Introduction to Marxan
Course Handbook
Day 1

Based on materials developed by:
Matthew Watts and Lindsay Kircher

With additional materials from
Norma Serra-Sogas, Heather Coleman, Dave Nicolson,
Carissa Klein, Dan Segan, and Eddie Game

info@pacmara.org

These materials are provided by PacMARA.
Do not distribute without permission.

Last Updated October, 2013

Disclaimer: This manual is intended to be used in Introduction to Marxan courses provided by PacMARA and not as a stand-alone document. In addition, this manual only covers some of the basic functionalities of Marxan. Please consult the Marxan Manual (http://www.uq.edu.au/marxan) and Marxan Good Practices Handbook (http://pacmara.org/tikiwiki/tiki-index.php?page=Marxan+Resources+and+Training) for technical information and additional functionalities on Marxan.

Table of Contents

Exercise 1: Reserve Design Activity (without Marxan)

Natural resource managers have a limited budget to use for protecting and maintaining biodiversity. As a result, it is important that this money is used efficiently. In this activity, you will act as the natural resource manager, trying to create a comprehensive, adequate, representative, and efficient reserve system. Your region consists of 100 square planning units that are 100 hectares in size, and they contain 3 biogeoclimatic zones that you are trying to represent in your reserve system. There are several things to consider when creating your reserve system:

1. Meeting conservation targets;
2. Achieving a spatially "clumped" solution for ease of management and to minimize fragmentation; and
3. Minimising the cost of your reserve system.

In this activity, you will design a reserve network (by hand) that reaches your conservation targets in a cost effective manner while considering the spatial compactness of your solution. This activity uses the same kind of information that Marxan requires to find potential reserve networks, and the aim is for you to consider the methods and challenges of finding efficient solutions. At the end of this exercise we will see the solutions that Marxan provides to the same problem.

You will use the "maps" (Figures 1 and 2) on the next page to help you with this exercise. Figure 1 shows the planning unit id numbers and the cost of including that planning unit in the reserve system, and Figure 2 shows how much of each of the biogeoclimatic zones are contained in each planning unit.

Your objective is to represent 20% of the total biogeoclimatic zone area for the biogeoclimatic types 1, 2, and 3, in the study region (see Table 1) at least cost (refer to spreadsheet and Figures 1 and 2).

Table 1. Biogeoclimatic zone description, type number, area occupied, and target area

Biogeoclimatic zone description	Biogeoclimatic zone type	Total area	Target area
Coastal Western Hemlock	1	1337	267.4
Coastal Douglas-fir	2	1256	251.2
Mountain Hemlock	3	1215	243

❖ Navigate to the folder "C:\IntroMarxan\Reserve_design_exercise" and open the Excel file titled "Reserve_design_activity.xls"

This spreadsheet has information for each planning unit (total of 100 planning units), each with an assigned Planning Unit Identification Number or PUID, the cost of the planning unit (COST) and how much of each of the three biogeoclimatic zone types is contained in that planning unit (BZ_1, BZ_2, BZ_3).

You can select a planning unit to include it in your reserve system by changing the value in the "SELECTED" field from 0 to 1. To see the spatial information about the cost of each planning

unit, and the habitat occupied by each species in each planning unit, see Figures 1 and 2 (next page). When you select a planning unit to be included in your reserve system, the **summary information table** is updated to provide you feedback on the selections; that is, the SUM COST (the summed cost of the reserve system selected), and the TARGET GAP (the amount needed to meet the targets for the reserve system) are updated:

Summary information

SUM COST (total cost of all planning units selected)	0
TARGET GAP (summed amount in area required to meet remaining targets)	761.6

You can also track the individual species targets as you change the planning units selected:

Biogeoclimatic zone (BZ) type	BZ#	total area	target	amt held	target gap	target met
Coastal Western Hemlock	1	1337	267.4	0	267.4	NO
Coastal Douglas-fir	2	1256	251.2	0	251.2	NO
Mountain Hemlock	3	1215	243	0	243	NO

- ❖ Use the spreadsheet and Figures 1 and 2 to find a reserve system that meets all of your conservation targets in a cost effective (and clumped) way
- ❖ When you find your solution, calculate the boundary length (the number of free edges * 100) and the sum cost, and then record here _____

After you have completed the exercise, as a class we will compare how well our solutions perform compared to a Marxan generated solution.

Data used in exercise

1_347	2_52	3_985	4_207	5_276	6_821	7_122	8_404	9_300	10_681
11_813	12_537	13_931	14_653	15_919	16_826	17_455	18_983	19_731	20_875
21_247	22_462	23_287	24_988	25_85	26_736	27_681	28_479	29_459	30_615
31_378	32_986	33_887	34_392	35_526	36_783	37_224	38_149	39_268	40_90
41_977	42_74	43_53	44_390	45_619	46_773	47_952	48_738	49_897	50_580
51_969	52_76	53_147	54_870	55_350	56_543	57_607	58_375	59_903	60_790
61_729	62_492	63_303	64_289	65_490	66_599	67_407	68_651	69_709	70_365
71_571	72_931	73_353	74_64	75_955	76_950	77_855	78_886	79_840	80_840
81_422	82_252	83_941	84_152	85_353	86_123	87_716	88_587	89_346	90_318
91_682	92_891	93_815	94_818	95_726	96_372	97_197	98_89	99_417	100_975

Figure 1: Planning unit identifiers and cost (PUID_COST).

		0_0_1		89_0_12	30_48_0	69_4_9			0_0_91
				71_43_12	99_0_1		17_0_0	0_0_35	31_0_0
	55_40_0		0_2_27	70_0_0	37_0_56		0_0_3	0_41_0	54_0_0
	80_8_0	0_47_0		0_78_0	0_0_87	66_0_38			0_91_0
	0_0_73	0_60_0	25_79_0		11_0_8				0_0_53
76_34_0	0_90_0	0_84_0	0_0_82	0_72_26		0_0_21	58_0_0		0_54_59
75_0_60						91_0_0	0_0_57	0_42_97	0_0_7
	0_37_0						0_23_0	0_41_0	81_0_37
0_0_12			0_53_24	0_72_0	93_0_0		0_23_59		
	11_0_0	0_14_50		0_0_88			48_0_0		0_76_0

Figure 2: Area occupied by each biogeoclimatic zone in each square: BZ1_BZ2_BZ3

Exercise 2: Case Study Profile

Marine case study region: The Pacific Ocean between Hakai Pass and Port Hardy to the edge of the continental shelf, British Columbia, Canada.

Objective: Identify cost-effective areas for inclusion in a marine protected area network that protect a specified portion of each conservation feature.

Decision support tools: Marxan and Zonae Cogito.

GIS tools: There are many ways to create and view Marxan files. In this course, we present one method using ArcGIS, and another using the open course GIS software Quantum GIS (QGIS) with the QMarxan plug-in.

Disclaimer: This project is for training purposes only and is not intended to represent a full analysis of the conservation features and costs that would be necessary to provide options for a marine protected area network in this region. Conservation feature data were generously provided for this purpose by the British Columbia Marine Conservation Analysis (BCMCA) a collaborative project designed to provide information about marine biodiversity and human activity in BC's marine waters (www.bcmca.ca).

The data layers are:
1. *Study_Region_BC_Ocean:* Marine extent of study region
2. *Study_Region_BC_Land:* Land extent of study region
3. *Vessel_cost*: Vessel movement data (including fishing, carrier, ferry, tug, oil tankers, and cruise ships) that represents the cost to industry as a result of inclusion in marine protected area network. These cost values were extracted based on the square root of the number of vessel counts in a cell and then reclassified in four classes (more information about the original data can be found here).

The biodiversity features that we provide you with are:
1. *Benthic_BC*: layer that details the extent of a selection of benthic classes (55 total classes; or 16 classes in the QMarxan example) in British Columbia. The benthic classification is based on depth class, substrate, and slope. Each benthic class will be considered as one conservation feature (more information about the original data can be found here).
2. *Spngereefs_BC*: distribution data for sponge (more information about the original data can be found here).
3. *Bull_kelp_BC:* distribution data for bull kelp (more information about the original data can be found here).
4. *Giant_kelp_BC:* distribution data for giant kelp (more information about the original data can be found here).
5. *Kelp_gen_BC:* distribution data for general kelp (more information about the original data can be found here).

A. Creating Marxan input files in ArcMap

This process involves the creation of four files necessary to run Marxan. These are:

1. **pu.dat** - this is a text file that describes each planning unit, the cost of selecting that unit and the selection status of that unit.
2. **spec.dat** - this is another text file that describes the "species" or conservation features that you want to include in your analysis. This file contains the "species" id, the conservation target (either proportion or absolute value), penalties for not meeting targets (SPF) and the descriptive names for each conservation features.
3. **puvsp.dat** – this text file contains the amount of each "species" in each planning unit.
4. **bound.dat** – this file describes the adjacency relationships between planning units and their shared boundary length.

To start navigate to "C:\IntroMarxan\ArcMap_data" and double click on "IntroMarxan.mxd" to launch the ArcMap project.

❖ Ensure the Spatial Analyst extension is enabled by clicking on "Customize", "Extensions", then enabling "Spatial Analyst" by checking its box

❖ You may have to repair the data source if you see a red exclamation sign (**!**) next to each data layer.

❖ Explore each data layer so that you have a better understanding of the type of information that you will be working with in this course

Step 1: Creating Planning Units

Planning units refer to the grid and cell size that you will use in your analysis.

The size of the planning units is determined by the scale of the analysis, the resolution of the data used, and the objective of the planning exercise (see Marxan Good Practices Handbook for more information). Planning units can be squares, hexagons, or irregular shape polygons. With help of GIS you will calculate the area within each planning unit covered by each conservation feature distribution, and use these quantities to run a Marxan analysis.

Next you will create a square grid for the British Columbia dataset with 2 x 2 km^2 cells (2000 x 2000 m^2). Note that the GIS data for the study region are in main units of meters.
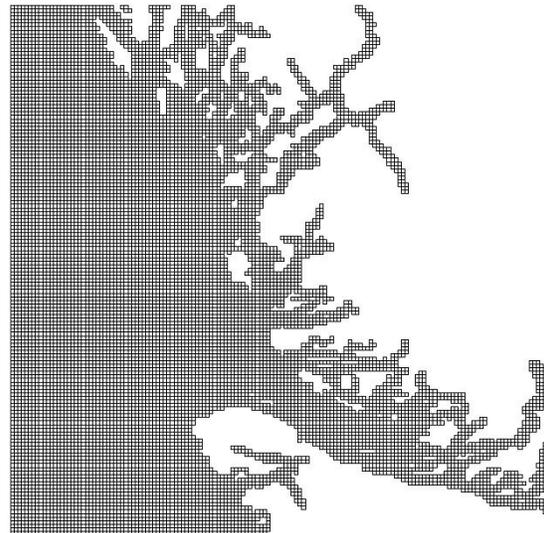
In this exercise you will learn how to create a planning unit layer using a free extension for ArcMap called ET Geowizards (http://www.ian-ko.com/). As an alternative, Jenness Repeating

Shapes (www.jennessent.com/arcgis/arcgis_extensions.htm) can also be used to create planning units as described in the Appendix of this manual.

Note: It is important to highlight that the different tools available to create grids or fishnets vary in the way they are produced and how they are indexed. For this reason it is important to explore the different tools and choose the one most suitable for the analysis at hand.

Step 1a: Creating Planning Units using ET GeoWizards
Note: there are many ways to create planning units – this is only one.

- ❖ Click on the yellow "ET" button on the ArcMap toolbar
- ❖ Click on the "basic" tab
- ❖ Click on "Vector Grid"
- ❖ Click "GO"
- ❖ Select "Study_Region_BC_Ocean" from the first drop down list box
- ❖ Click the browse button
- ❖ Browse to "C:\IntroMarxan\ArcMap_data"
- ❖ Type in "pulayer.shp" into the "Name" box, then click "Save". Ensure your windows have the parameters shown here:
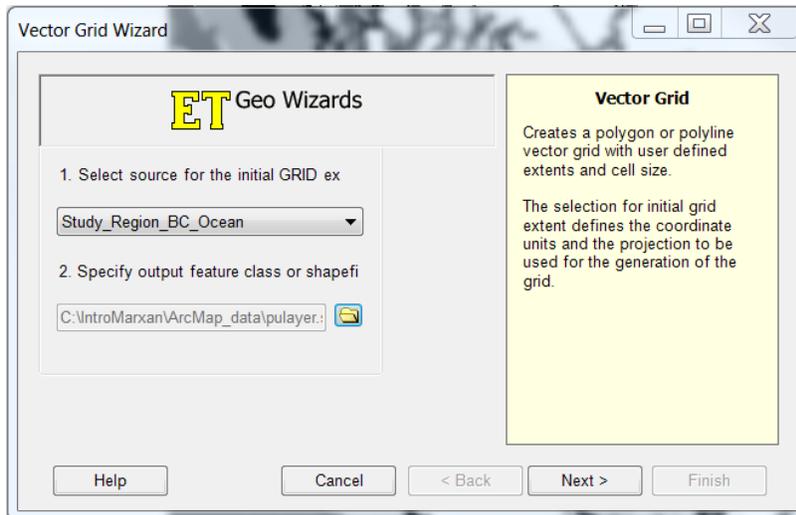
- ❖ Click "Next"
- ❖ Choose "Polygon" at the "Select GRID type"
- ❖ Click "Next"
- ❖ Here each unit in this British Columbia GIS dataset represents 1 m; thus (X=2000, Y=2000) means you are creating a grid of 2x2 km cells.
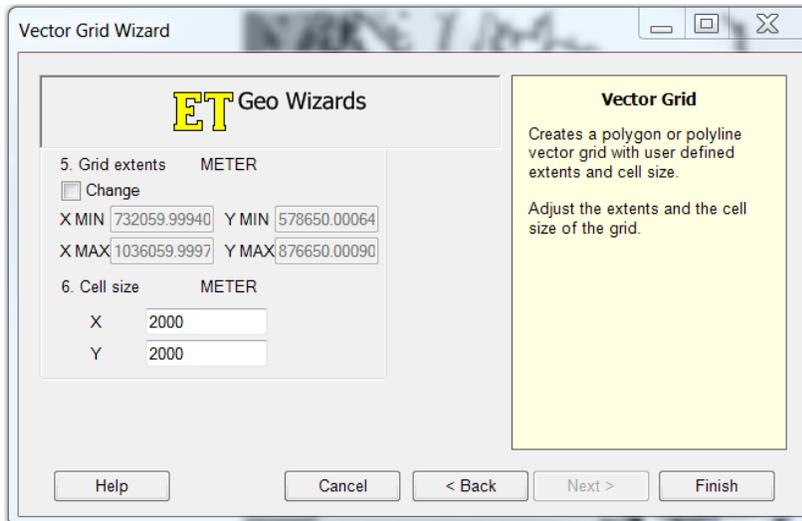
❖ Click "Finish"

You now have a planning unit grid for the entire study region, including the part that is on land.

Step 1b: Limiting the planning units to your study region.

Because we are only considering those planning units that intersect with the ocean and not the land, we are going to perform a selection to only select the planning units that intersect with the layer Study_Region_BC_Ocean, and later export that selection to create a new shapefile of planning units.

❖ In the main menu, choose "Selection," then "Select by Location"

❖ In the first drop down box, choose "select features from"

❖ Tick the box next to "pulayer"

❖ Choose the "Study_Region_BC_Ocean" in the second drop down box

❖ Choose the "Target layer(s) features intersect source layer features" in the third drop down box.

❖ Click "OK"

⇒ **Note that in ArcGIS 9.x** you should choose "intersect" in the second drop down box and "Study_Region_BC_Ocean" in the third drop down box
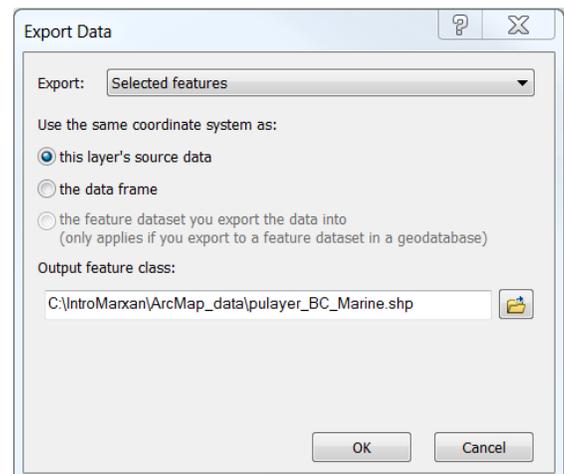
You can see that the planning units that intersect the Study_Region_BC_Ocean layer are selected (now highlighted in blue). Now you will create a new shapefile only using these planning units.

