日本財団 助成事業
The Nippon Foundation
競艇の交付金による日本財団の助成金を受けて作成しました。

平成１７年度
危険物の海面・大気拡散防止策
及び
予測モデル開発のための調査研究報告書
（その１）

平成１８年３月

社団法人　日本海難防止協会

ま　え　が　き

この報告書は、当協会が日本財団の助成金及び日本海事財団の補助金を受け、平成１７年度に実施した「危険物の海面・大気拡散防止策及び予測モデル開発のための調査研究」のうち、「HNS 海面・大気拡散予測モデルに関する研究」の結果について取りまとめたものである。

2000 年 3 月、OPRC 条約 HNS 議定書が採択され、従来、油の海上流出事故への準備及び対応を内容としていた同条約は、有害・危険物（HNS）の流出事故も対象とすることとなった。

ところで、HNS 輸送中の海上流出事故時の対応策に関しては、輸送されている HNS の種類及び特性が多種多様であることなどから、世界的にも確立した手法が存在しないのが現状である。また、緊急時の対応策を支援するための関係情報の整備も十分になされていない。

特に揮発性の高い HNS に関しては、海上輸送事故時の対応を安全、かつ、有効に実施するための海面･大気拡散防止策の研究が十分になされていない。加えて、海面･大気拡散の状況をできる限り正確に予測するためのパソコン起動の簡易なモデルが必要不可欠であるにもかかわらず、我国には存在しないのが現状である。

本調査はこのような状況に鑑み、当該予測モデル開発に係る前年度の基礎調査結果を踏まえ、プロトタイプモデルの試作及び同モデルの検証・評価を行うことを目的としたものである。

本調査が、HNS 海上流出事故時の準備及び対応能力の向上に資することを切に期待する。

平成 18 年 3 月

社団法人　日本海難防止協会

序章　調査概要

【調査目的】

2000 年 3 月、OPRC 条約 HNS 議定書が採択され、従来、油の海上流出事故への準備及び対応を内容としていた同条約は、有害・危険物（以下、「HNS」と呼ぶ）の流出事故も対象とすることとなった。

しかしながら、HNS 輸送中の海上流出事故時の対応策に関しては、輸送されている HNS の種類および特性が多種多様であることなどから、世界的にも確立した手法が存在しないのが現状である。また、緊急時の対応策を支援するための関係情報の整備も十分になされていない。

特に揮発性の高い HNS に関しては、海上輸送事故時の対応を安全かつ有効に実施するための海面・大気拡散防止策の研究が十分になされていない。加えて、海面・大気拡散の状況を出来る限り正確に予測するためのパソコン軌道の簡易なモデルが必要不可欠であるにもかかわらず､我が国に存在しないのが現状である。

本調査は、昨年度に実施された必要な基礎調査結果をもとに、パソコン軌道の予測モデルのプロトタイプを試作するとともに、その検証・評価を実施し、HNS 輸送中の海上流出事故時における準備及び対応能力の向上に資する事を目的とする。

【調査内容】

◎ 対象エリア等の選定

外航ケミカルタンカーによるＨＮＳの沖荷役が行われている東京湾内の特定錨地の周辺海域等、プロトタイプモデルによるケーススタディの対象とする海域、対象とするＨＮＳ等を過去の関連調査研究によって蓄積した知見に基づき選定する。

◎ プロトタイプモデルの試作

平成 16 年度の開発計画案および上記「対象エリア等の選定」結果に基づき、プロトタイプモデルを試作する。

◎ プロトタイプモデルの検証・評価

過去の関連調査研究によって蓄積した知見に基づき、プロトタイプモデルの有効性等について検証・評価を行う。

◎ **課題の整理等**

今後の課題等を抽出・整理する。

第１章 対象エリア等の選定

本年度事業で試作するプロトタイプモデルによるケーススタディの対象とする海域、対象とするＨＮＳ等を過去の関連調査研究によって蓄積した知見に基づき選定する。

１．１ 予測する海域の選定

HNS の海面・大気拡散予測を実施する海域を選定するためには、HNS 海上輸送状況についての把握が必要である。

平成 15 年度に（社）日本海難防止協会により実施された、わが国周辺の HNS 海上輸送船舶航行ルートの調査結果（図 1.1-1）から、船舶の航路線は、特に東京湾，伊勢湾，大阪湾に集中している事が判明している。また、海上保安統計年報第 53 巻（平成 14 年 1 月 1 日～平成 14 年 12 月 31 日）のデータに基づきまとめられた、わが国における港別危険物荷役状況（表 1.1-1(1)～(2)）を見ると、港別の荷役量は、千葉港が最も多く、次いで喜入港、京浜港（横浜区)、京浜港（川崎区）が続いている。さらに、港別取扱量を海域別に集計した結果（図 1.1-2）では、荷役量および隻数ともに東京湾，伊勢湾、大阪湾の 3 海域で全体の約半数（荷役量 48%、隻数 47%）を占める事が示され、特に東京湾が突出していた。

以上より、東京湾は、HNS 荷役量，隻数ともにわが国で最も多く、「衝突」と言った海上輸送事故発生の可能性が高い水域と言える。

さらに、海上輸送事故発生例（図 1.1-3）を見ると、図中のメッシュのうち、濃い色で示された航行頻度の高い水域に海難事故発生箇所を示す○印が点在し、航行頻度の高い箇所と海難事故が重なっていることが分かる。

以上の状況から、本年度事業においてプロトタイプモデルにより予測を実施する海域は、HNS 荷役量，隻数ともにわが国で最も多い東京湾を対象とする。

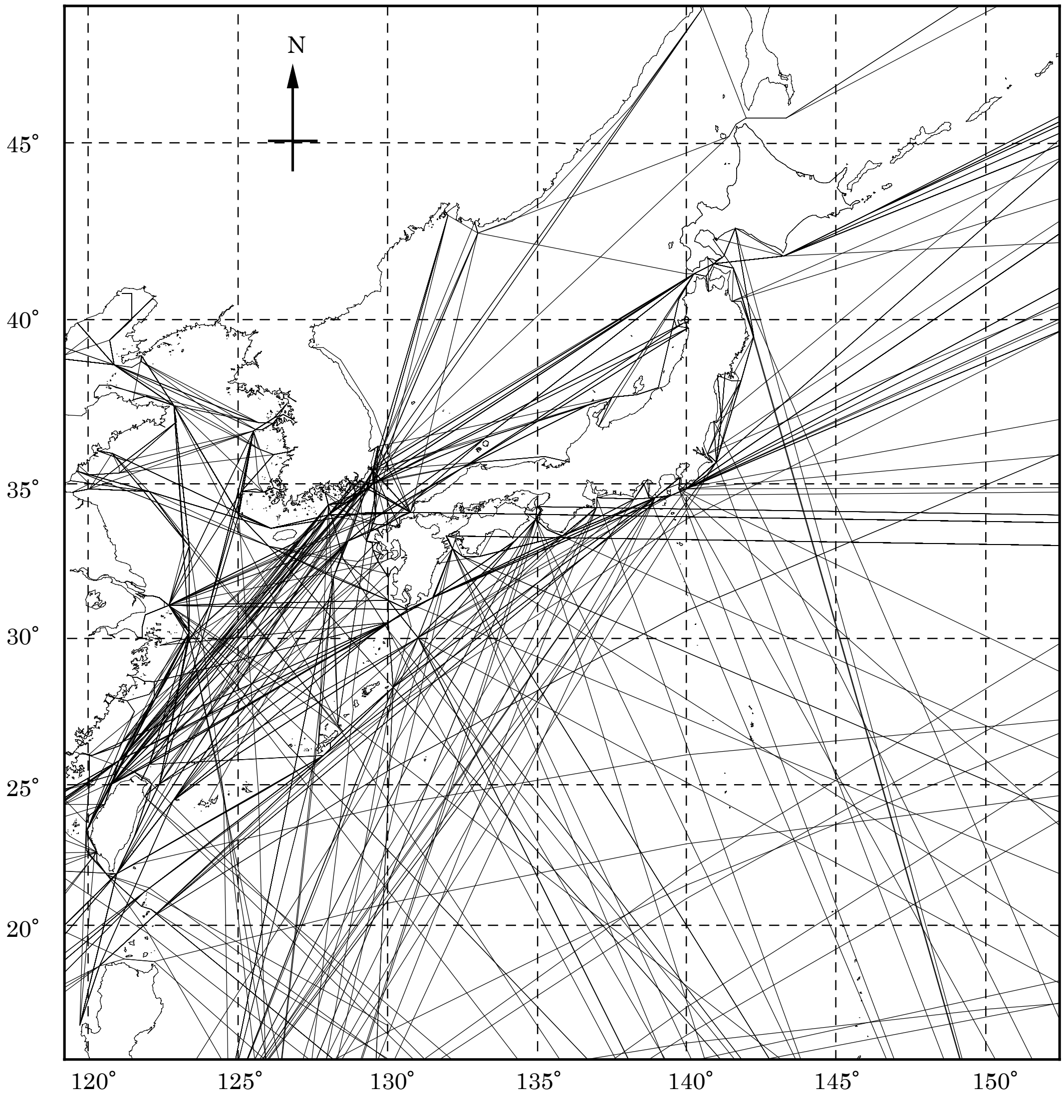
流出事故の発生を想定する水域は、平成 15 年度に（社）日本海難防止協会により実施された「HNS 海上輸送事故の典型モデルの活用」で特定された水域と同様に、次の選定条件のもと、「中の瀬航路北端付近」（図 1.1-3 赤丸印）とする。

