Hydrodynamic Modelling Program for the Proposed Gunns Pulp Mill

Report No. 2: Coupling Near and Far-Field Models

Prepared for

Gunns Limited

RPS MetOcean
26/05/2010
Job No. J2756
Report No. 1463v0
Copy No.

26 May 2010

Mr Lawson Harding
Gunns Limited
PO Box 572
LAUNCESTON TAS 7250

Dear Lawson,

Hydrodynamic Modelling Program
for the Proposed Gunns Pulp Mill
Report No. 2: Coupling Near and Far-Field Models

Enclosed please find our second report R1463 Version 0, entitled "Hydrodynamic Modelling Program for the Proposed Gunns Pulp Mill - Report No. 2: Coupling Near and Far-Field Models".

If you have any queries regarding this report please do not hesitate to contact me.

Yours faithfully
RPS MetOcean

Dr Chris Fandry
Environmental Manager

LIMITATIONS OF REPORT

This report was prepared for the use of *GUNNS LIMITED* in accordance with generally accepted consulting practice. No other warranty, expressed or implied, is made as to the accuracy of the data and professional advice included in this report. This report has not been prepared for use by parties other than the client, the owner and their respective consulting advisors. It may not contain sufficient information for purposes of other parties or for other users.

RPS MetOcean takes no responsibility for completeness or form of any subsequent copies of this report. Copying of this report without the permission of the client or RPS MetOcean is not permitted.

TABLE OF REVISIONS

Version	Description of Issue	Date of Issue	Prepared by	Reviewed by	Approved by
A	Draft	20/5/2010	C B Fandry	K Shimizu	S Buchan
B	Final/Draft	26/5/2010	C B Fandry	Client	S Buchan
0	Final	26/5/2010	C B Fandry		S Buchan

TABLE OF CONTENTS

Page No.

LIST OF TABLES

LIST OF FIGURES

1 BACKGROUND

Gunns Limited proposes to build a Bleached Kraft Pulp Mill at Bell Bay (near George Town, Tasmania) on the Tamar River. The proposed mill will discharge some 64 megalitres/day (64 million litres/day) of treated effluent 2.7 km offshore from Five Mile Bluff to the east of Low Head on Tasmania's north coast (Fig. 1). In support of the environmental approvals for the proposed mill, particularly in relation to the fate of the pulp mill effluent discharged into Bass Strait, the Commonwealth and the Tasmanian State have requested very detailed hydrodynamic modelling as described in the Scope of Works (SOW) (RPS MetOcean, 2008). The main purpose of the modelling is to examine the potential for mill effluent to compromise the water and sediment quality guidelines developed specifically for the pulp mill by the Commonwealth and the Tasmanian State.

The effluent will be pumped through a pipeline from the mill to an ocean outfall with a 250 m[1] long diffuser pipe at its end, in a mean water depth of 26.9 m (the tidal range here is 2.5 m). The hydrodynamic modelling program is required to address the near-field mixing zone and the subsequent far-field fate and transport of the effluent plume, including sediment transport.

The near-field modelling has been addressed separately by WorleyParsons (WP) in their report, "Bell Bay Pulp Mill, Outfall Near-Field Modelling, May 2010". RPS Metocean is undertaking the far-field modelling.

To achieve an integrated hydrodynamic model of the effluent discharge and the receiving environment, the near-field and far-field models must be coupled together and this report describes the methodology that will be used to achieve this.

1.1 Near-Field Region

The WP report defines the near field region "*as the area where the trajectory and mixing of the effluent jet generated by the diffuser is predominantly governed by the jet's initial momentum and buoyancy flux, the diffuser geometry and, the ambient velocity and stratification. This region also includes any initial interaction of the jet with any terminal layer, such as those produced by an internal stratified layer or either the bottom or surface layers of the receiving water body. In the near-field region mixing is generated by turbulence formed through the interaction of the effluent with the receiving water body. The formation of these turbulent eddies can be separated into two significant stages. The first of these stages is located in the region very close to the diffuser where the velocity of the discharge is relatively high. In this region, momentum dominates and mixing is related to the inertia of the turbulent eddies formed. As the jet moves away from the diffuser, it loses momentum and the buoyant forces due to a difference in the density of the effluent and the surrounding fluid begins to dominate. In this stage, the buoyant force produces the turbulence leading to mixing.*"

[1] Previous studies have used a 200m diffuser.