❖ Right click on "pulayer"

- ❖ Choose "Data", then "Export Data"

- ❖ Type "C:\IntroMarxan\ArcMap_data\pulayer_BC_Marine.shp" to name your new planning unit layer

- ❖ Ensure that your axport window matches the parameters in the figure to the right.

- ❖ Click "OK"

- ❖ Click "yes" to add the exported data to the map as a layer

- ❖ Choose "Selection" from the main menu, then "Clear Selected Features"

If you cannot see your new layer, turn off the pulayer layer, and you should be able to view pulayer_BC_Marine.

Step 1c: Creating planning unit unique ID field

Marxan requires each planning unit to have a unique identifier or "planning unit id" or "puid". In this step you will assign a unique ID to each planning unit.

- ❖ Right click on "pulayer_BC_Marine" and open the attribute table ("Open Attribute Table")

- ❖ Click "Table Options" (top-left corner of the table) then "Add field"

- ❖ Select "Long Integer" as the "Type"

- ❖ Name the field "PUID" and click "OK"

- ❖ Right click on the column title "PUID" and click "Field Calculator"

- ❖ Under "Fields," double-click on "FID" (so it appears in the calculation box), then on the "+" sign (on right hand side of window) then type in the number "1", and then Click "OK". This assigns a unique ID to each planning unit, starting with 1 instead of 0.

- ❖ Delete unnecessary fields by right clicking on the fields and select "Delete Field": "Id", "ET_ID", and "ET_Index"

- ❖ Close the table

Now you have created your final planning unit shapefile. This will be used to create the files: pu.dat, puvsp.dat and bound.dat. It will be also used by Zonae Cogito (Day 2) to visualize Marxan results.

Step 2: Creating the planning unit file (pu.dat)

Now you will be creating the pu.dat file that contains the planning unit id, cost (of that planning unit), and status (availability for selection).

Step 2a: Creating cost table

We have provided a GIS layer of vessel movement values to represent cost to industry if a planning unit is allocated to a marine protected area network. Here, the value for cost is high if there are many vessels moving through that planning unit (e.g. class 4), and lower elsewhere (e.g. class 1). You will calculate the expected cost of selecting each planning unit assuming that you are removing vessel access to a planning unit that is selected for reservation. Think about this cost

id	cost	status
1	256	0
2	320	0
3	320	0
4	320	0
5	320	0
6	320	0
7	320	0
8	320	0
9	320	0
10	320	0
11	320	0
12	320	0
13	320	0
14	320	0
15	320	0
16	320	0
17	320	0
18	320	0
19	320	0
20	320	0

layer. If you were doing this analysis, would you use this cost layer? Why or why not? What cost data might be more useful? Why do you think we did not use those data here?

Note: Please ensure that the tables for your layers "pulayer_BC_Marine" and "Vessel_cost" do not have any rows selected or the "Zonal Statistics" function will not work correctly

❖ Under "Spatial Analyst Tools" in the toolbox, "Zonal", click on "Zonal Statistics as Table"

   If you can't get to Spatial Analyst Tools, click on "Customize", "Extensions", then enabling "Spatial Analyst" by checking its box.

   ⇒ In **ArcGIS 9.X**, you will click on "Tools" instead of "Customize"

❖ Type "C:\IntroMarxan\ArcMap_data\zonalst_cost.dbf" in the Output table box

❖ Select "SUM" as the option for "Statistics Type"

   ⇒ In **ArcGIS 9.x,** you might have or not this option.

❖ Ensure your "Zonal Statistics as Table" window has the parameters shown in the figure to the right.

❖ Click "OK"

This will create a table (ZonalSt_cost.dbf) that sums the vessel value (cost to industry) contained in each planning unit (indexed by PUID).

Step 2b: Link data layers to create table for export
Note: If you cannot see the table in ArcMap, you will need to select the "List By Source" tab in ArcMap 10.

❖ Ensure that "Zonalst_cost.dbf" (cost of each planning unit) is added to ArcMap – to do this, click on the yellow sign " ✛ ", or right click the layer window and click "Add Data"

❖ Now we will join the information in this table to the information in the planning unit file

❖ First, right click on "pulayer_BC_Marine" and select "Joins and Relates" and then "Join". In the first drop menu select the field "PUID", in the second the table "ZonalSt_cost", and in the third select the field "PUID"

   ⇒ Note that in **ArcGIS 9.x**, the field in the table to base the join on in the third table should be set to VALUE rather than PUID.

12
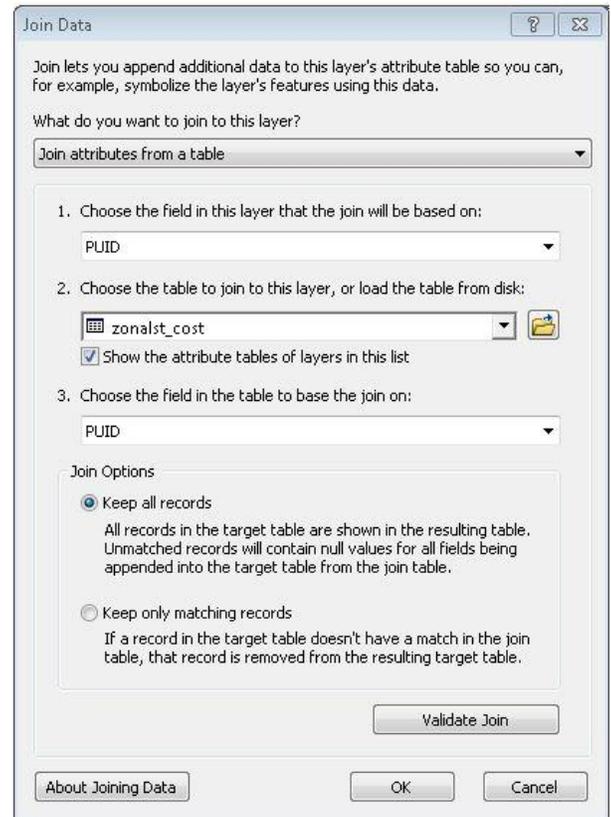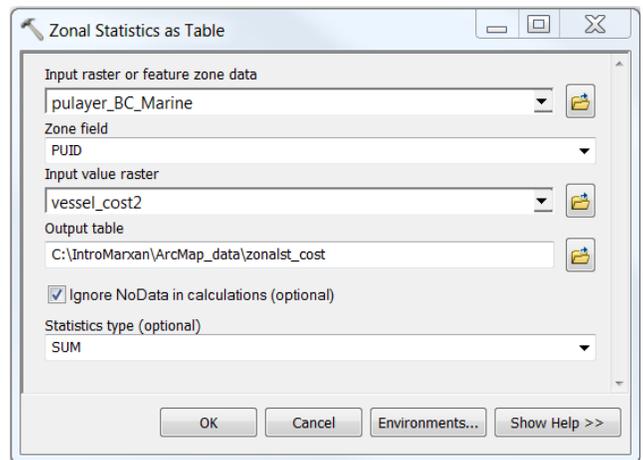
❖ Ensure your Join window has the following parameters shown in the figure in page 18.

❖ Click "OK"

⟹ In **ArcGIS 9.x** click "Yes" on the "Create Index" box that appears

Now the planning unit and cost data are joined, but you need to put this information into a format that Marxan can use. You will do this by exporting the data so you can open and edit it in Excel.

Step 2c: Export data to create planning unit file

You have now linked data on planning unit (PUID) and cost using the Join Data tool. This will let you export the cost and planning unit information into one file.

❖ To export the joined data to a new table, right click on "pulayer_BC_Marine", select "Data", then "Export Data"

❖ Type "C:\IntroMarxan\ArcMap_data\Export_puJoinCost. shp" in the "Output feature class:" box to rename the shapefile

❖ Ensure your Export window matches the figure to the right.

❖ Click "OK"

❖ Click "No" when asked to add exported data to map as a layer, this is so we can open it in Excel

❖ Launch Excel. Open "Export_puJoinCost.dbf" in Excel (make sure that you have selected "show all file types" so that the .dbf files are visible)

❖ Delete all columns EXCEPT: "PUID," and "SUM", which refer to your planning unit index and the cost of each planning unit

❖ Rename "PUID" to "id" (must be in lower case letters), as this is what it must be called for Marxan to run

❖ Rename "SUM" to "cost" (must be in lower case letters), as this is what it must be called for Marxan to run (**sum is the area weighted cost of each planning unit**)

❖ Select the "cost" column, and reformat the cells to remove extra decimal places (right click, "Format Cells…", "Category: Number", "Decimal places: 0")
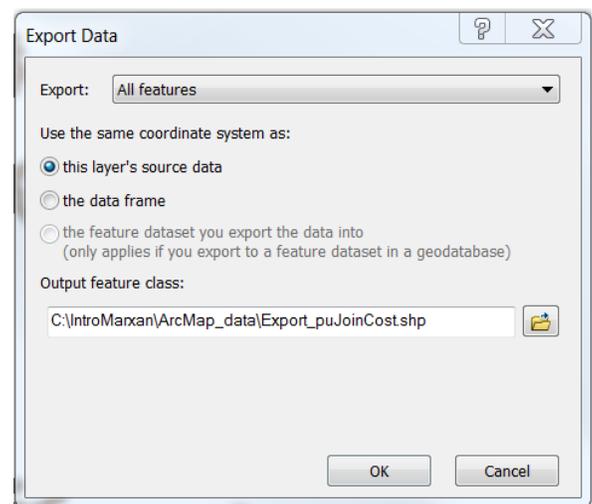
Now you have finished editing the first part of the pu.dat file, the information that indicates how much it costs for each planning unit to be included in a Marxan solution. Next you will add information about the status –whether or not a planning unit is available for selection.

Step 2d: Setting the planning units status

In Marxan, currently protected areas can be locked into the solution set to reflect that they are unlikely to be changed or moved. To do this, the planning unit status must be assigned a "2" in

the planning unit file (a status of "0" is available; "3" is locked out; "2" is locked in). You will have the opportunity to lock planning units in or out of solutions in Day 2. You will now assign the status to all planning units by giving them a status of "0", that is you will leave them all available for selection.

Still working on Export_puJoinCost.dbf in Excel:

❖ Click on cell C1 (should be blank if you deleted the correct number of fields), and type in "status" (must be in lower case letters)

❖ Click on cell C2 and type in a "0" to set the status to available for selection

❖ In cell C2, double click the bottom right hand corner of the cell to automatically fill the "0" value down the entire column

❖ Autofit the entire table (select the entire table and double click on the line between columns A and B)

❖ Click File, Save As, and select "Text (Tab delimited)" type and name the file "pu.dat"

❖ Ensure that your Save As window matches the figure to the right.

❖ Click "Save"

❖ Click "Yes" on the pop up dialog box

❖ Close the "pu.dat" file in Excel

❖ Click "No" on the pop up dialog box

❖ Double check that the saved file ends with the extension **.dat** by visiting the location where it was saved "C:\IntroMarxan\ArcMap_data\pu.dat".
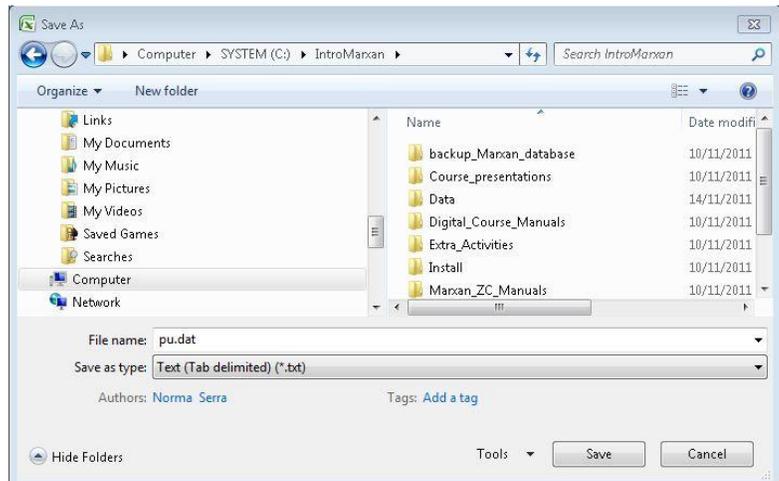
Now you have completed your pu.dat file. You have formatted it in this way and saved it as a .dat file because Marxan requires specific formatting and file types to work correctly. The pu.dat file contains information for each planning unit on the cost of selecting it for inclusion in a Marxan solution, as well as the status for selection (locked in/out or available for selection).

❖ Go to "C:\IntroMarxan\ArcMap_data\pu.dat" and open it with a text editor to review the information contained in the file and its format

id	name	prop	spf
1	0_20 Hard Depression	0.3	1
2	0_20 Hard Flat	0.3	1
3	0_20 Hard Ridge	0.3	1
4	0_20 Hard Slope	0.3	1
5	0_20 Muddy Depression	0.3	1
6	0_20 Muddy Flat	0.3	1
7	0_20 Muddy Ridge	0.3	1
8	0_20 Muddy Slope	0.3	1
9	0_20 Sandy Depression	0.3	1
10	0_20 Sandy Flat	0.3	1
11	0_20 Sandy Ridge	0.3	1
12	0_20 Sandy Slope	0.3	1
13	20_50 Hard Depression	0.3	1
14	20_50 Hard Flat	0.3	1
15	20_50 Hard Ridge	0.3	1
16	20_50 Hard Slope	0.3	1
17	20_50 Muddy Depression	0.3	1
18	20_50 Muddy Flat	0.3	1
19	20_50 Muddy Ridge	0.3	1
20	20_50 Muddy Slope	0.3	1
21	20_50 Sandy Depression	0.3	1
22	20_50 Sandy Flat	0.3	1
23	20_50 Sandy Ridge	0.3	1
24	20_50 Sandy Slope	0.3	1

Step 3: Creating the species file (spec.dat)

In this step you will be creating the species file. This is called the "species file" because when Marxan was first created, all of the conservation features were individual species. Now we call them "conservation features", but the file that lists them and their targets is still called the species file or spec.dat.

This file lists the conservation features (each with a unique identifier or "id") and the target amount for that conservation feature (in this case, as a proportion of the total in the region or "prop") to be included in the reserve system. The "spf" column refers to the species penalty factor. You can also include the name of the conservation feature in the field "name" to help you keep track (many analyses will have hundreds of conservation features).

For this exercise we provide you with:

1. A layer that details the extent of benthic classes in the study region (*Benthic_BC*). Each of the 55 benthic classes is a conservation feature.
2. Layers detailing the distribution of sponge reefs and 3 different kinds of kelp: *spngereefs_BC, giant_kelp_BC, bull_kelp_BC*, and *kelp_gen_BC*.

First, we will calculate the total area covered by each of the 55 benthic classes using the *Benthic_BC* layer, and the individual conservation feature distributions using their corresponding layers (for a complete list of features included and a description see page 12). Then we will set a target of 30% protection of each conservation feature distribution using the "prop" field in the spec.dat file (i.e., 0.3 of the total area calculated for that conservation feature). We will leave the spf (species penalty factor) as the default value "1". During Day 2 exercises you will have the opportunity to calibrate this value. Finally, you will repeat each step for each of the remaining conservation features: coral reefs and the 3 different kinds of kelp.

Note: Each of the conservation features has been pre-assigned a unique identifier. There are many ways that this can be done using GIS software. These data were prepared by giving each conservation feature a unique id when it was in feature class shapefile format, then it was converted to raster using this id. If you are unsure about how this might be accomplished, please ask your instructor for an explanation at this time.

Step 3a: Export conservation feature layers to Excel

First, we will export the information from Benthic_BC (benthic classes) and open it in Excel:

❖ In ArcMap, right click on the "Benthic_BC" layer and click on "Open Attribute Table"

❖ Click on "Table Option" in the upper left corner of the table, then "Export"

⇒ In **ArcMap 9.x** click on "Options", then "Export"

❖ Type in "C:\IntroMarxan\ArcMap_data**Export_benthic.dbf**"
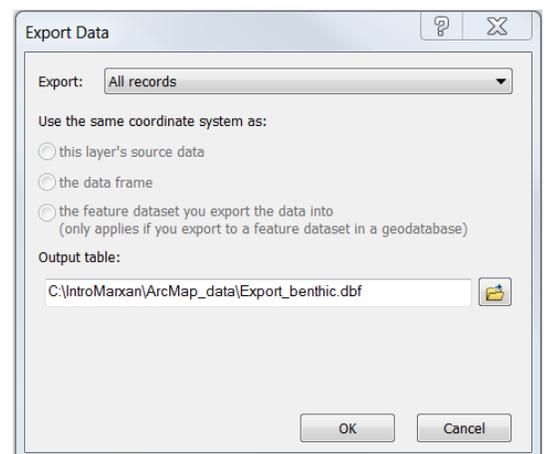
❖ Be sure that your Export window matches to the right.