【流出事故発生水域の選定条件】

(a) HNS 輸送船舶の航路上であること

(b) 航行密度の高い水域上であること

(c) 過去に衝突事故の発生している海域であること



（出典「危険物の海上輸送時の事故対応策の研究報告書（その１）」平成 13 年度, 平成 15 年度（社）日本海難防止協会）

図 1.1-1 わが国周辺海域を航行した HNS 輸送船舶の使用航路（1997 年）

表 1.1-1(1) わが国における港別危険物荷役状況 （平成 14 年 1 月 1 日～平成 14 年 12 月 31 日）

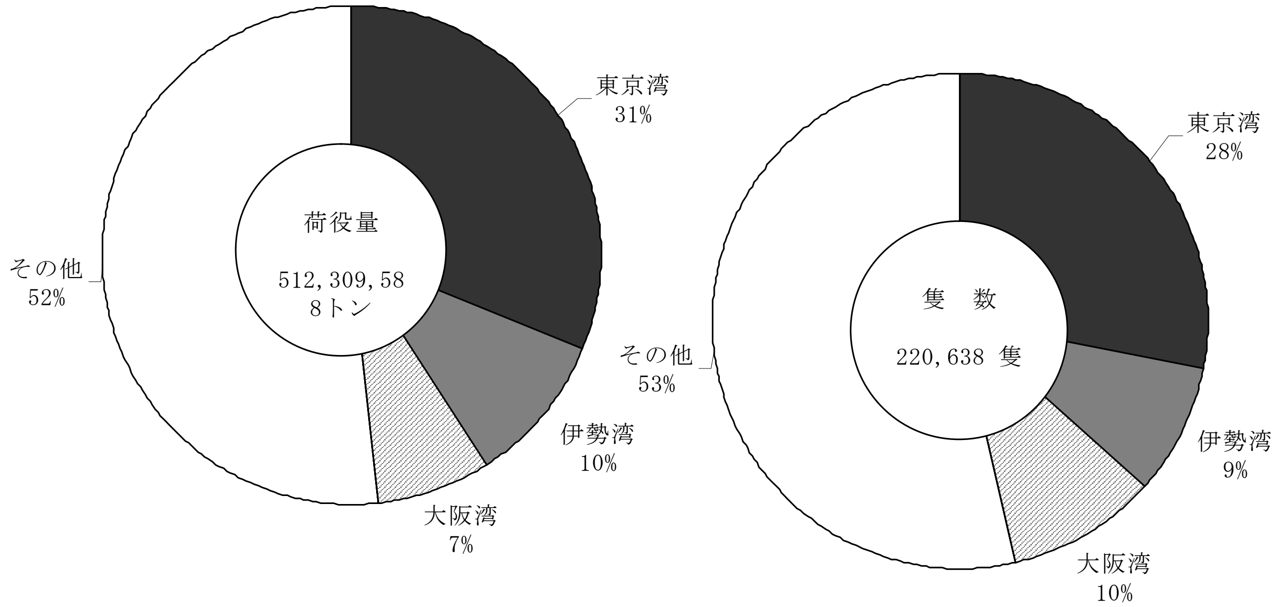
	種別	高圧ガス		腐食性物質		低引火性液体		中引火性液体		高引火性液体		その他		合計		割合	
順位	港別	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	単独	積算
1	千葉	3,923	11,337,569	1,585	747,268	6,782	48,292,161	3,286	2,360,578	9,129	13,914,964	658	637,605	25,363	77,290,145	15.1%	15.1%
2	喜入	0	0	0	0	380	45,053,191	2	123,722	89	5,439,143	0	0	471	50,616,056	9.9%	25.0%
3	京浜(横浜区)	2,717	11,791,927	1,188	1,211,277	3,788	16,478,055	1,658	1,064,912	5,915	7,328,820	1,000	637,070	16,266	38,512,061	7.5%	32.5%
4	京浜(川崎区)	1,951	2,098,936	267	109,847	3,896	26,817,639	1,692	1,038,124	3,383	3,870,021	348	254,401	11,537	34,188,968	6.7%	39.2%
5	水島	2,224	1,264,243	1,911	674,555	4,110	25,997,336	1,993	1,448,174	3,665	4,318,067	285	253,567	14,188	33,955,942	6.6%	45.8%
6	大阪(堺泉北区)	2,647	9,628,816	1,204	774,389	1,479	16,728,052	1,309	1,907,236	3,215	2,518,026	335	234,112	10,189	31,790,631	6.2%	52.0%
7	四日市	2,006	7,924,326	1,082	610,360	1,869	18,678,868	755	438,087	2,333	2,812,069	256	262,460	8,301	30,726,170	6.0%	58.0%
8	名古屋	1,055	8,243,168	1,602	741,522	2,991	3,880,812	1,220	1,898,621	2,270	2,964,259	730	449,761	9,868	18,178,143	3.5%	61.5%
9	鹿島	1,172	1,429,896	1,251	1,298,834	757	10,738,766	553	649,910	997	2,233,478	228	177,163	4,958	16,528,047	3.2%	64.8%
10	大分	2,050	3,477,586	597	276,087	1,575	9,373,164	697	508,979	1,466	1,755,889	164	103,492	6,549	15,495,197	3.0%	67.8%
11	徳山下松	2,514	2,465,372	4,598	2,001,334	1,220	7,978,411	1,190	712,837	1,601	2,048,329	314	145,724	11,437	15,352,007	3.0%	70.8%
12	室蘭	80	54,290	16	8,542	322	1,186,777	250	8,647,855	785	2,353,817	43	88,461	1,496	12,339,742	2.4%	73.2%
13	和歌山下津	221	102,025	449	133,310	2,362	8,298,147	448	1,587,830	1,545	2,120,844	88	52,392	5,113	12,294,548	2.4%	75.6%
14	仙台塩釜	738	1,371,351	63	33,540	400	7,297,562	106	201,479	1,016	1,882,191	59	120,131	2,382	10,906,254	2.1%	77.7%
15	苫小牧	318	185,463	192	160,325	483	7,252,510	165	61,624	1,008	3,058,284	358	74,996	2,524	10,793,202	2.1%	79.8%
16	岩国	283	187,183	201	67,513	1,058	4,840,923	285	229,794	1,259	2,952,171	100	59,857	3,186	8,337,441	1.6%	81.5%
17	姫路	669	4,649,736	1,061	572,315	707	1,201,568	844	1,010,290	900	804,754	97	44,679	4,278	8,283,342	1.6%	83.1%
18	坂出	399	409,681	240	74,780	1,173	6,009,333	159	125,632	802	936,653	122	77,192	2,895	7,633,271	1.5%	84.6%
19	新潟	329	4,437,193	143	102,377	409	1,082,805	188	470,074	465	1,311,354	94	3,610	1,628	7,407,413	1.4%	86.0%
20	木更津	813	6,294,533	0	0	0	0	65	58,800	133	124,300	0	0	1,011	6,477,633	1.3%	87.3%
21	金武中城	164	89,819	4	1,198	342	4,829,108	54	60,349	522	964,915	7	12,000	1,093	5,957,389	1.2%	88.4%
22	伏木富山	86	93,946	0	0	192	2,611,144	118	138,445	104	306,701	16	8,631	516	3,158,867	0.6%	89.1%