The traditional approach to modelling the near-field characteristics of an effluent plume being discharged from a diffuser is to use an empirical model such as JETLAG, CORMIX or VISUAL PLUMES. These models are applicable to receiving waters ranging from rivers to estuaries to oceans and require the specification of ambient conditions, such as the currents and background density stratification.

WP used JETLAG to obtain an initial characterization of the effluent plume in terms of the plume trajectory and its development as ambient fluid is entrained but supplemented this traditional approach with much more sophisticated Computational Fluid Dynamics (CFD) modelling. CFD is a very computationally intensive method based on the numerical solution of the governing equations of fluid motion (Navier-Stokes Equations). Traditional models such as JETLAG are no longer applicable once the plume reaches the surface and WP's use of CFD allowed the plume to be modelled over all of its stages in the near-field.

Details of the CFD approach can be found in the WP report. The results are summarized in their Table 7-6, which is reproduced here as Table 1. The table shows what the expected near-field dilutions and plume depth (Height) are for a range of diffuser lengths and ambient currents across the diffuser. As expected, dilutions increase with increasing current flow and with increasing diffuser length (except for some minor exceptions).

The most appropriate diffuser length is an optimal balance between dilution benefits, construction cost, constructability and ongoing maintenance costs and logistics. Setting the diffuser length is fundamental to the coupling of the near-field and far-field models. Once the length is set, the models can be coupled and the full integrated hydrodynamic model runs can be initiated.

A series of technical workshops were held between Gunns, RPS MetOcean and WP. As a result of these discussions both WP and RPS MetOcean undertook a series of model runs using different diffuser lengths, as described in the Recommendation section of their report. Through this process, Gunns determined to adopt a diffuser length of 250 m for the full hydrodynamic modeling task.

At the State level, adopting a greater length diffuser than was originally proposed (200 m) in the DIIS could have implications for the definition of the pulp mill project's effluent discharge mixing zone. This is because the State permit defines the zone by reference to the end points of the diffuser. Under this definition, a longer diffuser would prescribe a larger mixing zone. Gunns is not seeking a larger mixing zone and accordingly has sought and received confirmation from the State that the mixing zone definition could be amended to remove the link between diffuser length and the mixing zone boundaries

1.2 Far-Field Modelling

The far-field modelling describes the fate of the effluent plume once it loses diffuser induced momentum and entrainment and becomes subject to the ambient oceanic conditions. This occurs at distances usually less than ~100m from the diffuser. Subsequent transport, spreading and dilution of the effluent plume are then determined by physical processes of the receiving waters such as horizontal and vertical ocean currents, vertical mixing and horizontal diffusion. These processes are modified in the presence of density variations caused by salinity and/or temperature variations, particularly vertical density stratification which inhibits vertical mixing which in turn may lead to high concentrations of effluent near the surface.

The earlier RPS MetOcean report (RPS MetOcean Functional Nested Pilot Model Sept. 2009) describes in detail the far-field modelling approach.

1.3 Coupling Methodology

The WP report describes two alternative methodologies for coupling the near and far field models. These are referred to as “one-way coupling” and “two-way coupling.”

In one-way coupling, the near and far-field models are run independently, with the one-way model run first to obtain the quasi steady state spatial distribution of the effluent plume in the near-field. The far-field model is then run using this distribution to define an effluent loading pattern into the far-field.

In two-way coupling, the near and far field models are run simultaneously with a feedback between the two as time progresses. Thus the ambient currents and density structure produced by the far-field model are fed into the near-field model at each near-field model time step and in turn the spatial distribution of the effluent plume is fed into the far-field model at each far-field model time step. This is clearly a much more complex and computationally expensive method, and is rarely used because of the difficulty in its implementation. Furthermore, advice provided to the IEG, DEWHA and Gunns from Dr Ian Wallis (adviser to the IEG) suggests that such complex coupling methods are not warranted and a simple rule of thumb where the effluent is distributed over the top 30% of the water column is a good approximation with an acceptable degree of conservatism.

Following this advice and based on the recommendation of the WP report, a one-way coupling approach will be used in all further modelling.