❖ Click "OK" and *do not* add layer to the map, when prompted

You will now create the conservation feature file, spec.dat.

❖ Open "Export_benthic.dbf" in Excel. The values in the first column "VALUE" are the species id, the values in the second column "COUNT" indicate how many pixels there are for each specie (or conservation feature) in the benthic data layer, and the values in the third column "NAME" show the names for each of the species.

❖ Rename "VALUE" to "id" (must be in lower case letters), as this is what is required in Marxan

❖ Delete column B

In the "NAME" column, you will indicate the actual name of the species or conservation feature by following these steps:

❖ Rename cell C1 from "NAME" to "name" (must be in lower case letters)

❖ Use the find and replace function to replace all "–" with "_" (Marxan does not accept the

"–" symbol); do the same with the "+" symbol, replacing with the word "plus" (you may need to untick "match entire cell contents" under "Options")

The species file is the file in which you must set a target amount for each conservation feature to be included in the solution. We will set a target of 30% for each conservation feature by doing the following steps:

❖ Click in cell C1 and name it "prop" (must be in lower case letters)

❖ Type "0.3" in cell C2, (this will set the target at 30%)

❖ In cell C2, double click the bottom right hand corner of the cell to automatically fill this function down the column

Now you have unique ids, names, and targets for each conservation feature. The last step is adding the default value for the species penalty factor (spf).

Step 3b: Create species penalty factor

In the species file, you indicate the species penalty factor (spf) for each species (you will learn more about the use of and the meaning of spf later in this course). For this analysis, we will use a value of "1" for all species/habitat types.

❖ Click in cell D1 and name it "spf" (must be in lower case letters)

❖ Click in cell D2, type "1" (this will be the spf value for each species), and double click the bottom right hand corner of the cell to automatically fill this function down the column (this is the default value for spf)

Step 3c: Add individual conservation features layer and associated values to the spec table

You will now need to add information about the additional conservation features to the spec table (i.e., bull kelp, giant kelp, general kelp, and

☑ bull_kelp_bc
■ 57
☑ giant_kelp_bc
■ 58
☑ kelp_gen_bc
■ 59
☑ spngereefs_bc
■ 56
☑ benthic_bc
Value
High : 55

Low : 1
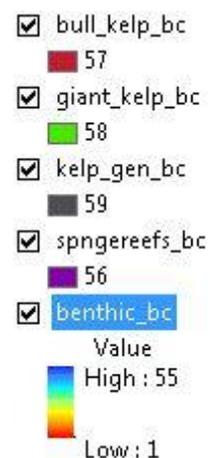
sponge reefs). Check the id for the additional conservation features. Your benthic features have ids from 1-55. Look at the Table of Contents in ArcMap (or the screenshot on the right hand side) to determine the conservation feature ids for your other conservation features (e.g., for sponge reefs "spngereefs_bc" the id number is 56).

At the bottom of your species file, add the appropriate conservation feature id, as well as the target and spf values that were entered for the benthic classes (prop "0.3", and a spf of "1").

Step 3d: Save completed Marxan species file

❖ Autofit the entire table (do this by selecting the entire table and double clicking on the line between columns A and B)

❖ Click File, Save As, save in "C:\IntroMarxan\ArcMap_data\" and select "Text (Tab delimited)" type and rename the file to "spec.dat"

❖ Click "OK" and click "Yes" on the pop up dialog box

❖ Close the "spec.dat" file in Excel

❖ Click "No" on the pop up dialog box

❖ Double check that the saved file ends with the extension **.dat** by visiting the location where it was saved "C:\IntroMarxan\ArcMap_data\spec.dat".

Now you have created the spec.dat, the file that tells Marxan the unique identifier for each of your conservation features, the name of your conservation features, the target you want to meet for each conservation feature, and the species penalty factor (spf) value (that you will calibrate tomorrow).

Step 4: Creating the planning unit versus species matrix (puvsp.dat)

In this exercise, you will determine how much of each of the conservation features (benthic classes, sponge reef, and kelp) distribution is contained in each planning unit.

```
species,pu,amount
37,1,470000.000000
39,1,3480000.000000
40,1,50000.000000
39,2,4000000.000000
37,3,10000.000000
39,3,3520000.000000
40,3,470000.000000
37,4,1090000.000000
38,4,830000.000000
39,4,910000.000000
40,4,1170000.000000
37,5,170000.000000
38,5,2390000.000000
39,5,50000.000000
40,5,1390000.000000
```

Step 4a: Calculate area in each planning unit covered by each benthic class

Note: Please ensure that the tables for your layers "pulayer_BC_Marine" and "Benthic_BC" do not have any rows selected or the tabulate areas function will not work correctly.

❖ In ArcMap, right click on "pulayer_BC_Marine", select "Joins and Relate" then "Remove Joins" then "Remove All Joins"

❖ Under Spatial Analyst Tools, Zonal, click on "Tabulate Area"

❖ Here you are calculating how much of each habitat type is in each planning unit. The "Input raster or feature zone data" is your planning unit layer, and you want to make the area calculation for each planning unit, so you choose the PUID field as your "Zone field". The "Input raster or feature class data" for the third drop down box is the conservation feature

dataset you want to calculate the amount of for each planning unit. Finally, you need to choose the "Class field" that identifies each conservation feature – the value field.

❖ Ensure your Tabulate Area window has the following parameters:
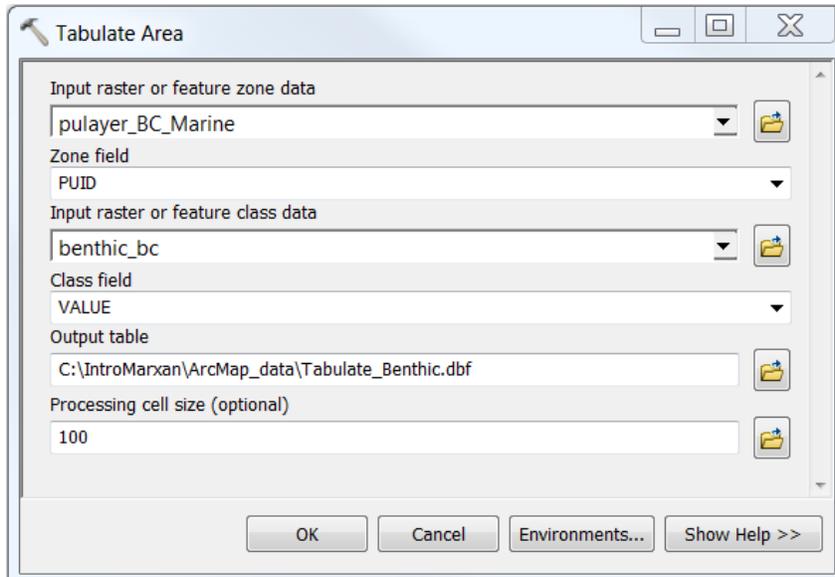
❖ Click the browse button to specify the output table, type "Tabulate_benthic.dbf", then click "OK"

❖ Ensure the "Output table" says "C:\IntroMarxan\ArcMap_data\Tabulate_benthic.dbf" – it must be in the .dbf format, not the default format

❖ Check to make sure that you are saving the .dbf file in the IntroMarxan folder

Step 4b: Calculate area in each planning unit for remaining conservation features

Repeat the tabulate area steps above for the remaining conservation feature datasets to calculate how much of each of the conservation features is contained in each planning unit. Name the tabulate area files something that makes sense for each feature (for example, for the sponge reefs, **tabulate_reef.dbf**.).

You can follow the directions above for the benthic classes to do this, just keep the feature zone data (pulayer_BC_Marine) and the Zone Field (PUID) the same, and vary the input raster and output table names. **Do not forget to save file as .dbf format.**

Step 4c: Join tables and export to Excel

❖ Ensure that all of your .dbf tables are added to ArcMap

❖ If you cannot see the tables in ArcMap, you will need to select the "List by Source" tab from the ArcMap 10 table of contents (or the "Source" tab in ArcMap 9.x)

❖ Join all of the .dbf tables you just created **to each other** (be careful – don't join them to the planning unit layer, you have already calculated the amount of each conservation feature in each planning unit, so you do not need to join the results back to the planning unit layer)

Note: the common information in all of the conservation feature tabulate area .dbf tables is the planning unit id or PUID – you will be joining them based on this field

❖ To do this, right click on "Tabulat_benthic", select "Joins and Relates", "Join", and join to the tabulate_reef.dbf.

❖ Be sure that your Join Data window has the following parameters in the figure to the right (you may need to select "value" if "puid" is not an option).

❖ Click "OK"

❖ Now you will need to join the other tabulate area .dbf files for the other conservation features to the Tabulate_benthic.dbf table. Join them in conservation feature number order (i.e., (1)Tabulate_bullkelp, (2) Tabulate_giantkelp, (3) Tabulate_genkelp). Use PUID field to join the different tables.

❖ Make sure you join ALL of the other conservation features to the Tabulate_benthic.dbf

❖ Export the joined data to a new table. Right click on "Tabulate_benthic", then "Data", and save the table in "C\IntroMarxan\ArcMap_data\" as "**tabulate_all_conservationfeatures.dbf".**

Note: Do not add this table to the map, or we won't be able to open it in Excel.
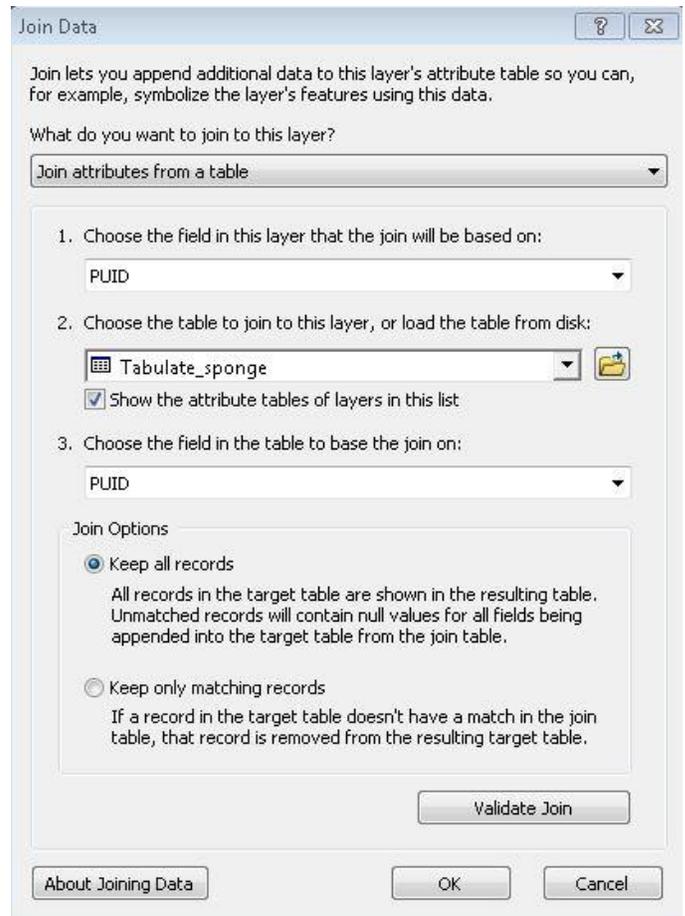
NOTE: Data validation is of utmost importance when constructing Marxan datasets. Your reputation, your organisation's reputation, and the reputation of the Marxan software is at stake if the data used in a planning process turn out to have errors in it. This is an unfortunate situation you can avoid with simple validation steps. This is one way to validate your calculations:

❖ Validate the calculation by overlaying the Benthic_BC GIS layer with the planning unit layer to check if the results in the table correlate with the spatial data. Some GIS skills are necessary to perform this task.

❖ Choose several planning units at random to perform these validation tasks
  • Zoom into the planning unit you have selected
  • Use the identify tool 🛈 to obtain the PUID of the planning unit
  • Record the PUID (ensure that you are looking at the puid and not the FID when you record this)
  • Overlay the Benthic_BC layer on the planning unit

- Use the identify tool 🛈 to obtain the VALUE identifier of one of the benthic types (one that only has a few grid cells will be easier to count)
- Write the benthic type identifier down (e.g. 55)
- Count how many grid cells of the benthic type appear in your planning unit
- Record the number of grid cells of that benthic type
- Ensure the spread sheet entry for the chosen planning unit (puid) and benthic type (eg. 55) pair has the correct value (that is 100 * 100 * grid cell count = value of spread sheet)
- You can repeat this procedure using the one of the other conservation feature layers (e.g. spngereefs_BC)

Once you have validated that the datasets you have created are consistent with the original datasets, you need to continue processing the Excel spreadsheet that will become the puvsp.dat file.

❖ Open tabulate_all_conservationfeatures.dbf in Excel and explore results. Each conservation feature will be listed in the top row (e.g. VALUE_1) and each planning unit will be listed in the PUID field. The area (meters squared) of each conservation feature was calculated for each planning unit.

❖ If you scroll to column BE and BF you will find the columns "OID_1" and "PUID_1". This is where the table Tabulat_reefs was joined to Tabulate_benthic. There are columns called "OID_X" and "PUID_X" where each of the other conservation features were joined to Tabulate_benthic. Delete all of these columns after BE (including BE), EXCEPT ones that start with the word VALUE_. For example, delete column BE (OID_1) and BF (PUID_1) but leave VALUE_56 because this column contains information about how much of the sponge reef conservation feature is contained in each planning unit. At the end, column A will be called "PUID", and all of the other columns will be "VALUE_XX" (where XX is conservation feature id 1-59).

❖ Rename "PUID" to "pu" (must be lowercase), as this is required in Marxan

❖ Now, select row 1 and find the "Find and Replace" tool (or hold ctrl and press the F key)

❖ Be sure that your Find and Replace window has the following parameters:
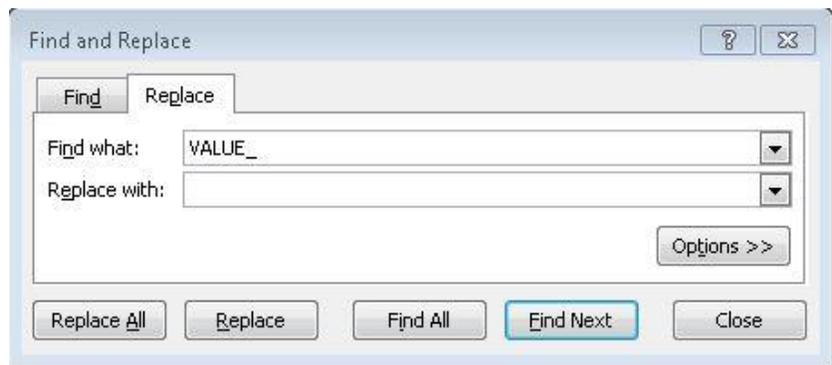
❖ Click "Replace All".

Now, we should get rid of the extra zeros after the decimal place as they are not needed:

❖ To get rid of the extra zeros after the decimal place, select the entire table

❖ Click "Format", "Cells"

❖ Be sure that your Format Cells window has the following parameters in the figure to the right.

❖ Click "OK"

❖ **Save the file in "C:\IntroMarxan\"** as "Text (Tab delimited)" type and rename the file to "**puvsp.dat**"

❖ Click "OK"

❖ Click "Yes" on the pop up dialog box

❖ Close the puvsp.dat file in Excel

❖ Click "No" on the pop up dialog box

❖ Double check that the saved file ends with the extension **.dat** by visiting the location where it was saved "C:\IntroMarxan\puvsp.dat". Note that if you do not have admin access on the lab computer, you may need to actively save the file as "puvsp.txt" and manually change the ending to ".dat"
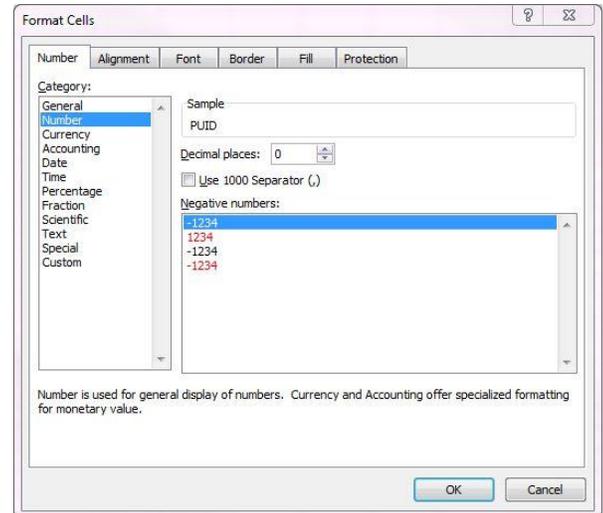
Example of the "Puvsp.dat" file in "old" Marxan format:

```
pu     1       2       3       4       5       6       7
1      0       0       0       0       0       0       0
2      0       0       0       0       0       0       0
3      0       0       0       0       0       0       0
4      0       0       0       0       0       0       0
5      0       0       0       0       0       0       0
6      0       0       0       0       0       0       0
7      0       0       0       0       0       0       0
8      0       0       0       0       0       0       0
9      0       0       0       0       0       0       0
10     0       0       0       0       0       0       0
11     0       0       0       0       0       0       0
```

The puvsp.dat you have just created is in the old Marxan format (see above). To run with the optimized version of Marxan, and to use ZC, we need to convert the file format using the convert _mtx.exe tool. You will find the tool, and directions for use in the C:\IntroMarxan folder.