23	宇部	572	323,435	1,356	502,667	1,100	871,639	348	360,599	840	829,840	416	222,644	4,632	3,110,824	0.6%	89.7%
24	秋田船川	90	68,070	181	464,110	97	1,372,368	48	244,360	296	888,768	13	8,067	725	3,045,743	0.6%	90.3%
25	京浜(東京区)	847	96,503	629	39,525	611	581,593	411	24,550	3,652	2,119,175	1,072	70,601	7,222	2,931,947	0.6%	90.8%
26	博多	1,073	516,056	29	842	1,617	1,410,719	199	41,907	893	850,343	161	7,516	3,972	2,827,383	0.6%	91.4%
27	関門(若松)	142	1,769,917	725	297,548	197	147,325	539	346,657	107	37,311	190	152,280	1,900	2,751,038	0.5%	91.9%
28	神戸	372	290,554	524	131,752	277	397,622	1,754	976,967	1,152	585,004	400	93,783	4,479	2,475,682	0.5%	92.4%
29	清水	479	479,058	497	155,012	450	1,121,522	205	199,592	263	461,366	215	39,578	2,109	2,456,128	0.5%	92.9%
30	小名浜	250	216,060	253	207,766	253	500,126	68	83,020	316	1,272,381	29	10,925	1,169	2,290,278	0.4%	93.3%
31	青森	189	693,318	24	1,722	79	321,231	10	554	294	710,673	345	3,058	941	1,730,556	0.3%	93.7%
32	八戸	259	90,460	222	186,248	4,477	406,007	67	168,823	296	878,191	10	476	5,331	1,730,205	0.3%	94.0%
33	釧路	123	74,890	99	99,202	94	369,359	29	66,801	452	1,103,388	10	10,813	807	1,724,453	0.3%	94.3%
34	東播磨	462	428,672	1,356	556,373	389	387,581	149	132,458	414	207,038	0	0	2,770	1,712,122	0.3%	94.7%
35	新居浜	856	422,054	1,396	735,938	205	134,495	471	202,744	441	84,981	837	119,315	4,206	1,699,527	0.3%	95.0%
36	関門(若松除)	357	167,106	324	111,912	545	590,319	192	58,149	577	681,038	407	48,044	2,402	1,656,568	0.3%	95.3%
37	大阪(大阪区)	63	1,054	2,240	893,206	687	181,380	771	182,428	1,324	264,843	620	64,622	5,705	1,587,533	0.3%	95.6%
38	松山	891	27,026	246	80,599	573	658,803	238	285,820	626	515,320	14	2,850	2,588	1,570,418	0.3%	95.9%
39	鹿児島	902	252,767	3	10	299	600,855	4	672	527	619,719	48	364	1,783	1,474,387	0.3%	96.2%
40	泉州	0	30	0	0	0	0	0	0	386	1,357,416	0	0	386	1,357,446	0.3%	96.5%
41	柳井	69	1,268,503	0	0	22	4,400	0	0	0	0	0	0	91	1,272,903	0.2%	96.7%
42	金沢	180	159,423	11	11,940	376	624,328	28	46,088	164	414,855	0	0	759	1,256,634	0.2%	97.0%
43	函館	193	48,042	135	4,558	68	170,804	45	22,490	612	808,234	459	3,300	1,512	1,057,428	0.2%	97.2%
44	田子の浦	45	19,850	199	96,436	396	438,435	70	61,537	189	275,714	29	19,670	928	911,642	0.2%	97.4%
45	伊万里	593	824,564	51	19,426	5	118	19	2,496	17	2,632	76	48,653	761	897,889	0.2%	97.5%
46	広島	607	387,200	1	0	257	266,738	78	20,059	183	220,338	19	12	1,145	894,347	0.2%	97.7%
47	直江津	8	94	15	9,569	102	354,570	59	145,822	128	317,394	12	245	324	827,694	0.2%	97.9%
48	境	74	540	36	34,119	131	351,794	0	0	166	418,335	0	0	407	804,788	0.2%	98.0%
49	三池	52	39,611	279	156,549	108	169,510	199	186,968	67	166,019	60	28,026	765	746,683	0.1%	98.2%
50	七尾	140	540,354	0	0	13	26,560	0	0	55	74,912	2	2	210	641,828	0.1%	98.3%
	小　計	36,247	86,742,270	28,485	14,396,402	49,723	287,086,533	23,038	28,604,913	57,039	82,184,277	10,746	4,652,148	205,278	503,666,543	98.3%	

(出典：「危険物の海上輸送時の事故対応策の研究報告書（その1）」：平成15年度（社）日本海難防止協会)

