in a layer named "marks", and finally, delete the original curves.

How to use the command:

1. Open Rhino and a new document.
2. Select the curves you want to generate traces from.
3. Wait for the script to finish running.

Example:

Suppose you have several curves in your Rhino document generated by “Intavolatore”, and you want to generate traces from them. Follow the steps above to use the script. Once the script finishes running, you will see marks and their directions appear on the curves you selected, with the text of the curve name in the centroid.

Troubleshooting:

- If you encounter any issues running the script, make sure that you have selected curves and they are all valid curves.
- If the traces are not appearing or the text is not centered, make sure that the curve is not too small, and that you have a proper layer setup with the correct settings.
- If the script runs for too long or the Rhino document appears to be unresponsive, try selecting fewer curves, or check your computer's performance to ensure that it can handle the script's processing demands.

## Pesotagliati

The "pesotagliati" command is a Rhino Python script that calculates the weight of selected parts in kilograms. It prompts the user to input the specific weight of the material in kg/m^3 and then outputs the total weight of the selected objects in the command line.

Description:

The "pesotagliati" command calculates the weight of selected parts based on their volume and specific weight. It uses the Rhino Python library and RhinoScriptSyntax module to interact with the Rhino interface and perform calculations.

How to use the command:

1. Open Rhino and load the script containing the "peso" command.
2. Type "peso" in the Rhino command line and press enter.
3. Select one or more surfaces or polysurfaces to calculate their weight.

4. Input the specific weight of the material in kg/m^3 when prompted.
5. The total weight of the selected objects will be outputted in the command line.

Example:

1. Type "peso" in the Rhino command line and press enter.
2. Select a surface or polysurface to calculate its weight.
3. Input the specific weight of the material in kg/m^3 (e.g. 2713 for steel).
4. The total weight of the selected object will be outputted in the command line (e.g. "Il peso complessivo e' di 1000.0kg").

Troubleshooting:

1. If the "peso" command is not working, ensure that the script containing the command is loaded in Rhino and that there are no errors in the script.
2. If the command is still not working, ensure that the selected objects are surfaces or polysurfaces and that they have a name and material assigned to them.
3. If the command is still not working, ensure that the specific weight of the material is inputted correctly and that it is in kg/m^3.

## ProfLength

The "proflength" function is a command that allows the user to calculate the length of profiles and assign them to parts in the model. The command can be used for both plate and bulb profiles, and for all profiles that can be modeled.

Description:

The function starts by asking the user to select the profiles to which to assign the length. It then checks if the selected objects are plate profiles or not. If they are plate profiles, it calculates their net length and assigns it to the "length"  of the objects. If the objects are also part of a shell structure, the function also calculates their total length and assigns it to the "length2" user text.

How to use the command:

1. Run the script.
2. Follow the instructions in the command line to select the profiles to which to assign the length.

Example:

Suppose the user has a model with several steel profiles and wants to calculate their length. The user can use the "ProfLength" function as follows:

1. In the command line, select the profiles to which to assign the length.
2. The script will calculate the net length of the profiles and assign it to the "length" user text of the objects.
3. If the profiles are also part of a shell structure, the script will calculate their total length and assign it to the "length2".

Troubleshooting:

- If the command does not select any objects, make sure that the 3D model contains objects that can be modeled with polysurfaces.
- If the command does not calculate the length of some profiles, make sure that the objects have a "type" that identifies them as plate or bulb profiles.
- If the command assigns the wrong length to some profiles, make sure that the objects are correctly modeled as polysurfaces.

## ProfSag [OLD]

Description:

It will ask the user to select one or more stiffners to work with, then the script will divide a curve into equally spaced segments based on a user-defined step value, and calculate the height and angle for each segment. The output will be written to an Excel file, which will open automatically upon completion.

How to use the command:

1. Open the Rhino software on your computer.

2. Open a new or existing file that you want to work with.
3. Open the Python editor in Rhino by typing "EditPythonScript" in the command prompt and pressing enter.
4. Click "New" to create a new script.
5. Copy and paste the script into the editor.
6. Save the script with a unique name.
7. Click on the "Run" button in the editor to execute the script.