To implement the one-way coupling, the near-field characteristics of the effluent plume summarised in Table 1 are matched as closely as possible to the actual diffuser length, the ambient cross-flow currents and the horizontal grid spacing of the far-field model (which in the Very Fine Grid is 50 m). WP has done this in their Table 8.1 which is reproduced here as Table 2. This has been derived from Table 1 by using the values in the 0.018 m s^{-1} column with the Height scaled by the ratio of the Length to the model grid length of 50 m.

The adopted 0.018 m s^{-1} cross-flow velocity is the most conservative (but non-zero) low flow condition in Table 1. The exceedance table of measured currents (Table 3) shows that velocities of 0.02 m s^{-1} and 0.01 m s^{-1} are exceeded 99% of the time in the upper and lower water column respectively and 0.018 m s^{-1} lies within this range.

From Table 2, a diffuser length of 250 m and a cross-flow velocity of 0.018 m s^{-1} will produce an effluent plume Height of 9.9 m.

The coupling between the near and far-field models will therefore comprise:

- A horizontal grid resolution of 50 m in the very fine grid domain
- A 250 m diffuser extending horizontally across exactly 5 grid cells
- A uniform horizontal distribution of effluent along the 5 grid cells of the diffuser length
- A uniform vertical distribution of effluent in those 5 grid cells from the surface down to a depth of 9.9 m.

2 RUNNING THE COUPLED MODEL

To achieve the horizontal resolution of 50 m that is required to accurately predict the concentration of effluent in the far-field, a system consisting of a regional coarse grid domain and three nested domains is used. The regional domain covers the whole of Bass Strait with a grid resolution of 2500 m, and the nested domains range from an Intermediate domain with a 1000 m grid, to a Fine domain with a 250 m grid and ultimately to a Very Fine domain with a 50 m grid. These are shown in Figure 2 (a,b) which is reproduced from the RPS MetOcean (2009) report. This report contains a detailed description of the far-field model and presents some early results, including comparisons with measurements collected in 2007.

2.1 Effluent Loading

A passive tracer of concentration, 0.001 kg m^{-3} (1 g m^{-3} or 1 mg L^{-1} or 1 ppm), and volume rate corresponding to the proposed effluent discharge rate of 0.738 m^3 s^{-1} (equivalent to 64 million litres per day) is to be released into the cells surrounding the outfall diffuser in a distribution that is equivalent to that described in Section 1.3 above (see Table 4). As explained above, the depth of the effluent cloud for the very fine grid is 9.9 m with the discharge distributed uniformly over 5 grid cells exactly matching the length of diffuser. The depth is also 9.9 m for the fine grid with the discharge going into exactly one 250 m grid cell. For the Intermediate and Regional domains the discharge is into one grid cell, but to account for the larger grid size the depth is smaller by a factor to ensure that the effluent is being discharged into the same volume (ie. horizontal grid spacing x cloud depth = 2475 m^3).

For input concentrations different from 0.001 kg m^{-3}, model output concentrations are scaled linearly, so that for an input concentration of 3.7 g m^{-3} (or 0.0037 kg m^{-3}) all model output concentrations would be multiplied by 3.7.

3 REFERENCES

RPS MetOcean (2009) Hydrodynamic Modelling Program for the Proposed Gunns Pulp Mill Report No. 1: Functional Nested Pilot Model. R1433, 23 Sept. 2009.

Worley Parsons (2010) Bell Bay Pulp Mill Outfall Near-field Modelling. 301010-00512 _ RP001 19 May 2010.

TABLES

Table 1 Reproduction of WP report Table 7-6, "Summary plume cloud characteristics for coupling."

	Velocity (m/s)	0.000	0.018	0.035	0.053	0.073	0.094	0.153	0.250	0.400
200m Diffuser	Height (m)	9.2	9.3	10.9	13.7	15.2	16.1	13.6	9.7	5.9
	Width (m)	68	71	68	53	55	61	90	147	233
	Offset (m)	0	9	20	34	38	42	56	84	123
	Initial Dilution	280	290	340	330	370	440	550	640	620
300m Diffuser	Height (m)	9.4	10.0	13.0	14.4	13.6	12.2	9.5	7.6	6.2
	Width (m)	48	50	38	41	50	60	92	148	233
	Offset (m)	0	9	23	26	32	37	53	80	129
	Initial Dilution	310	340	330	400	450	490	590	760	980
400m Diffuser	Height (m)	9.8	11.2	13.4	14.7	13.3	11.9	8.8	6.7	5.4
	Width (m)	41	43	34	38	47	57	91	147	233
	Offset (m)	0	10	21	26	31	37	53	82	131
	Initial Dilution	360	430	410	500	560	610	720	890	1130
500m Diffuser	Height (m)	10.5	12.7	13.9	14.5	13.0	11.2	7.9	6.2	5.2
	Width (m)	37	37	31	36	45	56	91	147	233
	Offset (m)	0	10	20	25	30	36	52	82	135
	Initial Dilution	440	520	480	580	660	700	810	1030	1360