Note: Make sure that the conversion tool convert _mtx.exe and the file puvsp.dat are in the same folder C:\IntroMarxan

❖ From the start menu, select "Start" > "Run". From the run window, type "cmd", and click "OK"
❖ **Note:** in Windows 7 the "Run" window is launched through "Accessories"
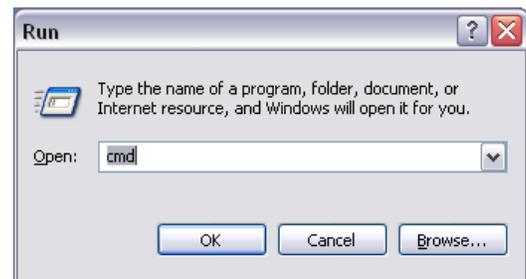
You should now see the command window, at the command prompt:

❖ Type "cd c:\ IntroMarxan" (or the full pathname where data resides) and click return to change directories into the folder where the puvsp.dat file is located

❖ Type "convert_mtx.exe 1 puvsp.dat puvsp2.dat" to execute the command

❖ The conversion tool will then convert the file into the proper format (see below):


```
species,pu,amount
37,1,470000.000000
39,1,3480000.000000
40,1,50000.000000
39,2,4000000.000000
37,3,10000.000000
39,3,3520000.000000
40,3,470000.000000
37,4,1090000.000000
38,4,830000.000000
39,4,910000.000000
40,4,1170000.000000
37,5,170000.000000
38,5,2390000.000000
39,5,50000.000000
40,5,1390000.000000
```

❖ Open puvsp2.dat (in C:\IntroMarxan) with a text editor software (such as notepad). Note the change of formatting of the file (sample above to the right).

❖ Copy the new puvsp2.dat in the folder C:\IntroMarxan\ArcMap_data

Step 5: Creating the bound.dat file

Create boundary file using Marxan tool. This file indicates the spatial relationship between the planning units and is essential for solutions that require spatial compactness. Id1 indicates the PUID for one planning unit, and id2 is the PUID for an adjacent planning unit. The boundary is the measurement of the boundary between the two planning units.

id1	id2	boundary
1	2	2000
1	73	2000
1	1	4000
2	3	2000
2	74	2000
2	2	2000
3	4	2000
3	75	2000
3	3	2000
4	5	2000
4	76	2000
4	4	2000
5	6	2000
5	77	2000
5	5	2000
6	7	2000
6	78	2000
6	6	2000
7	8	2000
7	79	2000
7	7	2000
8	9	2000
8	80	2000

Step 5a: Using ABP Marine Environmental Research (when using ArcGIS 10)

❖ Ensure the ABP toolbar is showing, in ArcMap as per the installation instructions (circled in the screenshot below)

❖ Select "pulayer_bc_marine.shp" fom the ArcMap TOC and click the ABP tool

- ❖ Select "PUID" as the PU field and click "Create file"
- ❖ Save and rename the file created to "bound.dat"
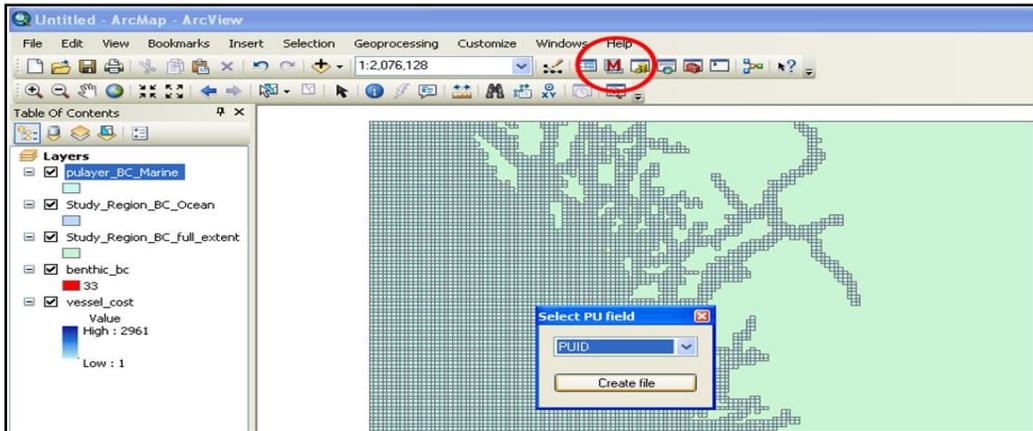
Step 5a: Using JNCC (when using ArcGIS 9x)

To ensure the JNCC toolbar is showing, in ArcMap, click "View", "Toolbars", "JNCC ArcGIS Extensions".

- ❖ ****Ensure**** that pulayer_BC_Marine is the **only** layer on your layer list (remove all other layers) or the program may not execute (yes, it is picky). To make sure all other layers and tables are removed, click the display and source tabs in the GIS layers window of your ArcMap project. If there are any layers or tables other than the pulayer_BC_Marine in under either of these tabs, remove them from the map.
- ❖ Click the Create Marxan File button on the ArcMap tool bar 🖳
- ❖ Type in "C:\IntroMarxan\ArcMap_data**bound.dat**" in the "Enter the output filename" box.
- ❖ Be sure that your Create Marxan File window has the following parameters:
- ❖ Click "OK"
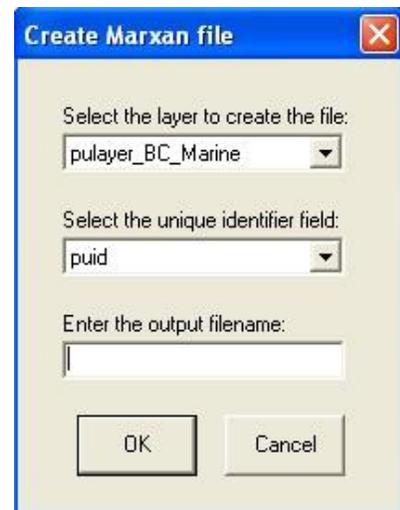
Step 5b: Inspect the boundary

We will now inspect the boundary file we have just generated.
- ❖ Open Microsoft Excel
- ❖ From Excel, use file > open to open "bound.dat", import the file as a comma separated value file
- ❖ You may see that the boundary file contains a lot of significant digits; for our analysis, we will round the length of each boundary to the nearest meter – to do this, format the cells in the boundary column by selecting "number," then "0" decimal places in the drop down menu

❖ Save the changes to the file, but keep the file in the tab delimited format – make sure to close the file when you are done

❖ Browse to the " C:\IntroMarxan\ArcMap_data" folder and right mouse click on the "bound.dat" file, selecting "Open" with Notepad

❖ Scroll to the end of the file and delete the blank line as Marxan may not run with this extra blank line

❖ Save the file and exit the program

Note: The boundary maker tool has a sporadic glitch that sometimes results in malformed rows. For example, if you see a row in your file that ends in ",5000." do a search and replace with notepad to replace "5000." with "5000". In some cases, Marxan will give an error message when running with a boundary file that contains these malformed rows. The problem here is that there are no zeroes following the decimal place, which confuses the program when it tries to convert the text into a number.

Step 6: Editing the input file (input.dat) file

Congratulations! You are done creating the data files with ArcMap, and will now edit the input file for running Marxan, and organize the files you've just generated into the standard format.

The **input.dat** file is like a list of instructions for Marxan where you will indicate the name and location of your other input files and set several parameters. A detailed description of all the parameters that can be set in the input.dat is in the Marxan manual, which is in the documents section of your course folders. Note that we will not cover all of these parameters in this course.

First, you will need to establish a specific structure for your files:

❖ Browse to the IntroMarxan folder, create a new folder and name it "Marxan_Database"

❖ In the Marxan_Database folder, create three new folders called "input", "output", and "pulayer"

❖ Copy the "Marxan.exe" file into the "Marxan_Database" folder if your computer is 32-bit, or "Marxan_x64.exe" if your computer is a 64-bit.

❖ Finally, copy the input.dat file provided into the "Marxan_Database" folder.

❖ The files in the "Marxan_database" should appear as in the figure below:
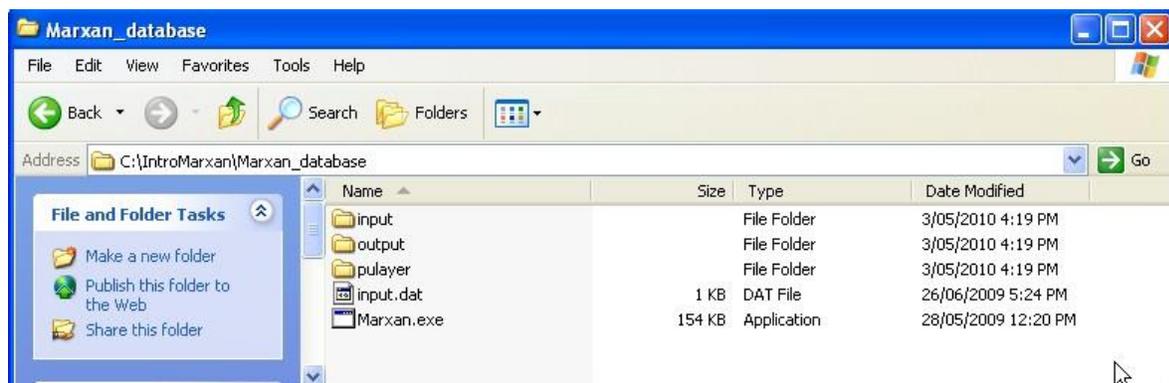
❖ Copy the following files into the input folder: "bound.dat", "pu.dat", "puvsp2.dat" and "spec.dat"

❖ Copy all 7 the files associated with the layer "pulayer_BC_Marine" into the pulayer folder (i.e.,"pulayer_BC_Marine.shp", "pulayer_BC_Marine.shx", "pulayer_BC_Marine.dbf", "pulayer_BC_Marine.prj", "pulayer_BC_Marine.sbn", "pulayer_BC_Marine.sbx", "pulayer_BC_Marine.shp.xml")

❖ Then, in the Marxan_Database folder, open "input.dat" in Notepad. This input.dat is the same as the sample input.dat you receive when you download Marxan or Zonae Cogito. Some of the parameters specified in this file will be set by you. These parameters are indicated in the example input.dat (see next page).

```
Input file for Marxan program, written by Ian Ball and Hugh Possingham.
iball@maths.adelaide.edu.au
hpossing@maths.adelaide.edu.au

This file generated by cplan.exe
emailto: cplan@ozemail.com.au

General Parameters
VERSION 0.1
BLM 1
PROP  0.5
RANDSEED -1
BESTSCORE  10
NUMREPS 10

Annealing Parameters
NUMITNS 1000000
STARTTEMP -1.00000000000000E+0000
COOLFAC  6.00000000000000E+0000
NUMTEMP 10000

Cost Threshold
COSTTHRESH  0.00000000000000E+0000
THRESHPEN1  1.40000000000000E+0001
THRESHPEN2  1.00000000000000E+0000

Input Files
INPUTDIR input
SPECNAME spec.dat
PUNAME pu.dat
PUVSPRNAME puvsp2.dat
BOUNDNAME bound.dat

Save Files
SCENNAME output
SAVERUN 2
SAVEBEST 2
SAVESUMMARY 2
SAVESCEN 2
SAVETARGMET 2
SAVESUMSOLN 2
SAVELOG 2
SAVESNAPSTEPS 0
SAVESNAPCHANGES 0
SAVESNAPFREQUENCY 23
OUTPUTDIR output

Program control
RUNMODE 1
MISSLEVEL  1
ITIMPTYPE 0
HEURTYPE -1
CLUMPTYPE 0
VERBOSITY 3
SAVESOLUTIONSMATRIX 3
```

❖ Compare your input.dat to this one. Are the parameters the same?

❖ Find the first highlighted parameter, the BLM. Set this parameter to 0 for the base Marxan database you are creating. BLM stands for Boundary Length Modifier, and the higher the BLM, the more "clumped" together your solutions will be.

❖ Set the NUMREPS to 100. This is the number of solutions Marxan will find each time you run it. NUMREPS stands for number of repetitions.

❖ Leave the NUMITNS as 1000000. NUMITNS stands for number of simulated annealing iterations in each solution.

❖ Make sure that the Input file and folder names match the names of your files.

❖ SCENNAME You will change this to help identify which scenario the results are from. For now, just leave it as "output" as we are not yet setting up a scenario.

❖ Make sure the OUTPUTDIR (output directory folder name) matches the name of your output folder.

❖ Save the any changes made to your input.dat in the "Marxan_database" folder.

To find out more about these and other parameters not described in this handbook, please consult the Marxan Users Manual.

Well done! You've successfully created your first Marxan database with ArcMap.
To continue go to Section C. Running Marxan for the first time.

B. Creating Marxan input files with QMarxan in QGIS

QGIS is an Open Source Geographic Information System (GIS) licensed under the GNU General Public License. It is an official project of the Open Source Geospatial Foundation (OSGeo). It runs on Linux, Unix, Mac OSX, Windows and Android and supports numerous vector, raster, and database formats and functionalities. For more information visit http://www.qgis.org/

Qmarxan is a set of free software tools for the creation of Marxan input files within QGIS. It was written by Apropos Information Systems. This software suite allows you to build and run Marxan datasets on the computing platform of your choice (Windows, Mac OSX, or Linux). Here we describe the use of the software on Windows with a case study. For more information visit http://aproposinfosystems.com/products/qmarxan/

❖ Navigate to "C:\IntroMarxan\QMarxan_data" and double click on **IntroMarxan.qgs** to launch the QGIS project.

❖ Explore each data layer so that you have a better understanding of the type of information that you will be working with in this course.
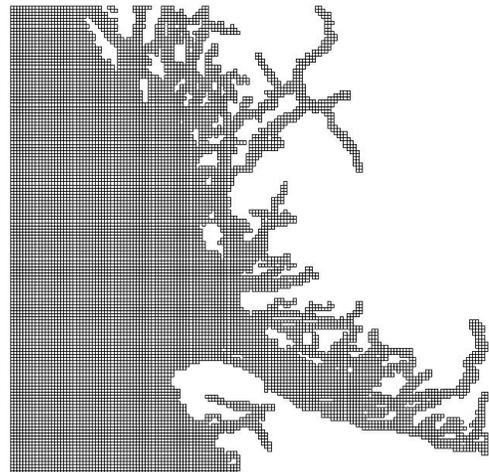
Step 1: Creating Planning Units
The size of the planning units is determined by the scale of the analysis, the resolution of the data used,

and the objective of the planning exercise (see Marxan Good Practices Handbook for more information). Planning units can be squares, hexagons, or irregular shape polygons. With help of GIS you will calculate the area within each planning unit covered by each conservation feature distribution, and use these quantities to run a Marxan analysis.

Next you will create a square grid for the British Columbia dataset with 2 x 2 km^2 cells (2000 x 2000 m^2). Note that the GIS data for the study region are in main units of meters.

Step 1: Creating Planning Units using QMarxan

This tool creates a planning unit grid with the following fields automatically created: pu_id, pu_cost, pu_status, bnd_cost, area, perimeter and side length.

❖ In QGIS, click the "Plugins" menu, select "Qmarxan", then click "Create Planning Grid".

❖ On the form that displays, select the "Planning Grid Extent" to be "Study_Region_BC_Ocean.shp", then click "Update extents from layer".

❖ Check the option "Limit creation of grid by intersection with layer" to extract only those planning units that only fall within the study area.

❖ Select "Side Length" to be "2000".

Note: a larger side length will generate fewer planning units, and a smaller side length will generate more planning units.

❖ Select Squares as planning grid shape,

❖ Click the Browse button, browse to "C:\IntroMarxan\QMarxan_data", and save the output shapefile as "pulayer_BC_marine.shp", then click "Save"

❖ Your form should then look like the figure to the right.