表 1.1-1(2) わが国における港別危険物荷役状況（平成 14 年 1 月 1 日～平成 14 年 12 月 31 日）

	種別	低引火性液体		中引火性液体		高引火性液体		高圧ガス		腐食性物質		その他		合計		割合	
順位	港別	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	隻数	荷役量(トン)	単独	積算
51	福井	146	380,461	10	14,003	79	162,343	0	0	57	57,584	0	0	292	614,391	0.1%	98.4%
52	衣浦	15	60,350	0	0	0	0	312	522,449	12	6,900	28	12,538	367	602,237	0.1%	98.6%
53	福山	59	130,108	279	203,423	137	187,334	83	20,028	141	23,684	47	26,733	746	591,310	0.1%	98.7%
54	むつ小河原	6	536,696	0	0	0	0	0	0	0	0	35	11,638	41	548,334	0.1%	98.8%
55	留萌	14	104,218	0	0	126	435,997	0	0	0	0	0	0	140	540,215	0.1%	98.9%
56	日立	214	330,881	26	157	92	139,310	2	4	0	1	18	340	352	470,693	0.1%	99.0%
57	唐津	126	109,318	0	0	25	40,483	203	317,501	0	0	0	0	354	467,302	0.1%	99.1%
58	酒田	57	133,244	0	0	181	319,249	12	11	0	0	11	234	261	452,738	0.1%	99.1%
59	三河	157	164,994	35	3,790	175	142,191	196	135,845	0	0	0	0	563	446,820	0.1%	99.2%
60	細島	111	201,100	0	0	81	169,410	129	61,625	6	1,270	16	769	343	434,174	0.1%	99.3%
61	尼崎西宮芦屋	191	223,716	51	17,270	173	79,220	18	6,019	233	73,042	5	615	671	399,882	0.1%	99.4%
62	長崎	287	249,292	0	0	74	93,915	116	36,924	0	0	14	76	491	380,207	0.1%	99.5%
63	高知	315	257,416	0	0	38	79,386	26	12,990	36	8,345	0	0	415	358,137	0.1%	99.5%
64	釜石	15	90,617	0	0	76	181,627	48	30,140	0	0	0	0	139	302,384	0.1%	99.6%
65	高松	526	218,717	342	4,053	358	63,892	79	3,315	24	645	179	6,499	1,508	297,121	0.1%	99.7%
66	阪南	92	142,270	0	0	212	99,610	0	1,100	0	0	1	9	305	242,989	0.0%	99.7%
67	稚内	61	32,293	0	0	313	135,498	418	5,211	0	0	8	12	800	173,014	0.03%	99.7%
68	三島川之江	0	0	0	0	61	5,152	5	6	534	141,743	71	22,823	671	169,724	0.03%	99.8%
69	小松島	374	100,696	1	50	175	63,635	0	0	63	3,866	1	0	614	168,247	0.03%	99.8%
70	小樽	4	11,507	0	0	85	135,275	6	8,280	0	0	1	0	96	155,062	0.03%	99.8%
71	佐世保	229	103,248	4	0	87	41,998	96	6,856	0	0	0	0	416	152,102	0.03%	99.9%
72	三田尻中関	1	0	167	104,998	9	1,164	125	21	95	21,575	0	0	397	127,758	0.02%	99.9%
73	那覇	263	5,811	33	30,091	133	16,235	172	26,716	3	48	20	279	624	79,180	0.02%	99.9%
74	呉	0	0	2	600	27	568	113	186	251	69,499	217	2,552	610	73,405	0.01%	99.9%
75	尾道糸先	98	12,223	0	0	21	8,423	19	148	150	40,319	1	1	289	61,114	0.01%	99.9%
76	横須賀	41	41,000	6	10,536	23	6,681	13	5	0	0	207	565	290	58,787	0.01%	99.9%
77	両津	34	23,529	0	226	71	23,911	25	5,735	0	132	68	143	198	53,676	0.01%	99.9%
78	石巻	6	80	0	0	23	197	125	24	74	42,500	27	10,766	255	53,567	0.01%	99.9%
79	浜田	22	21,699	0	0	31	30,331	0	0	0	0	0	0	53	52,030	0.01%	99.9%
80	敦賀	0	0	20	13,846	9	13,000	1	12	22	13,914	8	7,500	60	48,272	0.01%	99.9%
81	三角	0	0	28	14,700	0	0	0	0	17	9,240	0	0	45	23,940	0.005%	99.9%
82	宇野	145	2,368	183	1,663	433	5,464	465	6,604	118	2,082	184	2,357	1,528	20,538	0.004%	99.9%
83	今治	35	53	0	246	112	5,268	184	1,490	78	4,720	1	30	410	11,807	0.002%	99.9%
84	厳原	34	4,017	10	147	8	1,950	225	230	0	0	32	33	309	6,377	0.001%	99.9%
85	名瀬	0	0	0	0	0	0	161	2,978	0	0	6	19	167	2,997	0.001%	99.9%
86	舞鶴	0	0	0	0	0	0	0	0	0	0	86	1,253	86	1,253	0.0002%	99.9%
87	田辺	0	0	0	0	6	1,053	0	0	0	0	5	2	11	1,055	0.0002%	99.9%
88	萩	153	74	0	0	109	91	181	41	0	0	0	0	443	206	0.0000%	100.0%
89	根室	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0.0%	100.0%
90	宮津	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0.0%	100.0%
小　計		3,831	3,691,996	1,197	419,799	3,563	2,689,861	3,558	1,212,494	1,914	521,109	1,297	107,786	15,360	8,643,045	1.7%	
合　計		53,554	290,778,529	24,235	29,024,712	60,602	84,874,138	39,805	87,954,764	30,399	14,917,511	12,043	4,759,934	220,638	512,309,588		
割合（単独）			56.8%		5.7%		16.6%		17.2%		2.9%		0.9%		100.0%		
（積算）			56.8%		62.4%		79.0%		96.2%		99.1%		100.0%				