Table 2 Reproduction of WP report Table 8-1, "Recommended effluent cloud parameters for one way coupling."

Diffuser Length	200m	250m (est)	300m	350m (est)	400m	450m (est)	500m
Height (m)	13.2	9.9	10	10	9.6	9.5	9.4
Width (m)	50	50	50	50	50	50	50
Offset (m)	0	0	0	0	0	0	0
Initial Dilution	300	280	340	390	430	480	530
Nom. Port Diam.	150mm	100mm	100mm	100mm	100mm	100mm	100mm

Note: The 250m, 350m and 450m length diffuser values were estimated by extrapolating and interpolating the 300m, 400m and 500m diffuser values. The distinct difference in the 200m diffuser cloud height, relates to the increase from a 100mm to a 150mm duckbill valve

Table 3 Exceedence table of currents measured at the proposed outfall site

Location	**Outfall**		
Latitude:	41°00' 17" S	Client:	Gunns Limited
Longitude:	146°51' 35" E	Project:	J2759
Location Water Depth:	26.90 m MSL		

Period: 10:00 02 September 2009 to 08:50 03 December 2009
Current Speed (m s^{1-})
Current Direction (°)

	Current Speed (m s^{1-})				Total	Exceedence Percentile											Main Direction(s)[1]
	Min	Max	Mean	Std. Dev	Records	99	95	90	75	50	30	20	10	5	2	1	(towards)
14.4 m ASB	0.00	0.57	0.13	0.0793	5033	0.02	0.03	0.04	0.07	0.12	0.17	0.20	0.24	0.28	0.33	0.36	NE ENE SW WSW
13.3 m ASB	0.00	0.50	0.13	0.0761	10542	0.01	0.03	0.04	0.07	0.12	0.16	0.19	0.24	0.27	0.32	0.34	SSW SW WSW
12.2 m ASB	0.00	0.49	0.13	0.0710	10577	0.01	0.03	0.04	0.07	0.12	0.16	0.18	0.22	0.26	0.30	0.32	SW WSW
11.1 m ASB	0.00	0.46	0.12	0.0666	10628	0.01	0.03	0.04	0.07	0.11	0.15	0.18	0.21	0.25	0.28	0.30	SW
10.0 m ASB	0.00	0.44	0.12	0.0630	10645	0.01	0.03	0.04	0.07	0.11	0.15	0.17	0.20	0.23	0.27	0.29	SW
8.9 m ASB	0.00	0.41	0.12	0.0595	10658	0.01	0.03	0.04	0.07	0.11	0.14	0.16	0.20	0.22	0.26	0.28	NE SW
7.8 m ASB	0.00	0.39	0.11	0.0569	10664	0.01	0.03	0.04	0.07	0.11	0.14	0.16	0.19	0.22	0.25	0.27	SW
6.7 m ASB	0.00	0.37	0.11	0.0538	10668	0.01	0.03	0.04	0.07	0.11	0.14	0.15	0.18	0.21	0.23	0.26	SW
5.6 m ASB	0.00	0.38	0.11	0.0511	10669	0.01	0.03	0.04	0.07	0.10	0.13	0.15	0.17	0.20	0.23	0.25	SW
4.5 m ASB	0.00	0.37	0.10	0.0476	10669	0.01	0.03	0.04	0.07	0.10	0.13	0.14	0.17	0.19	0.22	0.24	SW
3.4 m ASB	0.00	0.37	0.10	0.0443	10669	0.01	0.03	0.04	0.07	0.10	0.12	0.13	0.15	0.17	0.20	0.22	SW

Notes: 1) Main directions are where occurrence is greater than 10.0%.