❖ Now click "Ok" to generate the planning units. When it is finished running, click "Close".
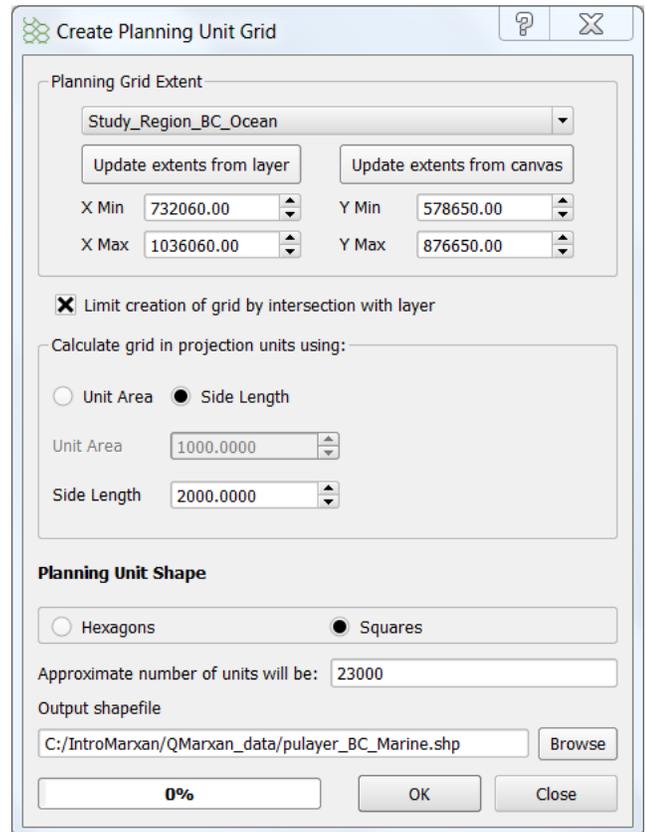
Step 2: Creating the planning unit file (pu.dat)

Now you will be creating a Marxan planning unit file that contains the planning unit id, cost (of that planning unit), and status (availability for selection).

Step 2a: Creating Cost Field

❖ In the "Plugins" menu, select "Qmarxan", then click "Calculate Planning Grid Conservation Factor Values".

❖ First click on the "New" botton to be able to fill out the form.

❖ On the form that displays, set "Planning Unit Layer" to "pulayer_BC_Marine", and "Planning Unit ID" as "pu_id".

❖ Set "Select input layer type" to "Area",

❖ Set "Select layer to measure" to "vessel_cost",

❖ Select "Measure x Field", and select the field "GRIDCODE" for the calculation

Note: this option multiplies the measured value (count, length, or area) by a field attribute in the layer you are measuring. This option would be best used for lengths or areas where there is a per unit value associated with the features you are measuring within the planning units (for more information visit http://aproposinfosystems.com/products/qmarxan/qmcalc/*),*

❖ Select "Sum" in "On multiple intersection use"

❖ Select "pu_cost" in the drop down menu bellow "Select Output Field Name".

❖ Your form should then look like the figure below:

❖ Now click "Run" to calculate the planning unit cost.

❖ When it is finished running, click "Close".

The planning unit cost has been saved to the planning unit attribute table as a field called "pu_cost".

Step 2b: Export data to create planning unit file

❖ In the "Plugins" menu, select "Qmarxan", then click "Export to Marxan",

❖ On the form that displays, set "Select Planning Grid" to "pulayer_BC_Marine",

❖ Click the "Browse" button to set the output directory to " C:\IntroMarxan\QMarxan_data"

❖ Click "pu.dat" tab to select the planning unit output file,

❖ Check "Create Planning Units File (pu.dat)" to switch it on,

❖ Set "Planning Unit Cost Field" to "pu_cost",

❖ Set "Planning Unit Initial Status Field" to "pu_status",

❖ Your form should then look like the figure to the right.

❖ Click "Ok" to create the planning unit file "pu.dat" in the output folder "C:\ IntroMarxan\QMarxan_data".

❖ Click "Close" to return to QGIS.
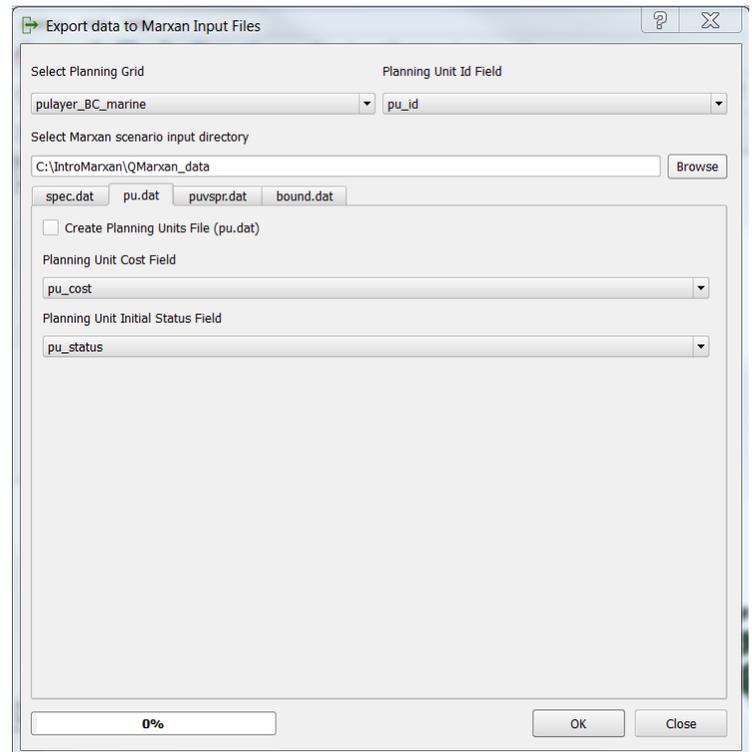
Step 3: Calculate area in each planning unit covered by each conservation feature

In this exercise, you will determine how much of each of the conservation features (benthic classes, sponge reef, and kelp) distribution is contained in each planning unit. This information will be used in Step 4 to produce the species file and the planning unit versus species matrix file.

For this exercise we provide you with:

1. One layer that detail the extent of 13 benthic classes in the study region. Each of the benthic classes is a conservation feature.

2. Layers detailing the distribution of sponge reefs and 3 different kinds of kelp: *Sponge_reefs_BC, Kelp_Giant_BC, Kelp_Bull_Gen_BC,* and *Kelp_General_BC*.

Before you start with the processing steps, it is a good idea to open up the layers and their attribute tables to familiarize yourself with the data. If you have questions about the data, ask your instructor.

Note: Geometrically complex features may take longer to process. To speed processing times, it is recommended to use the "Multipart to singlepart" tool in QGIS. This tool can be found in Vector→Geometry Tools

Once you have briefly explored the data, follow the next steps:

❖ In the "Plugins" menu, select "Qmarxan", then click on "Calculate Planning Grid Conservation Factor Values".

❖ First click on the "New" botton to be able to fill out the form.

❖ On the form that displays, set "Planning Unit Layer" to "pulayer_BC_Marine", and "Planning Unit ID" as "pu_id".

❖ Set "Select input layer type" to "Area",

❖ Set "Select layer to measure" to "Sponge_Reefs_BC",

❖ Select "Measure" in "On intersection use the following value"

❖ Set "Enter New Measure Field Name" to "sponge_ref",

Note: field names cannot be longer than 10 character name for a new field. This is the field width limit for shape files (see http://aproposinfosystems.com/products/qmarxan/qmcalc/).

❖ Click "Save"

❖ Your form should then look like the figure below:
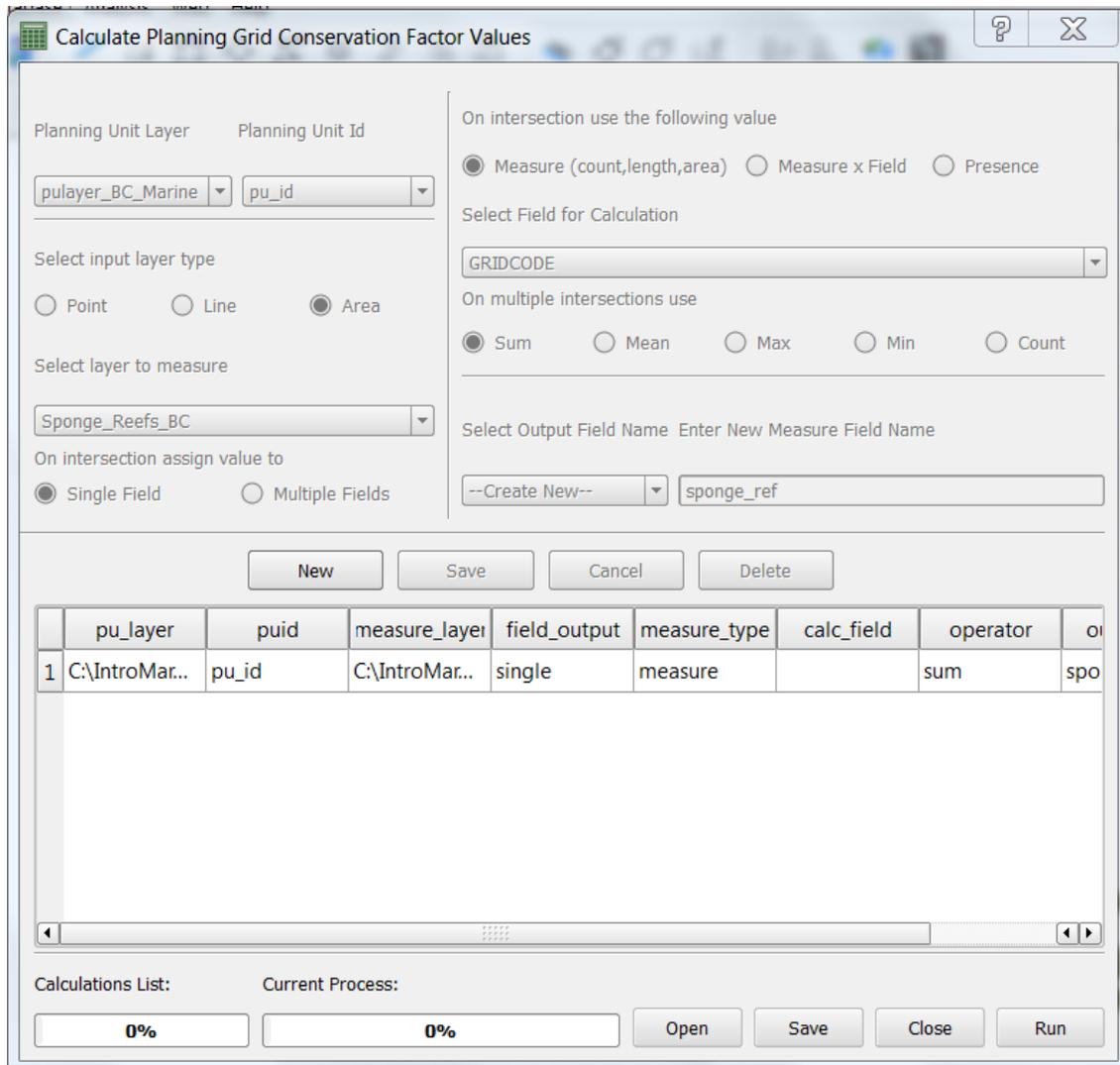
❖ Now click "Run" to calculate the amount of sponge reefs found in each planning unit.

❖ When it is finished running, click "Close". The planning unit attribute table now contains the amount of sponge reefs for each planning unit in the field "sponge_ref".

❖ Repeat the above steps for each of the remaining conservation features (total of 5). Each time you repeat the step, substitute the layer name "Sponge_Reefs_BC" with the layer name you are currently processing, for example with the layer name "Kelp_Giant_BC".

You can run each conservation feature one at a time, or in group by clicking on "New" after you have "Save" your setting. For example, the form with 4 conservation features (sponges and kelps) should look like the figure below:

Note: Doing this might slow down the processing time.
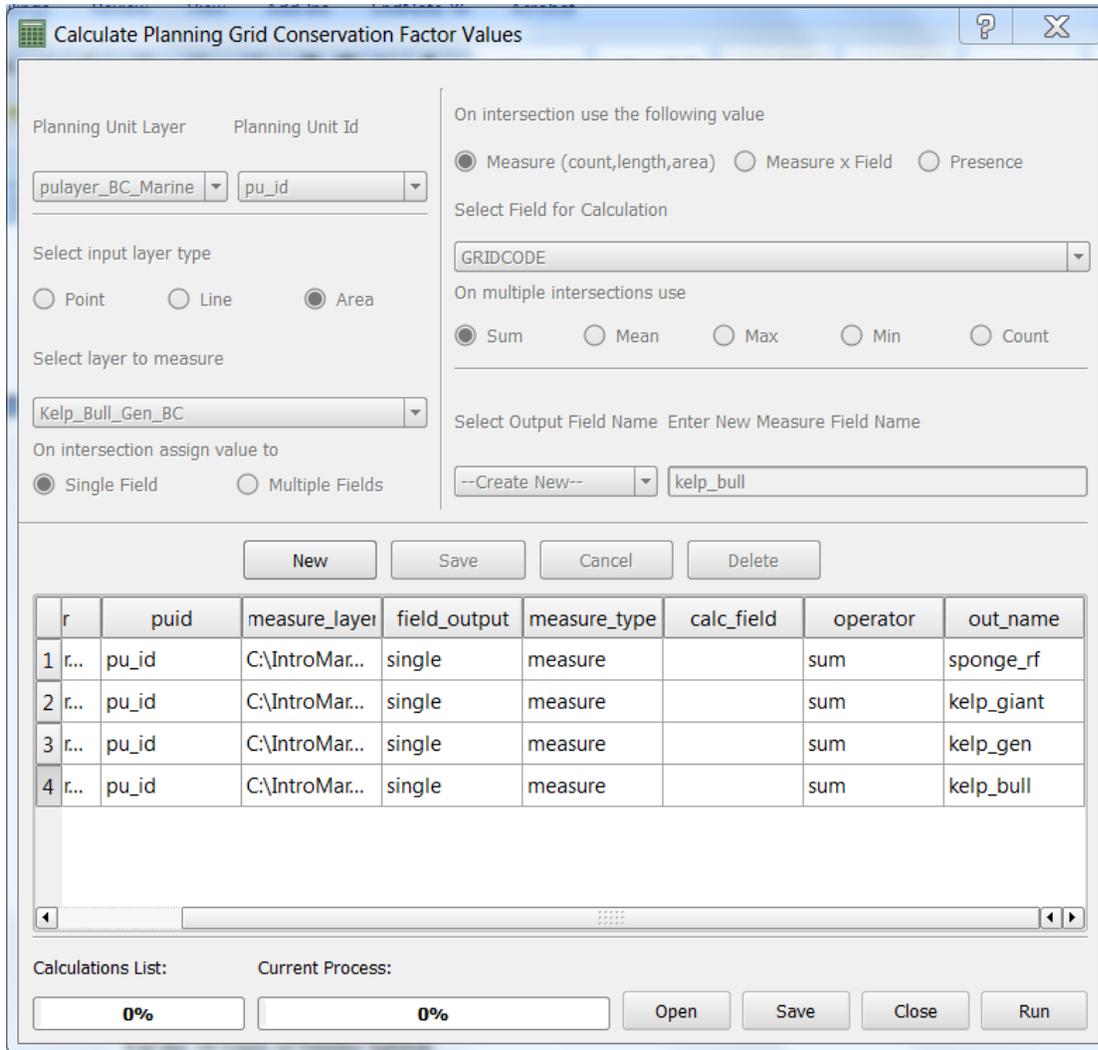
❖ For layers like the benthic habitat type form by more than one class within the same layer, it is possible to use the option "Multiple Fields" in "On Intersection assign value to", and then select the field with the unique identifier for each class. In addition, we also need to assign a prefix for the new field names that will be created and added to the planning unit layer. For example, we can add a prefix of "benth_" (it only accepts a maximum of 6 characters), which will result in a new field benth_1 for the habitat type 1. The form should look like the image below:
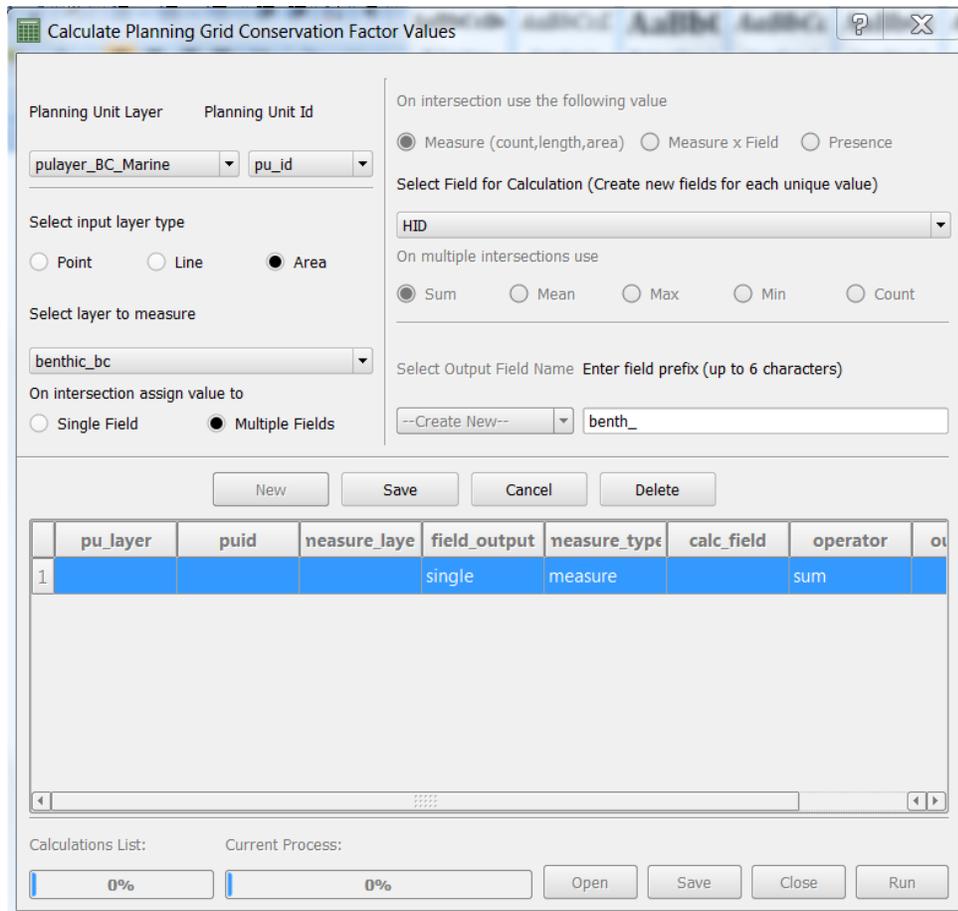
❖ Now when you open the attribute table of the layer "pulayer_BC_Marine" should contain a field for each of the 18 conservation features considered in each case study (open the attribute table to make sure you have 18 new fields). Each field contains the amount of each conservation feature in each planning unit. Take some time to see if the field values make sense and that you haven't missed adding any conservation feature.