（出典：「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成15年度（社）日本海難防止協会）



（出典：「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成15年度（社）日本海難防止協会）

図 1.1-2 東京湾、伊勢湾及び大阪湾の全体における荷役量及び隻数の関係

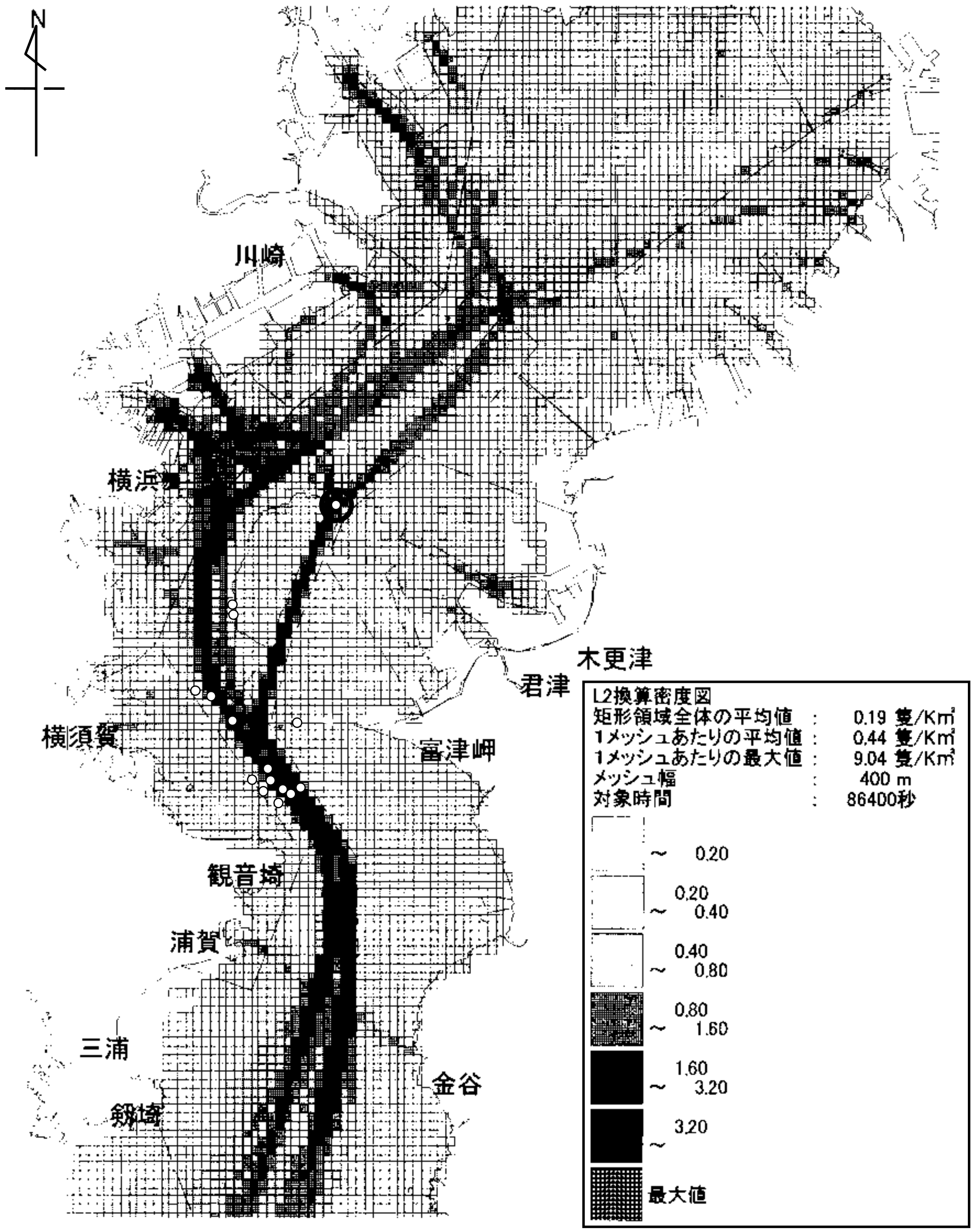
注）東京湾、伊勢湾及び大阪湾以外の海域を「その他」とした。

東京湾、伊勢湾及び大阪湾の各港は、以下のとおりである。

東京湾
千葉
京浜（横浜区）
京浜（川崎区）
木更津
京浜（東京区）
横須賀

伊勢湾
四日市
名古屋
衣浦
三河

大阪湾
大阪（堺泉北区）
神戸
大阪（大阪区）
泉州
尼崎西宮芦屋
阪南



（資料 :「東京湾船舶航行実態調査報告書」: 平成 12 年度（社）日本海難防止協会

出典 :「危険物の海上輸送時の事故対応策の研究報告書（その1）」: 平成15年度（社）日本海難防止協会）

図 1.1-3 東京湾内の航行頻度と海難事故の関係

１．２ 拡散物質の選定

（１）拡散物質

東京湾を航行する HNS 輸送船舶には外航船舶および内航船舶があり、各船舶で流出事故の可能性がある。

前掲表 1.1-1(1)で見られる様に、東京湾を代表する千葉港、京浜港（横浜）のうち、流出事故発生を想定した地点に近い京浜港（横浜）では、大型の外航ケミカルタンカーに小型の内航ケミカルタンカーが接舷し、ケミカル類を「瀬取り」あるいは「瀬渡し」する「沖荷役」が行われている。この京浜港（横浜）における外航船舶の沖荷役のＨＮＳ取扱量（表 1.2-1(1)）は、エタノールが最も多く、次いで二塩化エチレンそしてメチルエチルケトンとなっている。

また、内航船舶のよるわが国のＨＮＳ輸送品目と輸送量を見ると（表 1.2-1(2)）、キシレンが最も多く、次いでベンゼンそしてスチレンとなる。

これら取扱量や輸送量の多い物質が、流出事故の対象物質となる可能性が高くなると考えられる事から、外航船舶の取扱量および内航船舶の輸送量の多い上位種を対象物質として選定する。ただし、外航船舶の取り扱い第２位の二塩化エチレンは比重が大きく、流出後すみやかに海中へ沈み込む事が想定される。本年度研究では、流出物質の海面および大気中の拡散予測を行うためのモデル開発である事を鑑み、この二塩化エチレンを対象外とする。

【選定する対象物質】

外航船舶の取扱量上位２種：エタノール,メチルエチルケトン

内航船舶の輸送量上位３種：キシレン,ベンゼン,スチレン

（２）設定する各物質の流出量

HNS 流出量を設定するためには、HNS 輸送船舶の船型を推定する必要がある。

外航船舶については、京浜港（横浜）における外航ケミカルタンカーの船型別入港実績が図 1.2-1 に示す様にとりまとめられており、これによれば 4,000～5,000GT 級が約 30%を占め、最も多い。

また、内航船舶については、国内航行船による HNS の総トン数階級別輸送量が表 1.2-2 にとりまとめられており、100～500GT 級が最も多い結果となっている。

以上の結果より、本年度事業では比較的大きな被害の発生を想定して、5,000GT 級の外航船舶を HNS 輸送船舶の対象とする。

次に、設定された 5,000GT 級による HNS 流出事故が発生した場合の流出量を想定する。

表 1.2-3 に示すケミカルタンカーの要目および参考として掲載した海上保安庁「排出油防除計画」記載の「海洋汚染想定の指針」より、本船の載貨重量トン数は 9,306.1 トンと推定され、その９％の 838kl が流出量になると推定される。

従って、設定する流出量は 838kl とする。

【設定する流出量】

各物質共通：838kl

表 1.2-1(1) わが国における外航船舶の沖荷役による HNS 取扱量

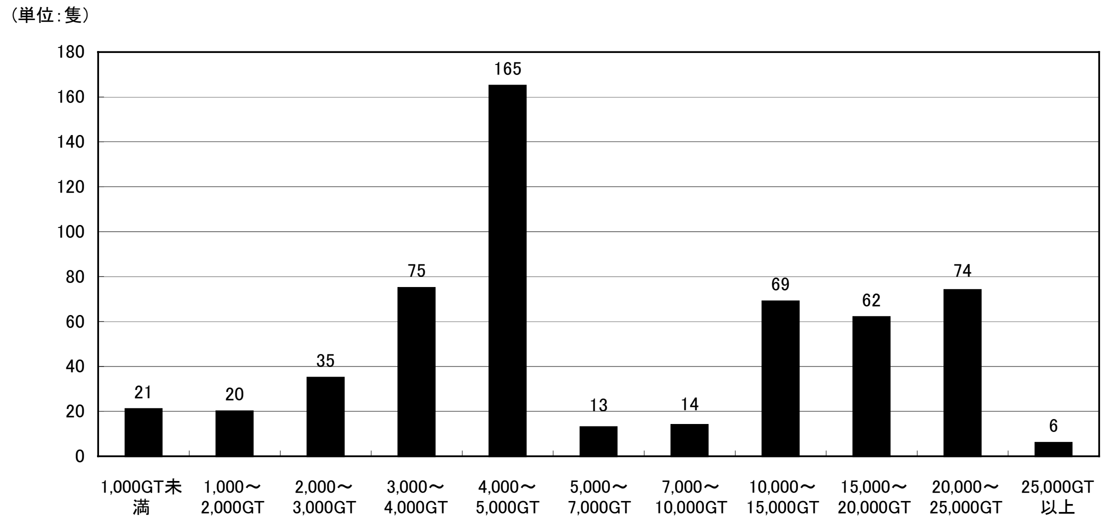
順位	品　　名	取扱量 (M/T) 京浜港 (横浜)
1	Ethanol	124,082
2	Ethylene Dichloride	113,152
3	Metyl Ethyl Ketone	50,698
4	Nonene	42,052
5	Cyclohexane	34,375
6	Phenol	32,400
7	Isopropy Alcohol	29,344
8	Acetone	27,608
9	Bunkai Gasoline	15,290
10	N-Butyl Acrylate	14,200
11	Styrene Monomer	12,000
12	Diisobutylene	11,380
13	1-Methoxy-2-Propanol	10,355
14	Ethyl Acrylate	9,574
15	n-Butanol	8,493
16	Epichlorohydrin	8,040
17	Dimethy Formamide	6,970
18	Decen-1(α-Olefin C-10)	6,910
19	Hexene-1	4,626
20	Methanol	4,200
21	Acrylic Acid	4,000

(出典：「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成 16 年（社）日本海難防止協会)

表 1.2-1(2) わが国における内航船舶の輸送品目と輸送量

順位	品　　名	輸送量 (M/T)
1	Xylene	1,999,866
2	Benzene	1,503,659
3	Styrene Monomer	1,439,635
4	Methanol	717,782
5	Toluene	604,530
6	Cyclohexane	528,988
7	Acrylonitrile	519,324
8	Coal tar	506,744
9	Ethylene Dichloride	418,217
10	Creosote	350,358
11	Ethanol	237,594
12	Butanol	236,113
13	Acetone	222,269
14	Methyl methacrylate	211,576
15	Acetic anihydride	201,806
16	Metyl Ethyl Ketone	200,839
17	Vinyl acetate	197,900
18	Propylbenzene	193,412
19	Octanol	167,591
20	Phenol	146,735
21	Ethlene glycol	121,374

(出典:「危険物の海上輸送時の事故対応策の研究報告書（その１)」: 平成 16 年（社）日本海難防止協会を一部改変)