Sample Interval: 10.00 minutes

Time Zone: UTC +10:00 hours	Data Source: /jobs/J2759/measured/awac/{oct09,dec09}/2759out{014,049}_current_tide.c.ext.nc
© RPS MetOcean Pty Ltd	moematrix: 11:11 28/Jan/2010 by susan (/jobs/J2759/measured/awac/dec09/stats_2759out014-049.currents.AWAC.ps)

Table 4 Effluent loading pattern into grid cells of each model domain

	Regional	**Intermediate**	**Fine**	**Very Fine**
Horizontal Grid Spacing (m)	2500	1000	250	50 m
Depth of Effluent cloud (m)	0.99	2.475	9.9	9.9 (over 5 grid cells)
Initial dilution for a the median cross-flow current of 0.1 m s^{-1} *	338:1	338:1	338:1	338:1

* This is based on the steady-state solution of a point source of known discharge rate (0.738 $m^3 s^{-1}$) mixing with a volume flux equal to the product of the horizontal grid length, depth of effluent cloud and the median cross-flow current, assuming a 100% mixing efficiency

FIGURES

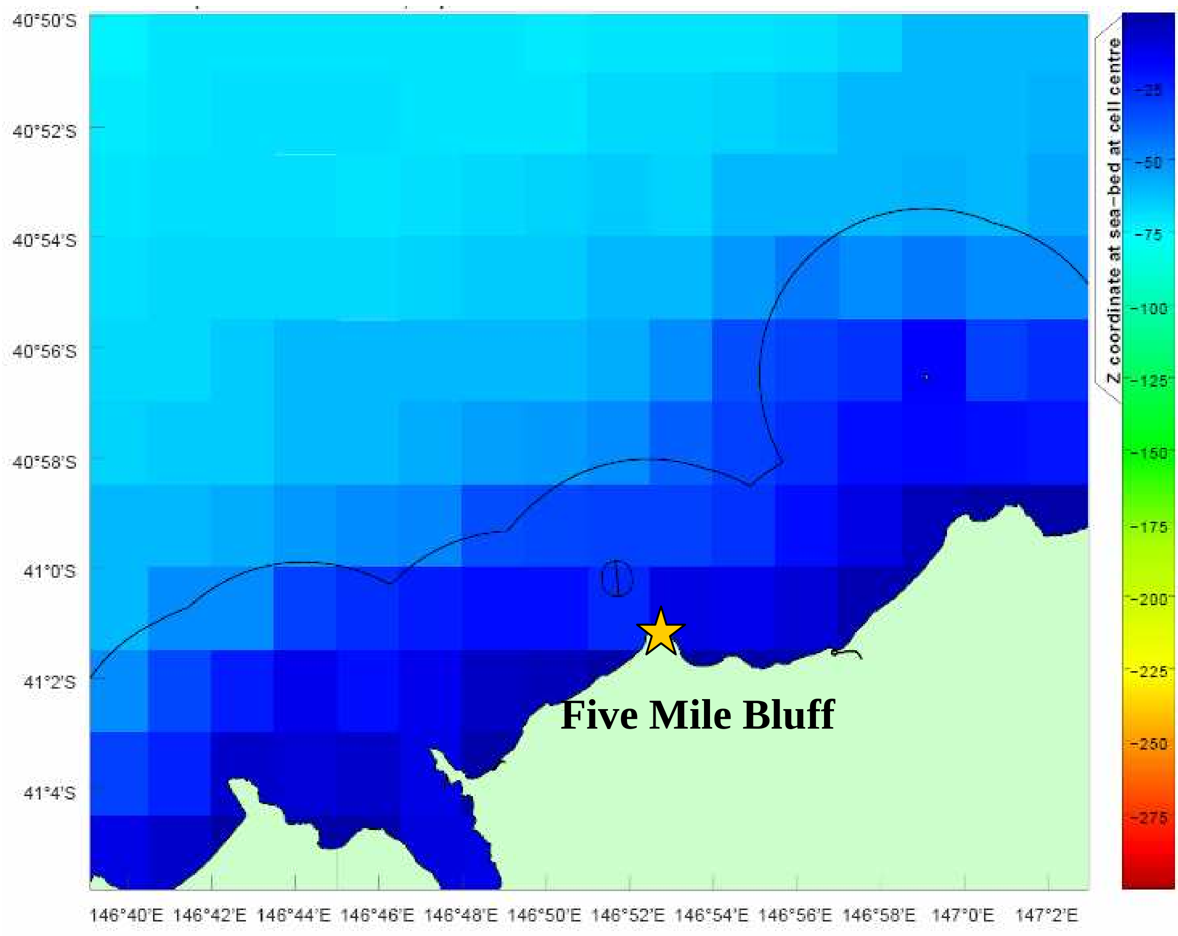

Figure 1 Location of the outfall pipe in the centre of the mixing zone (small ellipse), 2.7 km offshore Five Mile Bluff. Also shown is the boundary between State and Commonwealth waters.