❖ When you are finished, click "Close" to return to QGIS.

NOTE: Data validation is of utmost importance when constructing Marxan datasets. *Your reputation, your organisation's reputation, and the reputation of the Marxan software is at stake if the data used in a planning process turn out to have errors in it. This is an unfortunate situation you can avoid with simple validation steps.*

❖ Validate the calculation by overlaying for example the Sponge_Reefs_BS, Benthic_Muddy_Flat layers, and the planning unit layer to check if the results in the table correlate with the spatial data. Some GIS skills are necessary to perform this task.

❖ Choose several planning units where the two conservation feature layers overlap to perform these validation tasks. Start with selecting one planning unit:
  • Zoom into the planning unit you have selected

- Use the identify tool [icon] to obtain the PUID of the planning unit
- Explore the Identify Results (See figure below). Note how much area is covered by the sponge reef and the benthic muddy flat habitat for that particular planning unit.
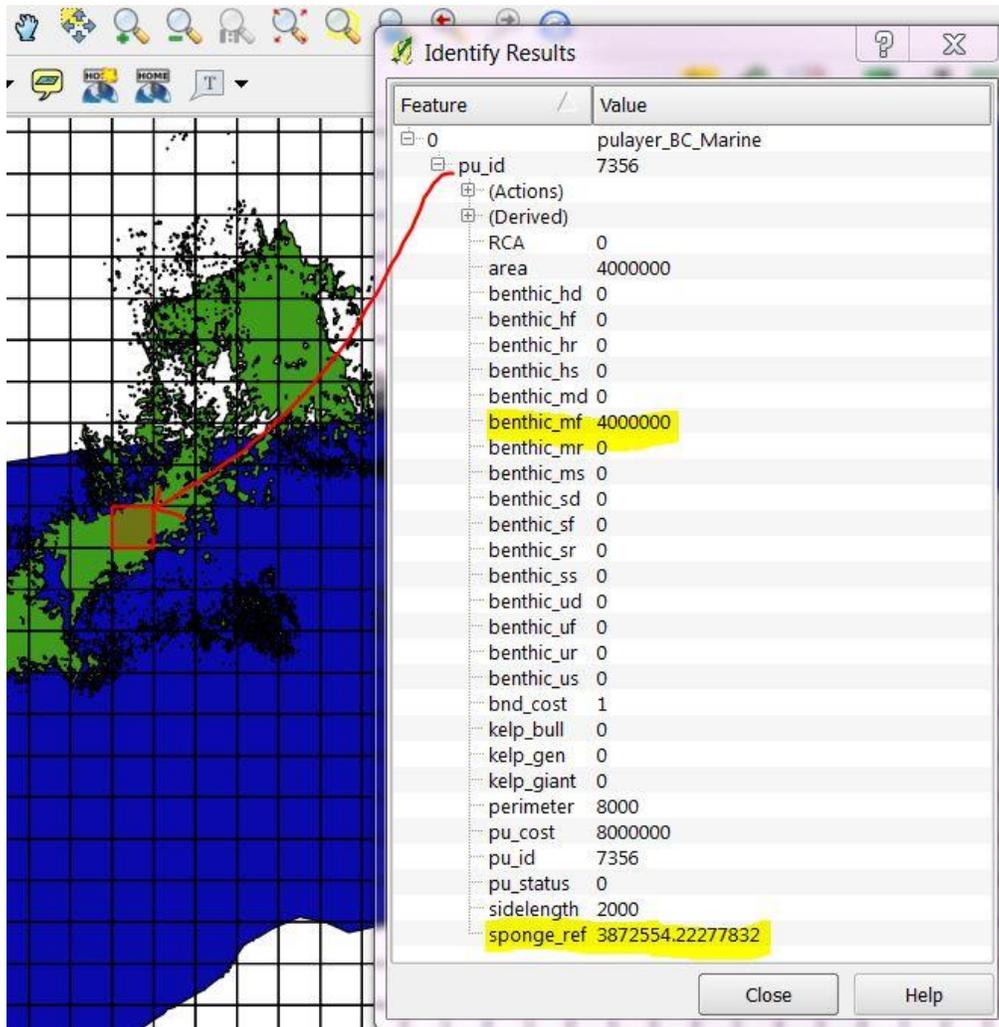
- In this example, the benthic muddy flat habitat covers the entire planning unit, with a total area value of 4000000 m^2 (which is also the total size of the planning unit). The sponge reefs almost completely cover the planning unit, with a total area of 3872554 m^2.
- When we visually inspect the spatial overlay of the conservation feature layers with the planning unit layer, we see that the estimated values by QMarxan correspond with what we observe.
- You can repeat this procedure using the one of the other conservation feature layers or a different planning unit. There are other ways to perform data validations, this is just one way.

Step 4: Creating the species file (spec.dat) and the planning unit versus species matrix (puvsp.dat)

In this step you will be creating the species file. This is called the "species file" because when Marxan was first created, all of the conservation features were individual species. Now we call them "conservation features", but the file that lists them and their targets is still called the species file or "spec.dat".

The "spec.dat" file lists the conservation features (each with a unique identifier or "id") and the target amount for that conservation feature (in this case, as a proportion of the total in the region or "prop") to be included in the reserve system. The "spf" column refers to the species penalty factor. In QMarxan, the "prop" and the "spf" values can be easily edited without the need to use a text editor. The only field that cannot be modified is "name" of each feature, which is extracted from the field name created in the previous step. This name can be edited outside QGIS with a text editor. In any case, it is very important to keep track of the names or codes assigned to each of the conservation features to help you keep track (many analyses will have hundreds of conservation features).

For the British Columbia dataset we will set a target of 30% protection of each conservation feature distribution using the "prop" field in the spec.dat file (i.e., 0.3 of the total area calculated for that conservation feature). We will leave the spf (species penalty factor) as the default value "1". You will have the opportunity to calibrate this value in the Day 2 exercises.

Step 4a. Creating the spec.dat file

❖ In the "Plugins" menu, select "Qmarxan", then click "Export to Marxan".

❖ On the form that displays, set "Select Planning Grid" to "pulayer_BC_Marine",

❖ Click the "Browse" button to set the output directory to "C:/IntroMarxan/QMarxan_data"

❖ Click "spec.dat" tab to select the planning unit output file,

❖ Click "Create Species File (spec.dat)" to switch it on,

❖ In the "field_no" field, click the check boxes to switch on each of the conservation features (total 20),

❖ Edit the "prop" field for each of the conservation features to "0.3" (30% protection of each conservation feature).

❖ Your form should then look like the figure to the right.

field_no		id	prop	target	spf	name
4	☐ 3	3	1.0	0.0	1.0	bnd_cost
5	☐ 4	4	1.0	0.0	1.0	area
6	☐ 5	5	1.0	0.0	1.0	perimeter
7	☐ 6	6	1.0	0.0	1.0	sidelength
8	✖ 7	7	0.3	0.0	1.0	sponge_ref
9	✖ 8	8	0.3	0.0	1.0	kelp_giant
10	✖ 9	9	0.3	0.0	1.0	kelp_gen
11	✖ 10	10	0.3	0.0	1.0	kelp_bull
12	✖ 11	11	0.3	0.0	1.0	benthic_hd
13	✖ 12	12	0.3	0.0	1.0	benthic_hf
14	✖ 13	13	0.3	0.0	1.0	benthic_hr
15	✖ 14	14	0.3	0.0	1.0	benthic_hs
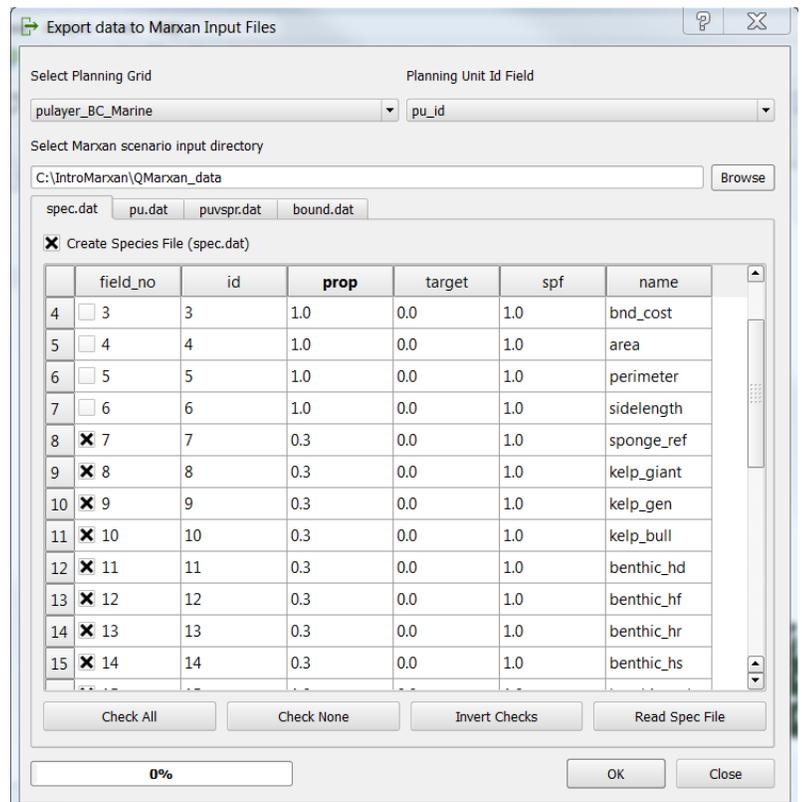
❖ Now click "Ok" to create the species file.

Step 4b. Creating the puvspr.dat file

❖ Uncheck "Create Species File" since it has been already created.

❖ Now Click "puvspr.dat" tab to select the planning unit output file,

❖ Click "Create Planning Units vs Species File (puvspr.dat and puvspr_sporder.dat)" to switch it on,

Note: the puvspr_sponder.dat is an inverse of the puvspr file (it is in species order) and its sole purpose is to speed up the burn in time when Marxan first runs. Its inclusion in my experience cuts the burn in time by about half.

❖ Set "Scale species data from 0 to 1" to "Raw".

❖ Your form should then look like the figure to the right.

❖ Now click "Ok" to create the planning unit versus species matrix (puvspr.dat).

❖ Click "Close" to return to QGIS.
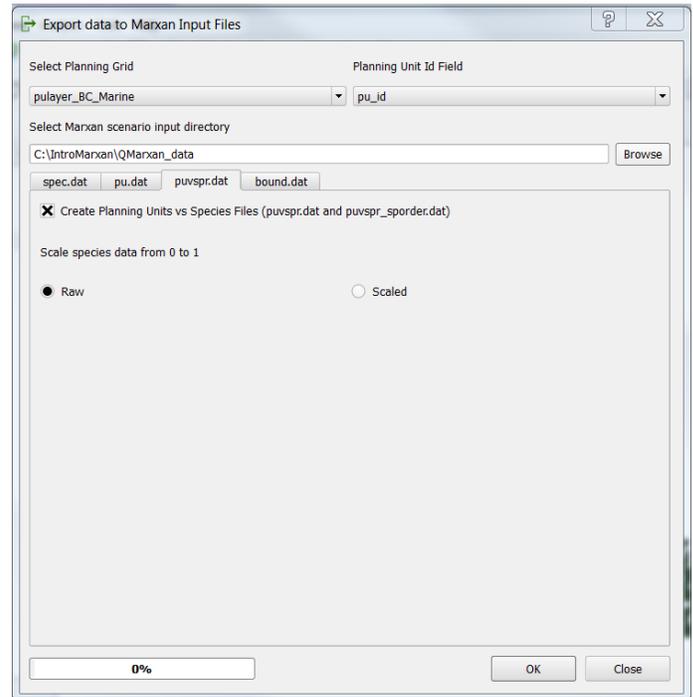
Step 5: Creating the boundary length file (bound.dat)

This file indicates the spatial relationship between the planning units and is essential for solutions that require spatial compactness. Id1 indicates the PUID for one planning unit, and id2 is the PUID for an adjacent planning unit. The boundary is the measurement of the boundary between the two planning units.

❖ In the "Plugins" menu, select "Qmarxan", then click "Export to Marxan".

❖ On the form that displays, set "Select Planning Grid" to "pulayer_BC_Marine",

❖ Click the "Browse" button to set the output directory to "C:/IntroMarxan/",

❖ Click "bound.dat" to select the boundary length output file,

❖ Click "Boundary File (bound.dat)" to switch it on.

❖ Your form should then look like the figure below:

id1	id2	boundary
1	2	2000
1	73	2000
1	1	4000
2	3	2000
2	74	2000
2	2	2000
3	4	2000
3	75	2000
3	3	2000
4	5	2000
4	76	2000
4	4	2000
5	6	2000
5	77	2000
5	5	2000
6	7	2000
6	78	2000
6	6	2000
7	8	2000
7	79	2000
7	7	2000
8	9	2000
8	80	2000
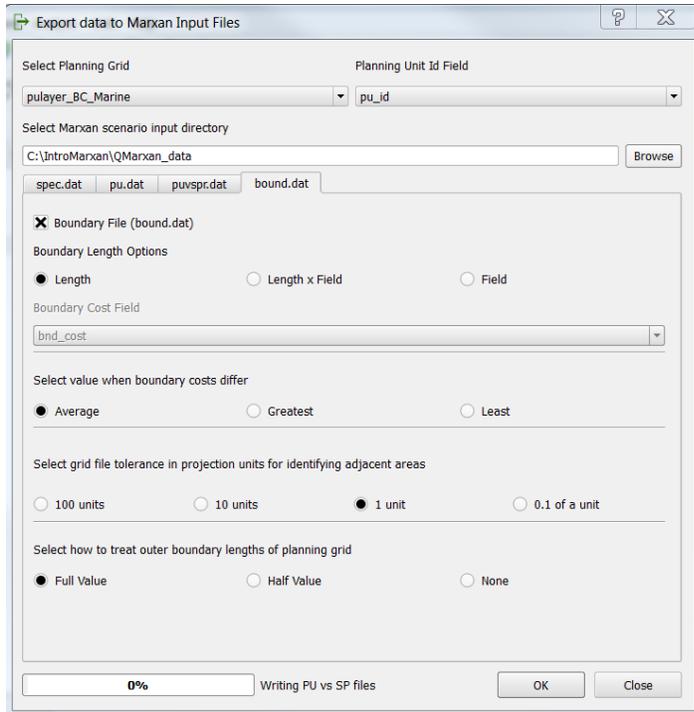
❖ Now click "Ok" to create the boundary file (bound.dat).