（出典：「ばら積み液体化学物質の積替荷役に関わる調査委員会報告書」：平成６年（社）日本海難防止協会

「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成16年（社）日本海難防止協会）

図 1.2-1 平成 3 年度の京浜港における外航ケミカルタンカーの船型別入港実績

表 1.2-2 国内航行船による HNS の総トン数階級別輸送量（表 2.4.2-6(3)の再掲）

単位：千トン

	計	100G/T未満	100G/T以上 500G/T未満	500G/T以上 1,000G/T未満	1,000G/T以上 2,000G/T未満	2,000G/T以上
石油製品　LPG及びその他のガス	7,855	0	451	7,149	255	0
%	100.0	0.0	5.7	91.0	3.2	0.0
化学薬品	19,056	0	12,999	4,994	428	635
%	100.0	0.0	68.2	26.2	2.2	3.3
その他の化学工業品	1,247	0	563	447	121	115
%	100.0	0.0	45.1	35.8	9.7	9.2

注：貨物船を除く。　（資料：平成14年度内航船舶輸送統計年報）

（出典：「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成16年（社）日本海難防止協会）

表 1.2-3 ケミカルタンカーの要目

区分	船型	総トン数 (GT)	載貨重量トン (D.W.T)	タンク容積 (m^3)	タンク数	全長 (m)	幅 (m)	深さ (m)	満載喫水 (m)	主機馬力 (ps)	航海速力 (k't)
外航ケミカルタンカー	1,000 トン型	999.0	1,933.0	1,327	8	71.86	11.00	5.49	5.10	1,500	11.50
	3,000 トン型	3,194.0	5,568.2	6,129	15	104.14	15.02	7.80	6.47	4,500	13.50
	5,000 トン型	5,266.0	9,306.1	9,236	13	116.90	18.20	9.80	7.99	4,900	12.70
	7,000 トン型	6,738.0	12,010.7	13,485	21	124.80	18.30	10.85	9.06	7,000	14.00
	10,000 トン型	10,499.0	16,617.0	21,608	24	149.60	22.81	12.02	8.66	6,200	13.25
	15,000 トン型	14,912.0	23,299.0	29,805	36	169.75	24.47	12.50	9.56	11,500	16.25
	20,000 トン型	20,760.0	36,501.0	44,238	36	176.82	29.61	15.17	11.56	14,400	15.50
	20,000 トン型	22,587.0	39,782.0	47,140	26	179.94	32.00	−	11.20	14,400	15.00
	25,000 トン型	25,000.0	41,500.0	50,000	36	183.00	33.00	−	12.00	14,400	15.00
内航ケミカルタンカー	199 トン型	199.8	515.0	611	3	48.80	8.00	3.30	3.20	750	10.50
	299 トン型	299.0	695.0	648	3	52.52	8.60	3.90	3.45	900	10.00
	499 トン型	499.8	999.5	1,140	4	59.95	9.80	4.30	3.95	1,200	10.50
	699 トン型	699.8	1,230.0	1,211	4	67.55	11.00	5.00	4.40	1,800	12.00
	1,000 トン型	1,076.1	2,198.5	2,309	5	77.70	11.40	5.00	4.80	1,600	11.00

（出典：「ばら積み液体化学物質の積替荷役に関わる調査委員会報告書」：平成６年（社）日本海難防止協会

「危険物の海上輸送時の事故対応策の研究報告書（その１）」：平成 16 年（社）日本海難防止協会）

【参　考：海上保安庁「排出油防除計画」】

＜海洋汚染想定の指針＞

排出油事故の発生場所の想定の考え方

油が著しく大量に排出される事故発生の蓋然性の高い海域は、原則として、次の海域を考えるものとする。

港内のタンカー係留施設付近海域

タンカーの常用航路である狭水道及びその周辺海域

外洋域における貨物船の常用航路付近海域

＜排出油規模の想定の考え方＞

排出油事故の態様としては、他船との衝突、岸壁との衝突、座礁又は底触、タンクの爆発、バルブ操作のミス、油保管施設からの流出によるもの等が考えられるが、油が著しく大量に排出された場合における排出油量の想定をするに当たっては、次のような前提のもとに行うものとする。

○ 港内のタンカー係留施設付近海域

イ 排出油事故の態様として、港内のタンカー係留施設付近海域におけるタンカーの他船との衝突に伴う排出油事故とする。

ロ 排出油事故発生船舶の大きさは、当該係留施設を利用する最大級のタンカーとする。

ハ 排出油量の算定に当たっては、排出油量が破口の位置及び大きさ等により大きく異なるので、当該タンカーの載荷重量の 9%の油が排出されるものとする。((社) 日本海難防止協会「昭和 43 年度大型タンカーによる災害の防止に関する調査研究完了報告書」参照)

○ タンカーの常用航路である狭水道及びその周辺海域

イ 排出油事故の態様として、狭水道及びその周辺海域におけるタンカーの座礁又は底触に伴う排出油事故とする。

ロ 排出油事故発生船舶の大きさは、当海域を航行する最大級のタンカーとする。

ハ 排出油量の算定に当たっては、当該タンカーの最大センタータンク 2 個の底部に破口が生じたものとし、喫水線上の油が全量（最大センタータンク 2 個の全量の 1/5 が）排出されるものとする。

○ 外洋域における貨物船の常用航路付近海域

イ 排出油事故の態様として、外洋域の常用航路における貨物船の座礁または底触に伴う排出油事故とする。

ロ 排出油事故発生船舶の大きさは、当海域を航行する最大級の貨物船とする。

ハ 排出油量の算定に当たっては、当該貨物船の燃料タンクの船底部に破口が生じ、載燃料が全量排出されるものとする。

１．３ まとめ

本年度事業でプロトタイプモデルにより海面・大気拡散を予測する海域および物質に関して、これまでに蓄積された関連調査研究の成果をもとに表 1.3-1 に示す様に設定する。

表 1.3-1　プロトタイプモデルを適用する対象海域と対象物質および流出量

項　　目	設　　定
対象海域	東京湾（流出事故想定地点：中の瀬航路北端部）
対象物質	エタノール，メチルエチルケトン（外航船舶の取扱量上位種） キシレン，ベンゼン，スチレン（内航船舶の輸送量上位種）
設定流出量	各物質ともに 838kl（5,000GT 級のケミカルタンカーから流出事故を想定）

第２章 プロトタイプモデルの試作

プロトタイプモデルを試作するために、流動モデル、海面・大気拡散モデルに利用する式系や計算条件等を検討した上で、各モデルを設定する。

２．１ モデルの基本設計

プロトタイプモデルの試作のために、流動モデルおよび海面・大気拡散モデルの基本設計を行う。

２．１．１　流動モデル

海域に流出した HNS 物質は主に海表面を拡散すると考える事と、迅速な対応のためには計算時間の短縮が必要であるという観点から、流動モデルには平面二次元モデルを利用が適当である。また、正確な挙動を予測するためには、駆動力となる主要４分潮（M_2 潮流，S_2 潮流，K_1 潮流，O_1 潮流）の潮流及び河川水の流入効果を計算する必要があるが、平面二次元モデルはこれらの事が可能である。

以下に、流動モデルとして利用する平面二次元モデルの基礎式等を示す。

（１）平面二次元モデルの基礎式

平面二次元モデルにおける潮汐流や恒流などの海域の流動は、３次元の運動方程式を水面から海底まで積分して得られるものである。これは浅海長波の方程式であり、得られる流動は鉛直方向に分布を持たず、水平方向に分布を持つ。