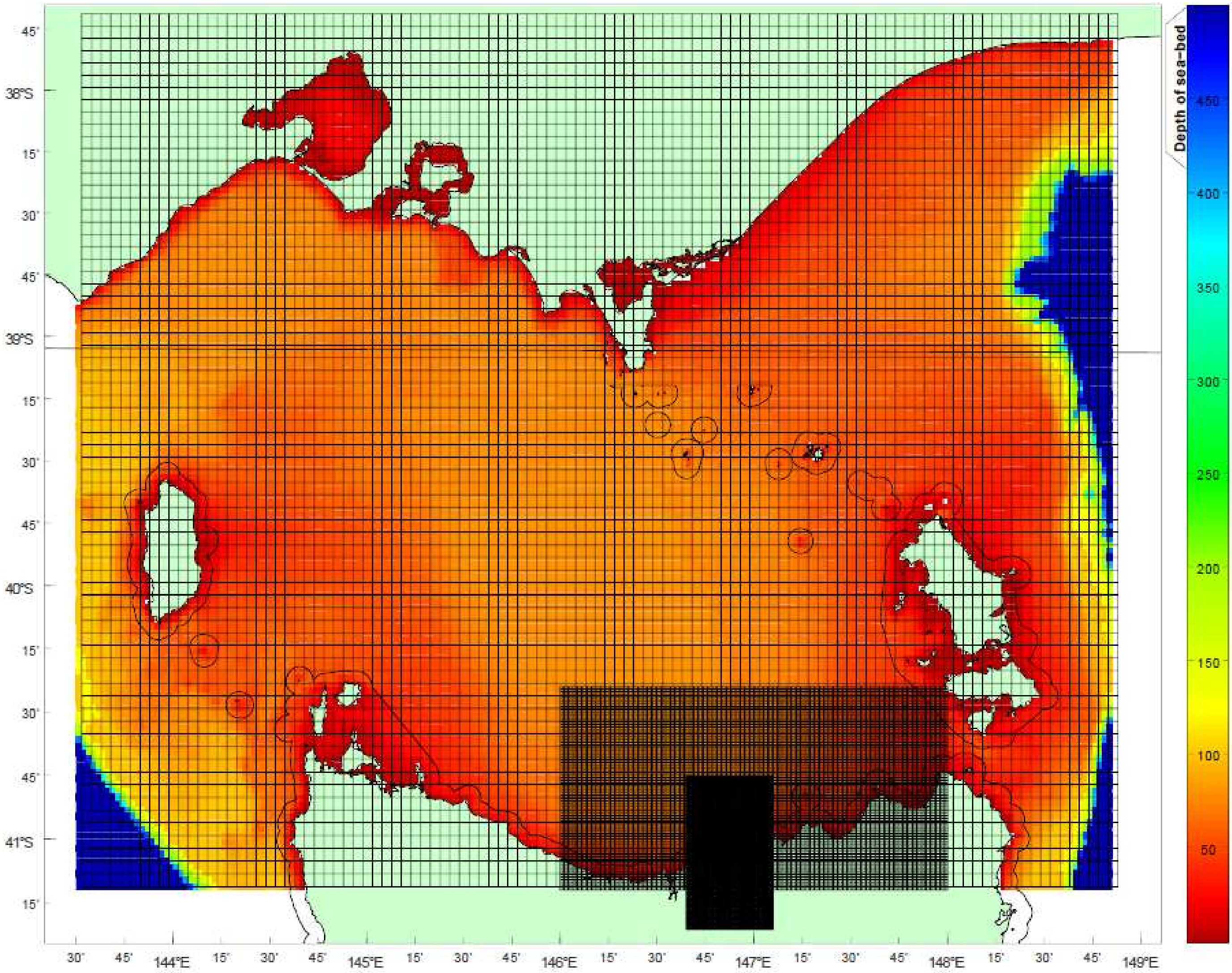

Figure 2(a) Proposed nested regional (0.025^0), intermediate (0.01^0) and fine (0.0025^0) grid model domains. The boundary between Commonwealth and State waters is also shown.