Example:

Here's an example of how to use the SetView command:

1. Open Rhino and create a new file.
2. Select the stiffner when prompted and enter "250" for the step value.
3. Select the curve when prompted.
4. The script will create an Excel file with the height and angle data for each segment.

Troubleshooting:

1. If the script does not run, check that you have the correct version of Rhino installed on your computer.
2. If the script produces errors, check that you have selected the correct objects and entered the correct values for any prompts.
3. Check that you have the necessary permissions to read and write files on your computer.

## SmartDIM

Description:

This script create linear dimensions in Rhino. The script prompts the user to select the type of dimension (either TRV/LONG or DECK), the reference line, the parts border, and the location where the dimension will be placed.

How to use the command:

1. Open Rhino and type "EditPythonScript" in the command line.
2. Follow the prompts to select the dimension type, reference line, parts border, and dimension location.

Example:

1. Type the script name in the Rhino command line.
2. Select "1" to create a TRV/LONG dimension or "2" to create a DECK dimension.
3. Select the reference line by clicking on a point in the Rhino viewport.
4. Select the parts border by clicking on multiple points in the Rhino viewport.
5. Select the location where the dimension will be placed by clicking on a point in the Rhino viewport.
6. Repeat steps 3-5 for each part that needs a dimension.

Troubleshooting:

- If the prompts do not appear, check that the code has been copied and pasted correctly into the script editor.
- If the dimensions do not appear as expected, check that the reference line and parts border have been selected correctly.

## WT (quick weight calculator)

Description: The "peso" command is a Python script for Rhino that calculates the weight of a selected object based on its material density (specific weight) and volume. The script prompts the user to input the material density and then calculates the weight of each selected object and the total weight of all selected objects. The weight is then stored in the part's and the total weight is stored in the 3d model. A point and sphere are also added to the centroid of the selected objects.

How to use the command:

1. Open Rhino.
2. Select one or more objects to calculate the weight.
3. Input the material density in kg/m^3 when prompted.

Example:

1. Select an object.
2. Input the material density of the box material (e.g. 7850 for steel) when prompted.
3. The script will print the weight of the part and add a point and sphere to the centroid of the box.

Troubleshooting:

- If the script does not work, make sure that the specific weight of the material is input in kg/m^3.
- If the script does not add the weight to the object's user text or the document's user text, make sure that the object is not locked or that the document is not read-only.
- If the script does not add the point and sphere to the centroid of the selected objects, make sure that the objects are not hidden or that the point and sphere are not added off-screen.

# [NESTING]

## A9S (templates)

Description:

This script creates a flattened (2D) view of a BENT plate / hp 3D object and generates a cartouche (title block) with custom text in Rhino 3D. The cartouche is based on a pre-defined block in the Rhino file, and the text fields can be filled in by the user.

How to use the command:

1. Follow the prompts in the dialog box to select the object(s) you want to flatten and set the step size for the resulting flattened view.

Example:

1. Open Rhino 3D.
2. Select an object in the viewport.
3. When prompted, set the step size to 200.
4. The script will generate a flattened view of the object and create a cartouche in the viewport.

Troubleshooting:

1. If the script does not generate a flattened view or a cartouche, make sure that the object is selected and that the step size is set correctly.
2. If the script generates an error message, make sure that all of the required modules are installed and that the Rhino file contains the pre-defined cartouche.

# A7 (autosketchprofili)

This script creates a flattened (2D) view of a 3D object and generates a cartouche with custom text in Rhino 3D. It is used to generate uniform sections along a curved path in Rhino. It creates sections of equal length along the path, perpendicular to the curve. It can be useful for creating molds or profiles for CNC machines.

How to use the command:

1. Follow the prompts in the dialog box to select the object(s) you want to flatten and set the step size for the resulting flattened view.

Example:

1. Open Rhino 3D.
2. Select an object in the viewport.
3. When prompted, set the step size to 200.
4. The script will generate a flattened view of the object and create a cartouche in the viewport.

Troubleshooting:

1. If the script does not generate a flattened view or a cartouche, make sure that the object is selected and that the step size is set correctly.
2. If the script generates an error message, make sure that all of the required modules are installed and that the Rhino file contains the cartouche.