以下に基礎式を示す。

１）運動方程式（海域の流動）

$$\frac{\partial M}{\partial t}+U\frac{\partial M}{\partial x}+V\frac{\partial M}{\partial y}=-g(\zeta+Hw)\frac{\partial \zeta}{\partial x}+fN$$
$$+Ah(\frac{\partial^2 M}{\partial x^2}+\frac{\partial^2 M}{\partial y^2})-\frac{1}{\rho}\tau sx-\frac{1}{\rho}\tau bx$$

$$\frac{\partial N}{\partial t}+U\frac{\partial N}{\partial x}+V\frac{\partial N}{\partial y}=-g(\zeta+Hw)\frac{\partial \zeta}{\partial y}+fM$$
$$+Ah(\frac{\partial^2 N}{\partial x^2}+\frac{\partial^2 N}{\partial y^2})-\frac{1}{\rho}\tau sy-\frac{1}{\rho}\tau by$$

２）運動方程式（河川水による流動）

$$\frac{\partial M}{\partial t}+\frac{\gamma}{\alpha}U_S\frac{\partial M}{\partial x}+\frac{\gamma}{\alpha}V_S\frac{\partial M}{\partial y}=-g(\zeta+Hw)\frac{\partial \zeta}{\partial x}+Ah(\frac{\partial^2 M}{\partial x^2}+\frac{\partial^2 M}{\partial y^2})$$

$$\frac{\partial N}{\partial t}+\frac{\gamma}{\alpha}U_s\frac{\partial N}{\partial x}+\frac{\gamma}{\alpha}V_s\frac{\partial N}{\partial y}=-g(\zeta+Hw)\frac{\partial \zeta}{\partial y}+Ah(\frac{\partial^2 N}{\partial x^2}+\frac{\partial^2 N}{\partial y^2})$$

３）連続式

$$\frac{\partial \zeta}{\partial t}+\frac{\partial M}{\partial x}+\frac{\partial N}{\partial y}=0$$

５）状態方程式

$$\rho=\rho(T,S)$$

ここで、M,N：線流量の x , y 方向成分であり、次式で表される。

$$M=U_S(\zeta+Hw)\alpha$$
$$N=V_S(\zeta+Hw)\alpha$$

また、U_S,V_S：海面での x , y 方向流速成分、ζ：水位、Hw：河川水厚さ、g：重力加速度、f：コリオリ力、Ah：水平渦動粘性係数、$\tau sx,\tau sy$：海表面の摩擦応力、$\tau bx,\tau by$：海底面の摩擦応力、$\alpha,\beta,\gamma,\delta,\gamma$：鉛直分布定数、$T_O$：環境水温、$T_S$：海表面の水温、Kh：水平渦動拡散係数、$Q_0$：海表面からの放熱項、$Q_1$：熱交換係数、c：海水の比熱、$\rho$：海水の密度、$U,V$：x , y 方向の沿岸流速成分である。

なお、$\alpha,\beta,\gamma,\sigma$ は次式で表される。

$$\alpha=\int_0^1 f(\eta)d\eta,\quad \beta=\int_0^1 g(\eta)d\eta,\quad \gamma=\int_0^1 f(\eta)^2 d\eta,\quad \delta=\int_0^1 f(\eta)g(\eta)d\eta,\quad \eta=h/Hw$$

（２）主な計算条件

平面二次元モデルの主な計算条件として、計算格子や潮位条件、そして河川流量が挙げられる。

以下に各計算条件について示す。

①計算格子

本年度事業では、HNS 流出事故発生地点として中の瀬航路北端部を想定するが、当該海域周辺の潮流を予測計算するためには、東京湾全域を計算する必要がある。これは、計算条件（境界条件）の設定を単純にし、結果の信頼性に配慮するためである。

そのため、計算範囲は東京湾湾口部以北の湾全域とし、1,000m 四方の正方格子で分割した。図 2.1-1 に計算範囲の地形条件を示す。

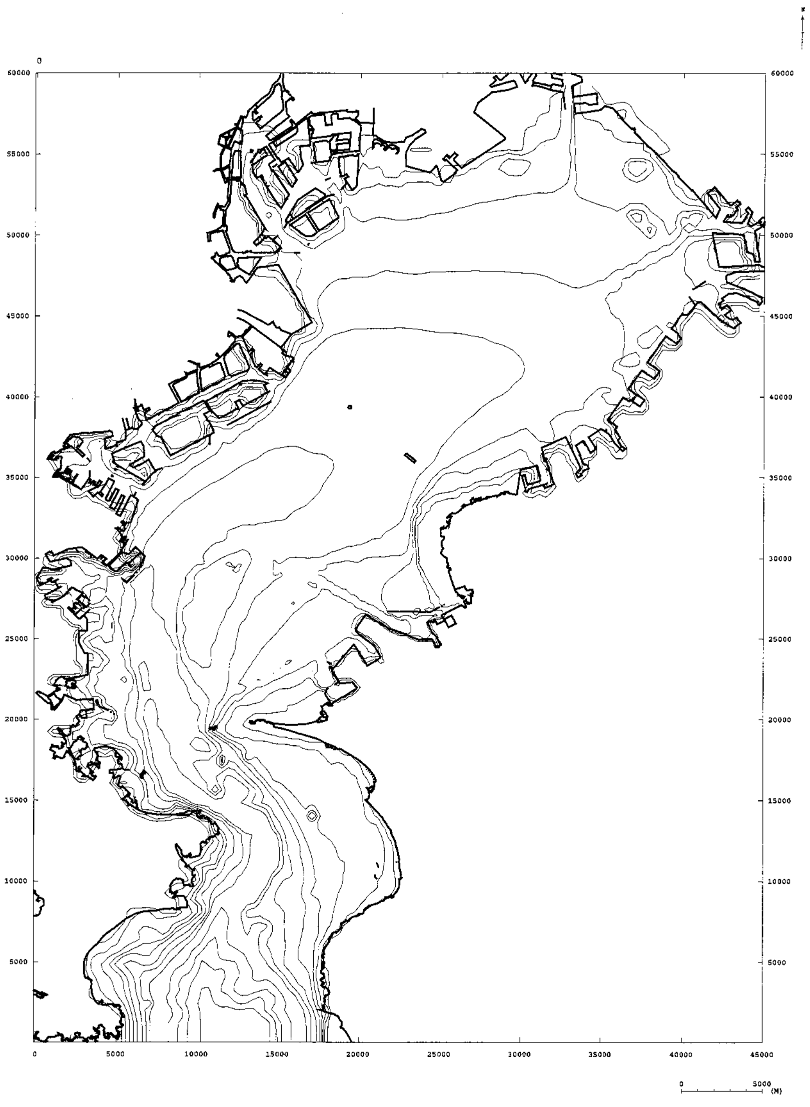


図 2.1-1 地形条件

②潮位条件

潮位条件は、日本沿岸潮汐調和定数表（海上保安庁水路部，平成４年）に記載された東京湾湾口部の間口の潮汐調和定数を設定した。

表 2.1-1 に間口の潮汐調和定数を示す。

表 2.1-1 潮位条件（間口の潮汐調和定数）

M_2潮		S_2潮		K_1潮		O_1潮	
遅角	振幅	遅角	振幅	遅角	振幅	遅角	振幅
146.1	37.3	173.7	16.8	175.7	22.9	152.5	19.7

単位：遅角（度)、振幅（cm）

③河川流量

設定する河川は東京湾に流入する主要５河川（１級河川：江戸川，荒川，隅田川，多摩川，鶴見川）とする。これら河川は、隅田川を除いて流量年表（国土交通省河川局）にデータが存在しており、各河川最下流の流量観測点の結果を利用する。しかし、流量観測点は河口から約 10～70 km上流に位置するため、その間の流量変化を補正する必要がある。そこで、環境省（1989 年）が調査した 1984～1986 年７～９月の河口部の平均流量（表 2.1-2）と、同期間の各流量観測点での流量データを用いて補正係数を求め、河口部での平水時流量を求めた。