❖ Click "Close" to return to QGIS.

Step 6: Establish a standard file structure for the Marxan database

You are almost done creating the data files with QMarxan. The only file left to create is the input.dat file (see Step 7). But first you need to organize the already created files into the standard format.

Using the Windows explorer:

❖ Browse to the folder "C:\IntroMarxan\",

❖ Create a sub-folder here called "Marxan_database_backup",

❖ Within the folder just created create a series folders called "input", "output", and "pulayer".

❖ Copy the "Marxan.exe" file into the "Marxan_database_backup" folder if your computer is 32-bit, or "Marxan_x64.exe" if your computer is a 64-bit.

❖ The files in the "Marxan_database_backup" should appear as in the figure below:
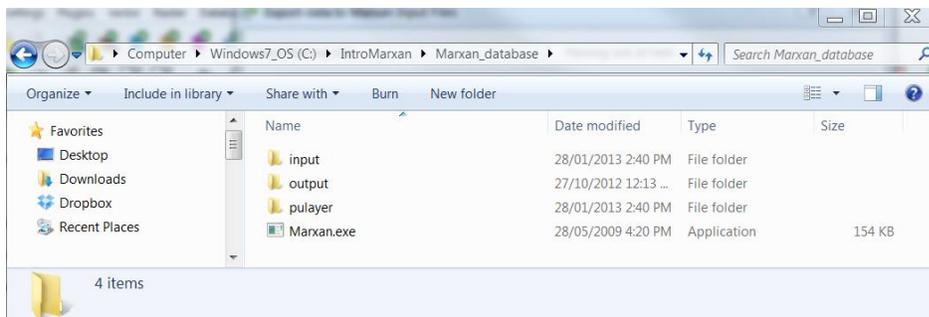
❖ Copy the following files into the "input" folder: "bound.dat", "pu.dat", "puvsp.dat", "puvspr_sporder.dat", and "spec.dat". Your "input" folder should look like the figure below:
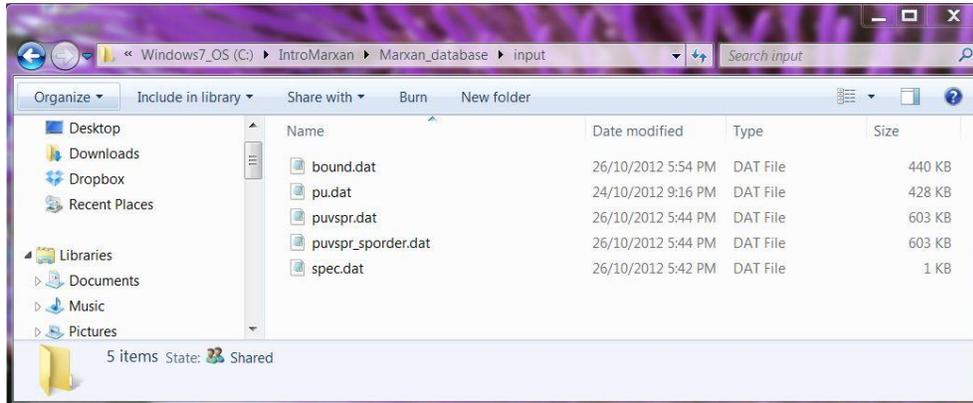
❖ Copy all 5 the files associated with the shapefile "pulayer_BC_Marine" into the "pulayer" folder (i.e.,"pulayer_BC_Marine.shp", "pulayer_BC_Marine.shx", "pulayer_BC_Marine.dbf", "pulayer_BC_Marine.prj", "pulayer_BC_Marine.qpj")

Step 7: Creating the input parameter file (input.dat)

The "input.dat" file is like a list of instructions for Marxan where you will indicate the name and location of your other input files and set several parameters. A detailed description of all the parameters that can be set in the input.dat is in the Marxan User manual, which is in the documents section of your course folders. Note that we will not cover all of these parameters in this course.

Note: Here we show you how to create this file with QMarxan. This file is also available when you download Marxan or Zonae Cogito.

❖ In the "Plugins" menu, select "Qmarxan", then click "Configure Scenario".

❖ On the form that displays, use the "Browse" button to set "Select scenario directory" to "C:\IntroMarxan\Marxan_database_backup",

❖ Now click on "Files and Costs" tab, and use the "Browse" buttons on that tab to specify the location of the different out files and Marxan files.

❖ Your form should then look like this:

❖ Now click "Ok" to create the input parameter file.

❖ Click "Close" to return to QGIS.
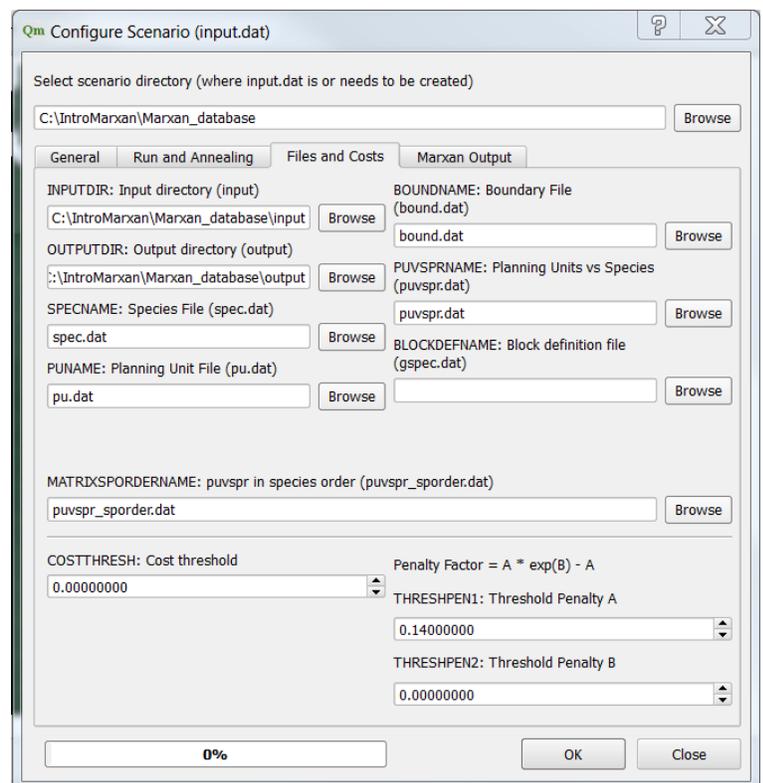
These and other parameters specified in this file will be set by you. You can edit the input file either in Notepad or using the "Configure Scenario" tool in QMarxan. These parameters are indicated in yellow in the example input.dat below.

❖ Open the input.dat file saved in "C:\IntroMarxan\Marxan_database\" with Notepad.

```
Input file for Annealing program.

This file generated by Qmarxan
created by Apropos Information Systems Inc.

General Parameters
VERSION 0.1
BLM 0
PROP 5.00000000000000E-0001
RANDSEED -1
BESTSCORE -1.00000000000000E+0000
NUMREPS 10

Annealing Parameters
NUMITNS 1000000
STARTTEMP -1.00000000000000E+0000
COOLFAC 0.00000000000000E+0000
NUMTEMP 10000

Cost Threshold
COSTTHRESH 0.00000000000000E+0000
THRESHPEN1 1.40000000000000E-0001
THRESHPEN2 0.00000000000000E+0000

Input File
INPUTDIR C:\IntroMarxan\Marxan_database\input
SPECNAME spec.dat
PUNAME pu.dat
PUVSPRNAME puvspr.dat
BOUNDNAME bound.dat
MATRIXSPORDERNAME puvspr_sporder.dat

Save Files
SCENNAME output
SAVERUN 2
SAVEBEST 2
SAVESUMMARY 2
SAVESCEN 2
SAVETARGMET 2
SAVESUMSOLN 2
SAVELOG 1
SAVESNAPSTEPS 0
SAVESNAPCHANGES 0
SAVESNAPFREQUENCY 0
OUTPUTDIR C:\IntroMarxan\Marxan_database\output

Program control.
RUNMODE 1
MISSLEVEL 9.50000000000000E-0001
ITIMPTYPE 1
HEURTYPE 0
CLUMPTYPE 0
VERBOSITY 3
```

❖ Compare your input.dat to this one. Are the parameters the same?
❖ Find the first highlighted parameter, the BLM. Set this parameter to 0 for the base Marxan database you are creating. BLM stands for Boundary Length Modifier, and the higher the BLM, the more "clumped" together your solutions will be.

❖ Set the NUMREPS to 10. This is the number of solutions Marxan will find each time you run it. NUMREPS stands for number of repetitions.

Note: you can also change the BLM values and NUMREPS in QMarxan, in the "General" tab in the "Configure Scenario" tool.

❖ Leave the NUMITNS to 1000000. NUMITNS stands for number of simulated annealing iterations in each solution.

Note: you can also change the NUMITNS in QMarxan in the "Run and Annealing" tab in the "Configure Scenario" tool.

❖ Make sure that the Input file and folder names match the names of your files.

Tip: because you can use the same input file to run different scenarios, it is better to set the input directory INPUTDIR only as "input". Otherwise, you will have to make sure that the input directory always matches exactly with path where your Marxan input files are stored.

❖ SCENNAME You will change this to help identify which scenario the results are from. For now, just leave it as "output" as we are not yet setting up a scenario.

Note: you can also change the SCENNAME in QMarxan in the "Marxan Output" tab in the "Configure Scenario" tool.

❖ Make sure the OUTPUTDIR (output directory folder name) matches the name and path of your output folder.

Tip: because you can use the same input file to run different scenarios, it is better to set the output directory OUTPUTDIR only as "output". Otherwise, you will have to make sure that the output directory path name always matches exactly with path where your Marxan output file is.

❖ Finally, under 'Program Control', in the last line of the file, add "SAVESOLUTIONSMATRIX 3". This file will be used during the second day to run the cluster analysis in R. (Currently QMarxan does not include this function automatically)

❖ Save the any changes made to your input.dat in the Marxan_database folder.

To find out more about these and other parameters not described in this handbook, please consult the Marxan Users Manual.

Well done! You've successfully created your first Marxan database with QMarxan. Not it is time to run Marxan for the first time.

C. Running Marxan for the First Time

After creating your Marxan database (either with the help of ArcMap or QMarxan), you are ready to run Marxan.

❖ First make a copy of this folder and call it Marxan_database_backup.

❖ Then, go to the Marxan_database folder, double click on "Marxan.exe" your computer is 32-bit, or "Marxan_x64.exe" if your computer is a 64-bit

If you have created all the input files properly, you should see Marxan generating solutions (see screen below). If you see something else ask your instructor for assistance.

D. Displaying Marxan spatial outputs

Next we show you how to display the spatial outputs created by Marxan in ArcMap (section D.1) and in QGIS (section D.2) in order to create maps suitable for presentations, etc..

D.1. In ArcMap

Step 1: Convert best solution and sum solution files text files to Excel files.

❖ Open "output_best.txt" from your local Marxan's output folder (C:\IntroMarxan\Marxan_database\output\) with Excel

❖ Select column A, and use Text to Columns wizard (in Data) to separate the fields into two columns. Use Comma as your delimiter.

❖ Save file as an Excel Workbook (.xls or .xlsx) with a file name "best_sol".

❖ Repeat the above steps to convert "output_ssoln.txt" to excel.

❖ Save file as an Excel Workbook (.xls or .xlsx) with a file name "sum_sol".

Step 2: Join output Marxan files to planning unit file

- ❖ Open a new ArcMap project, or start ArcMap if you closed the program.

- ❖ Add the planning unit file "pulayer_BC_Marine" saved in your Marxan_database (C:\IntroMarxan\Marxan_database\pulayer\).

- ❖ Right click on "pulayer_BC_Marine" and select "Joins and Relates" and then "Join". In the first drop menu select the field "PUID". In the second menu, browse for the "best_sol.xls" file. In the third menu, select "planning_unit"

- ❖ Ensure your Join window has the following parameters shown in the figure below:

- ❖ Click OK

- ❖ Repeat the above steps to join the "sum_sol.xls" data to the planning unit file.

Step 3: Visualize results in ArcMap

- ❖ Go to the layer properties to visualize the best solution using the Categories visualization method, or the sum solution (or selection frequency) using the Quantities visualization method.

- ❖ In the best solution, a value of "1" means that the planning unit was selected for the reserve, and a value of "2" that it was not selected.

- ❖ Your results should look like the figures below:
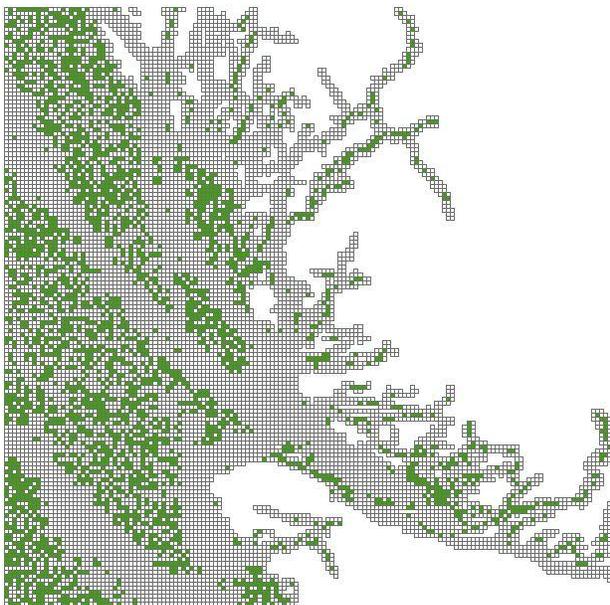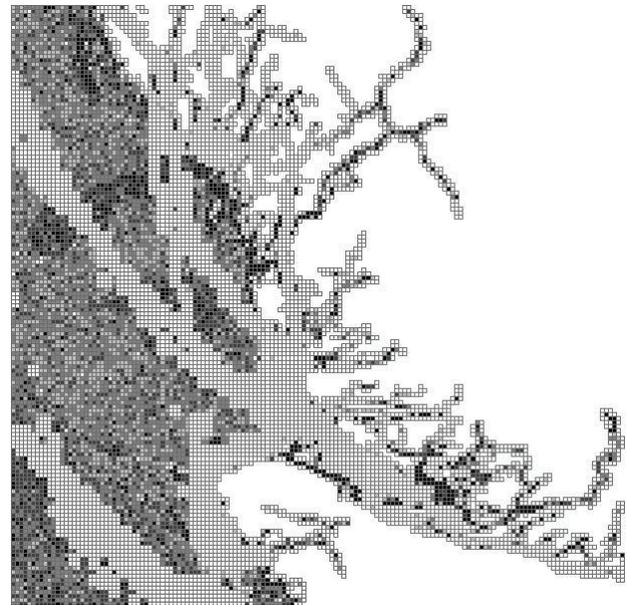
Best Solution   **Sum Solution**

This is the end of Day 1 Handbook. For extra exercises see <u>Appendix</u>.

D.2. In Quantum GIS

These steps show you how to import your results from Marxan and display them in Quantum GIS:

Step 1: Import Best Solution

❖ Open a new QGIS project, or start QGIS if you closed the program.

❖ Add the planning unit file "pulayer_BC_Marine" saved in your Marxan_database (C:\IntroMarxan\Marxan_database\pulayer\).

❖ In the "Plugins" menu, select "Qmarxan", then click " Import Marxan Results".

❖ Select your planning unit file "pulayer_BC_Marine" on the "Planning Grid" control,

❖ Select "pu_id" on the "Planning Unit Id Field" control,

❖ Browse "output_best" from your local Marxan's output folder (C:\IntroMarxan\Marxan_database\output\) to import the Best Solution,

❖ Leave the default "--Create New--" on the "Select Results Field Name" control. This will add a new field to your planning unit layer that will contain the "Best Solution" calculated by Marxan.

❖ Type "best_sol" on the "Enter New Results Field Name" control to give your new field a name.

❖ Your form should then look like the figure to the right.