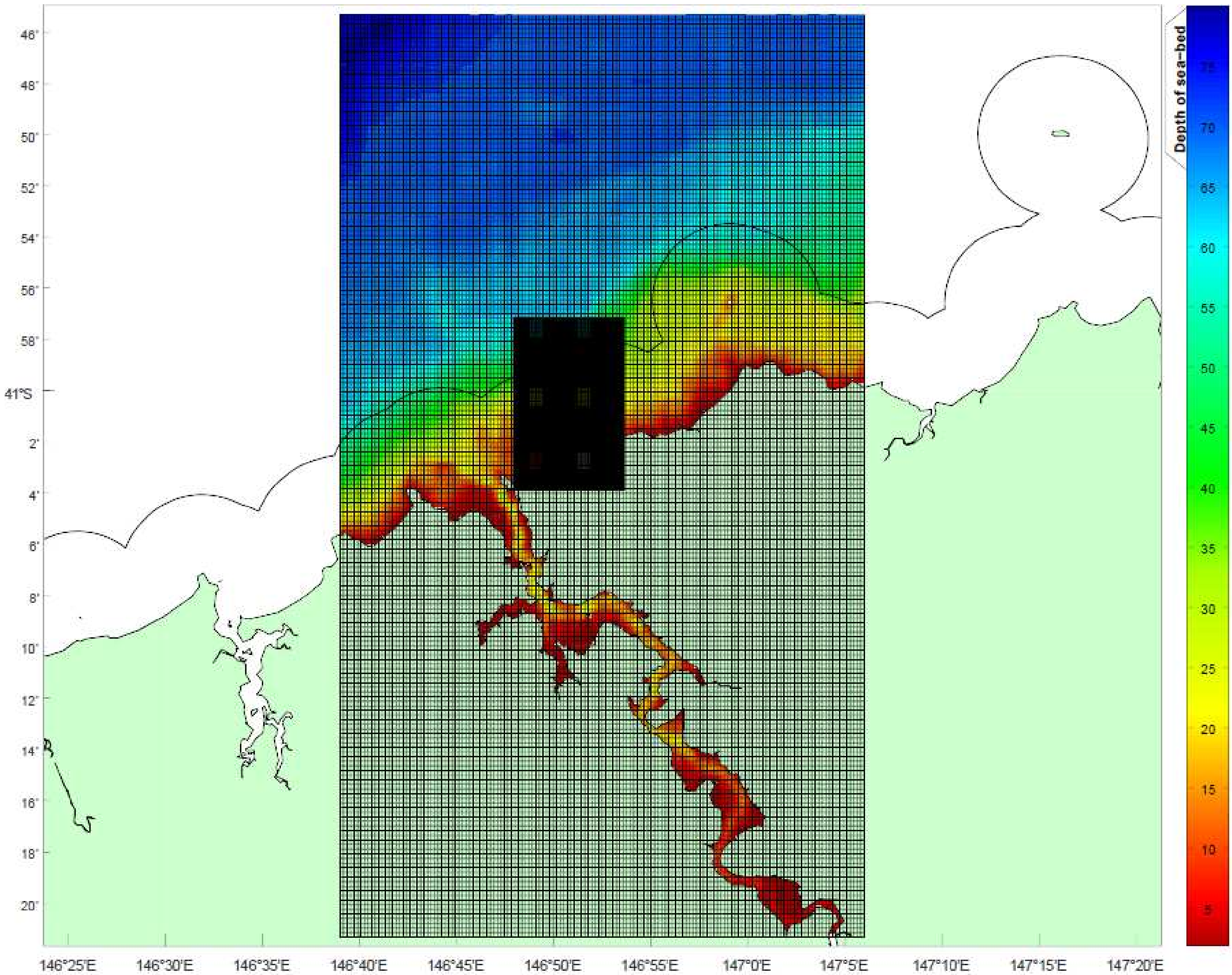

Figure 2(b) Proposed nested fine (0.0025^0) and very fine (0.0005^0) grid model domains. The boundary between Commonwealth and State waters is also shown.