❖ Now click "OK" to write the Marxan result file to the planning unit dbf table.
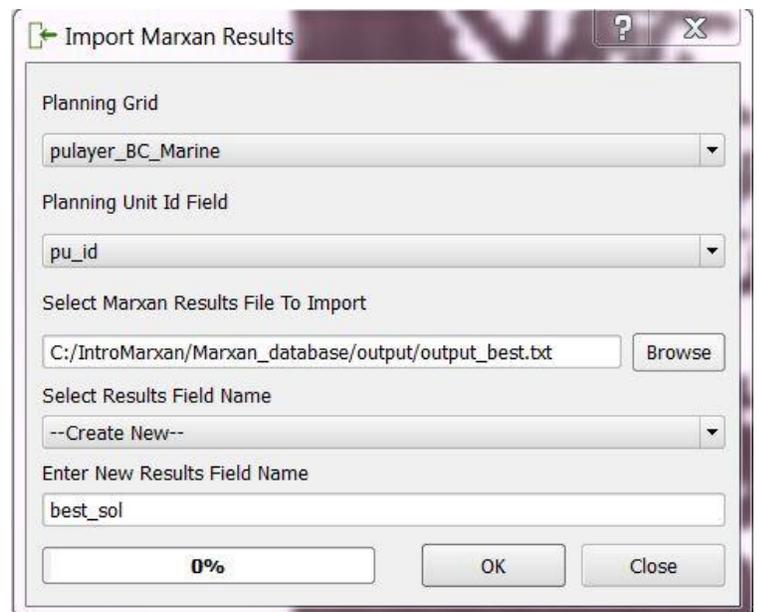
Step 2: Import Summed Solution

❖ If you closed the "Import Marxan Results" window, open it again.

❖ Select your planning unit file "pulayer_BC_Marine" on the "Planning Grid" control,

❖ Select "pu_id" on the "Planning Unit Id Field" control,

❖ Browse "output_ssoln" from your local Marxan's output folder
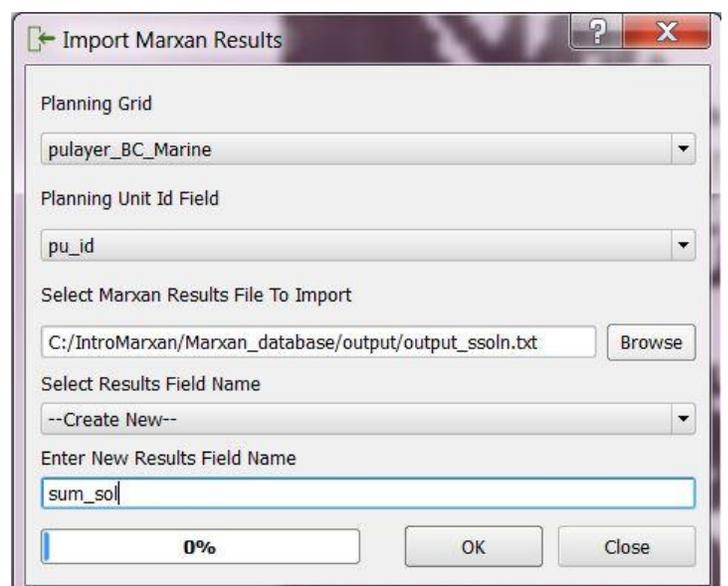
(C:\IntroMarxan\Marxan_database\output\) to import the Best Solution,

❖ Leave the default "--Create New--" on the "Select Results Field Name" control. This will add a new field to your planning unit layer that will contain the "Sum Solutions" calculated by Marxan, and also known as Selection Frequency.

❖ Type "sum_sol" on the "Enter New Results Field Name" control to give your new field a name.

❖ Your form should then look like the figure to the right:

Note: You can repeat this process to add any solutions produced by Marxan. In this case, any of the other 9 solutions produced by Marxan

Step 3: Display Best Solution or Summed Solution

❖ Double click on the planning unit layer in the list on the left panel of Quantum GIS,

❖ Go to the Style Tab

❖ To display the Best Solution, select "Categorized"

❖ Select "best_sol" in the Column menu, and click on Classify. Value "1" represent selected planning units in the best solution, and value "0" selected planning units.

❖ If you are happy with the colour, click OK. You can change the colors of the symbol by double clicking on the colored symbol icon.   Here, selected planning units are displayed in green and not selected planning units in white. Your form should then look like the figure to the right.

❖ Now click "OK" to display the Marxan result in Quantum GIS.

❖ If you want to display the "Summed Solution", select the "Graduated" option in the style menu. You can change the mode of classification and number of classes, as well as the range of values. Here the values are classified based on Natural Breaks, using 7 classes, and
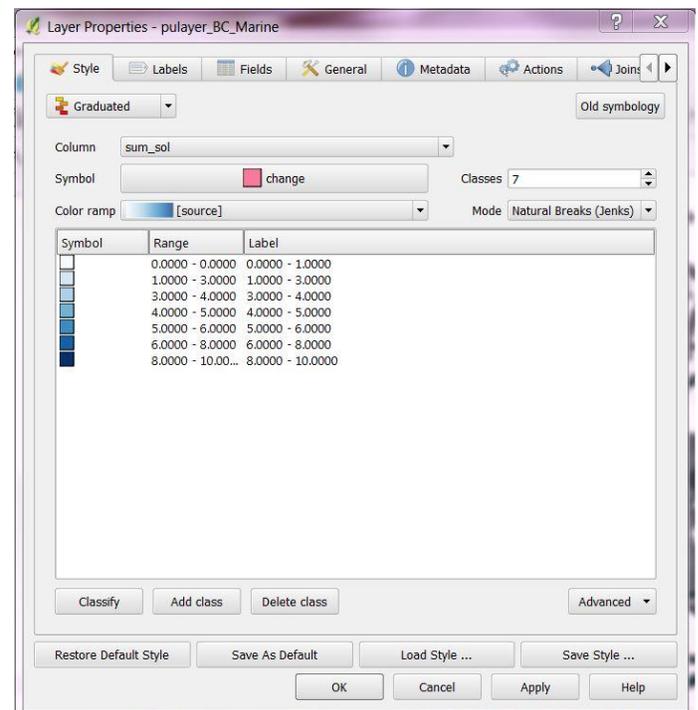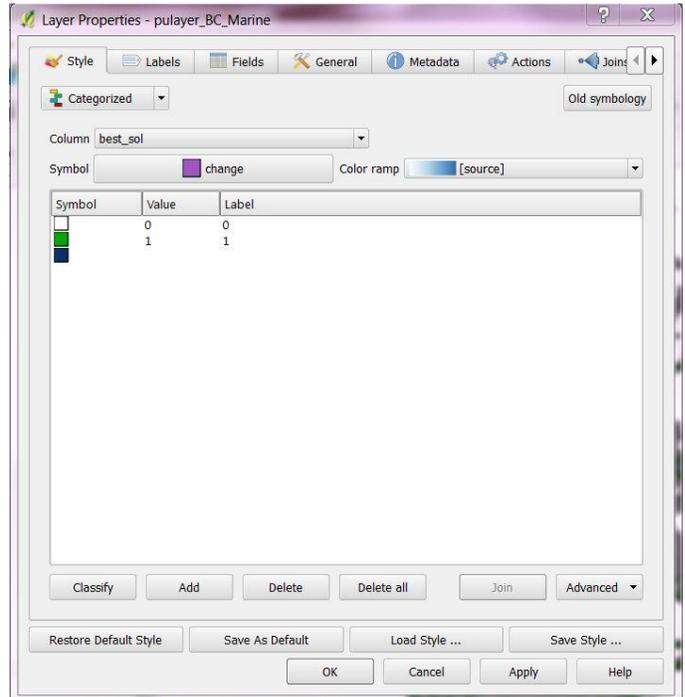
setting the first class to only display 0 values. Your form would look like the figure to the right:

❖ Now click "OK" to display the Marxan result in Quantum GIS.

❖ Your results should be similar to the figures below:
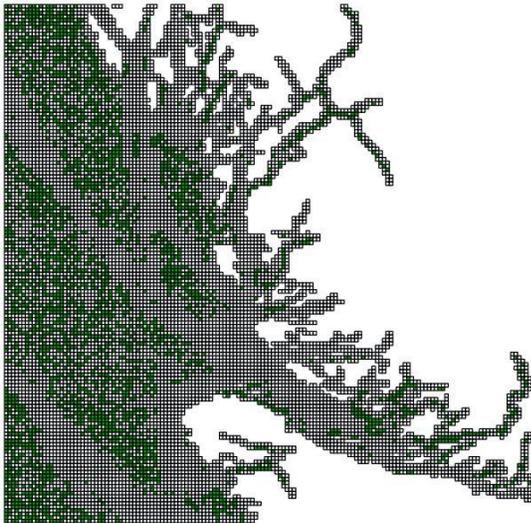
Best Solution
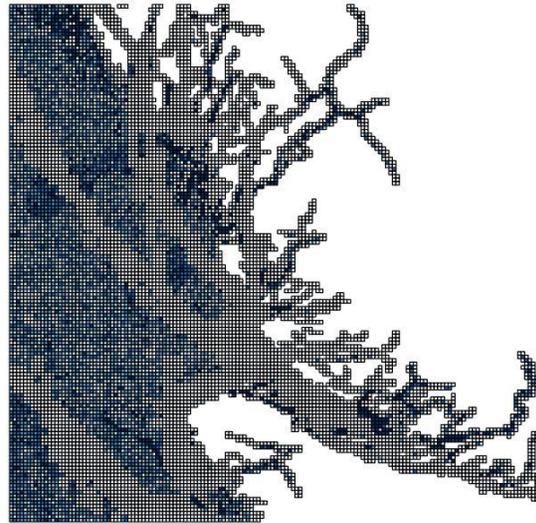
Sum Solution

This is the end of Day 1 Handbook. For extra exercises see Appendix.

Appendix: Extra Exercises

In the first extra exercise, you will learn an alternative method to create a planning unit layer using a free extension for ArcMap, called Jenness Repeating Shape (for more information about Jenness Enterprises, go to www.jennessent.com/arcgis/arcgis_extensions.htm).

In the second extra exercise, you will use a tool called Zonae Cogito to create some of your input of files. We will discuss this tool more in Day 2 of the course.

Creating Planning Units using Jenness Repeating Shapes

Note: This tool uses the cell centroids instead of the cell corners to match the extent of the grid to the extent of the study area. To match the grid perfectly with the extent of the study area, first you will have to snap the grid to the lower corner of the study area layer (i.e. Study_Region_BC_Ocean), and second delete the excess row and column on the upper right corner.

❖ In ArcMap, click on the repeating shapes tool

❖ In the box labelled "Select the region to cover:" select "Within the Extent of a Particular Layer" and click "Next…"

❖ Select 'Study_Region_BC_Ocean' and click "Next…"

❖ Select the "Squares" option and click "Next…"

❖ Enter the following parameters as shown here:

   • Type "2000" in the "Edge" box

   • Specify C:\IntroMarxan\pulayer.shp as the "Output Feature Class or Shapefile:"

❖ Click "OK"

❖ Click "Exit" after viewing the report
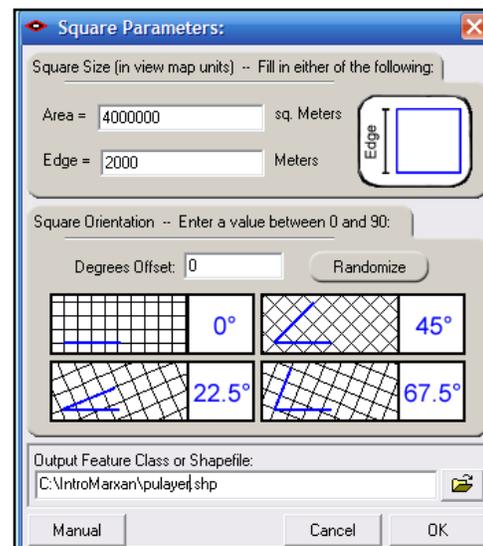
An alternative way to create some of the input files

You can use a tool in Zonae Cogito to help create some of the input files. We had you create these files by hand so you could be familiar with how they are put together and what they contain. Automated files include pu.dat, spec.dat, puvspr.dat, and sporder.dat (a file used to speed up processing). The pu.dat and spec.dat need to be updated after they are created automatically. This tool can be a real time saver for the creation of the puvspr.dat file. Before using this tool, you would need to (1) have created your planning units and given them a unique id, and (2) performed a Tabulate Area to create a .dbf for each of your conservation features (as you did above when creating your puvspr.dat file). At least one of your conservation feature groups needs to have full coverage of the study region. As a result, we have added a new

"conservation feature" that covers the entire region. To practice using the DBF tool, you can use the Tab Area .dbf files that you created earlier and the new entire region .dbf that we provide.

The first step is to place all of your tab area results (the .dbf files you used to calculate how much of each conservation feature was contained in each planning unit) into one folder. To do this:

❖ Create a folder in C:\IntroMarxan called "TabArea"

❖ Copy all of the Tabulate area .dbf files you created (e.g. Tabulate_benthic.dbf) to the TabArea folder

❖ Copy the file Tabulate_area_region.dbf from the C:\IntroMarxan folder into the TabArea folder

Zonae Cogito was created as a user interface/ decision support tool for use with Marxan. The DBF table join tool joins your tabulated conservation feature data (in DBF form) together into the correct format for Marxan. In addition, it can create a "base" spec.dat, pu.dat, and sporder.dat file.

❖ Open Zonae Cogito (ZC) from the program menu

❖ If Zonae Cogito does not appear in the program menu, browse to "C:\Program Files\Zonae Cogito", and double click ZonaeCogito.exe to launch the program

❖ In the ZC Main Menu, choose Convert Data>Tables>Join DBF tables puvspr

The window shown below will appear:
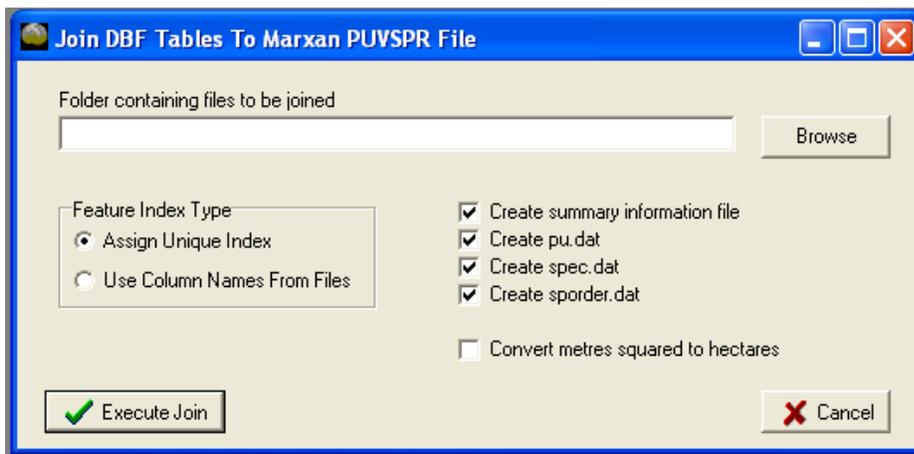
❖ Browse to the C:\IntroMarxan\TabArea folder where your .dbf files are saved

❖ Ensure that the rest of the settings are as below – the Feature Index Type will assign a unique identifier to each of your conservation features, and by ticking the "create pu.dat", "create spec.dat" and "create sporder.dat" boxes, ZC will create a **base file** for pu.dat and spec.dat for you to edit and a sporder.dat for you to include in your database (this will not need to be edited)

❖ Click "Execute Join"

The process might take a minute or two to complete.

- ❖ When it is finished running (you will see a screen pop up, run something, and close), open the TabArea folder

- ❖ Open the folder called "Marxan" – ZC has created an entire Marxan database

- ❖ Copy the entire Marxan folder and place it in C:\IntroMarxan, naming it "Marxan_database_auto"

- ❖ Open the puvspr.dat file (in the input folder) with a text editor – note that the file is now in the necessary format (species, pu, amount sorted by planning unit)

- ❖ Open the pu.dat file with a text editor – the ZC tool has produced a "base" pu.dat file, and to use this you would simply need to update the cost and status data (using spreadsheet software like Excel) like you did in the exercises above

- ❖ Open the spec.dat file – this is also a "base" file, and you can add in a name field (using a spreadsheet software like Excel) to more easily identify which id number refers to which conservation feature

The ZC tool assigned a unique index to each conservation feature.

- ❖ Open the file called "feature_index_assignment.csv" (respond "Yes", then "OK" to the pop-ups)

- ❖ Review the file, noting how the conservation feature ids have been assigned

You can use this tool to create all of the basic input files EXCEPT the bound.dat.

If you still have extra time (on Day 1), try creating another Marxan database using terrestrial data from Tasmania (located at C:IntroMarxan\Extra_Activities\Tasmania). 5000m x 5000m planning units are sensible for this problem, but feel free to experiment with different planning units. If you require step by step instruction for this extra activity, you can use the document Case Study Profile Tasmania.doc in the C:\IntroMarxan\Extra_Activities\Tasmania folder.