流量データの存在しない隅田川については、環境省（1989 年）調査結果の隅田川と荒川の流量比を求め、先の算出した荒川の平水時流量を用いて設定した。

設定した各河川流量を表 2.1-3 示す。

なお、各河川の平水時流量の統計期間は次の通りである。

【統計期間】

江戸川（昭和 29 年～平成 13 年)、荒川（昭和 41 年～平成 13 年）

多摩川（昭和 26 年～平成 13 年)、鶴見川（昭和 42 年～平成 13 年）

表 2.1-2 環境省（1989 年）が調査した 1984～1986 年７～９月の平均流量(m^3/s)

江戸川	荒川	隅田川	多摩川	鶴見川
122.3	190.6	57.4	27.5	10.1

表 2.1-3 河川流量条件(平水時流量：m^3/s)

江戸川	荒川	隅田川	多摩川	鶴見川
82.34	67.31	20.28	16.18	8.28

２．１．２ 吹送流モデル

吹送流とは海面上を吹く風の応力によって起こされる流れのことであり、流動モデルの境界条件として設定されることが多い。

以下に、吹送流モデルの基礎式等について述べる。

海面上に吹く風の接線応力 τ は一般に次式で表現される。

$$\tau = \rho_a C_d W^2$$

ここで、ρ_a は空気の密度、C_d は海面の抵抗係数、W は風速である。

なお、海面上の風が連吹して波浪が十分に発達し、大気から見た海面の接線応力と海面から見た大気の接線応力が等しい場合には次式となる。

$$\tau = \rho_a C_d W^2 = \rho_w C_d' U^2$$

ここで、ρ_w は海水密度、C_d'は大気面の抵抗係数、U は海表面流速である。

また、大気と海洋の境界層が相似で $C_d = C_d$'を仮定すれば、海表面流速は次式となる。

$$U = \sqrt{\frac{\rho_a}{\rho_w}} W$$

上式は合成ベクトルの流速値を表すが、風による海面の接線応力には東西方向や南北方向に加わる力がある。各成分を次式に示す。

$$\tau_{sx} = \rho_a C_d W_x \sqrt{W_x^2 + W_y^2}$$

$$\tau_{sy} = \rho_a C_d W_y \sqrt{W_x^2 + W_y^2}$$

このようにして得られた風による海表面の流れと潮流等との合成ベクトルにより、HNS は海面を移動することとなる。したがって、事前に潮流等の流れの結果があれば、これをもとに風の影響を考慮した移動予測が可能となる。

２．１．３ 海面・大気拡散モデル

海上に流出した HNS は、前節までの流動モデルおよび吹送流モデルで計算された流れにより移動する他、物質の拡散や移動過程で起こる物性の変化により海面上での分布が決定する。また、流出した HNS が蒸発すると、風により大気中へも拡がる事となる。

物質の拡散や物性の変化、そして大気拡散を予測するモデルは平成 16 年度研究で既往知見を収集・整理しとりまとめた。

この結果、HNS の海面・大気拡散予測モデルに該当する知見は見当たらなかったが、海面拡散予測に関しては米国で開発された「NRDAM/CME（Natural Resource Damage Assessment Model for Coastal and Marine Environments)」が参考となるモデルとして存在した。なお、本モデルは、有害物質の流出による沿岸域の被害額を評価し、損害賠償額を算定するための米国一律の標準手法として「1980 年の CERCLA（Comprehensive Environmental Response, Compensation, and Liability Act : 包括的環境対策・補償・責任法)」及び「1990 年の OPA（Oil Pollution Act : 油汚染法)」のもとでの使用を目的としたアセスメントツールである。

そこで、本研究では NRDAM/CME モデルを参考として、海面上での物質の拡散および物性変化に係わる式を設計する。

なお、巻末の資料編に NRDAM/CME モデルのユーザーズマニュアルを添付する。

（１）海面拡散モデル

NRDAM/CME モデルでは、ア．拡散、イ．蒸発（揮発)、ウ．溶解，混入、エ．沈降、オ．再蒸発、カ．水平・鉛直移動、キ．堆積が考慮されている。しかし、これら項目を全て考慮する事は、モデルを複雑にし、さらに計算に要する時間を考慮すると現実的とは言えない。

また、環境省が有害液体物質流出事故を想定して海面・大気拡散予測モデルを設定し解析した「平成 14 年度　有害液体物質流出事故時における環境影響評価手法検討調査」を見ると、海面上での物質の挙動には「拡散」「揮発」「溶解」「水平移動」を考慮しているのみである。

そこで、本研究においても NRDAM/CME モデルより①拡散、②蒸発、③溶解，混入、④水平移動の 4 項目を選択してモデル化する。

以下、各項目について示す。

①拡散

海表面での有害物質の拡散は、Mackay et al.（1980）が経験式に基づいて修正

した Fay（1971）及び Hoult（1972）の重力粘性式によって計算する。なお、海表面の拡散面積の変化率は次式で表現する。

$$\frac{dA}{dt} = K_1 A^{1/3} \left(\frac{V_m}{A} \right)^{4/3}$$

ここで、Aは拡散膜の面積、K_1は拡散係数、V_mは流出量、 t は時間である。

②蒸発

スリック表面からの蒸発過程については、Mackay and Matsugu（1973）の式を採用する。

海表面における蒸発過程での質量移動係数は次式により計算する。

$$K_2 = 0.029 W^{0.78} D^{-0.11} S_c^{-0.67} ((MW + 29) / MW)^{1/2}$$

ここで、Wは風速、Dは拡散膜の直径、S_cはシュミット数（表面組度の指標値)、MWは流出有害物質の揮発成分のモル分子量である。

また、質量移動速度は次式により計算する。

$$\frac{dm}{dt} = (K_2 P_{VP} A / RT) FMW$$

ここで、mは拡散膜からの蒸発量、P_{VP}は蒸気圧、Aは拡散膜の面積、Rはガス定数、Tは温度、Fは拡散膜中の揮発成分を構成する残存部分、MWは流出有害物質の揮発成分のモル分子量である。

③溶解・混入

海表面のスリックから水中への混入の過程については、単位時間あたりの量として、Mackay et al.,（1980）に基づき、次式を採用する。

$$D_a = K_3 (W + 1)^2 / (1 + 50 \mu^{0.5} \delta st)$$

ここで、Wは風速、K_3は定数、μは粘性、δはスリックの厚さ、st は流出物質の海水間の界面張力である。

Wolff and Poels（1986）による研究結果では、海表面にスリックを形成する有害物質の場合、上式による計算結果は蒸発量に比較すると過小評価する傾向にあることが示され、以下のような有害物質の溶解度に依存する補正係数D_bが提案

されている。

$$D_b = K_4 (S/MW)^{0.2}$$

ここで、Ｓは溶解度、ＭＷはモル分子量、K_4は定数（≈100）である。

浮遊する有害物質に対して、水中への混入率は D_a，D_bで計算される。水中への混入は、飽和濃度に達するまで溶解することとする。

④水平移動

本モデルでは流失した HNS を多くの粒子の集まりで表現し、各粒子の移動は潮汐流および吹送流によって計算される。

また、水平及び鉛直方向の粒子の分散は、ランダムウォークモデルのアルゴリズム（Reed,1980）を使用している。粒子の分散速度（v_i）は次式で表される。

$$v_i = R^* \sqrt{(6D_i / \Delta t)}$$

ここで、添字ｉは座標軸ｘｙｚであり、D_iは拡散係数、Δｔは計算のタイムステップ、R^*は乱数（-1.0≦R^*≦1.0）である。

（２）大気拡散モデル

①平成 16 年度研究成果

本調査が対象とするのは HNS 流出事故対応のためのモデルであり、事故発生直後の拡散予測がターゲットであるため、非定常解析が必要となる。したがって、平成 16 年度研究において HNS の大気拡散予測に関しては、簡易性に優れたパフモデルの利用が適当と判断するに至った。

以下に、パフモデル基礎式を示す。

パフモデルの式系は次式のとおりである。本モデルは次々に放出されるパフを計算し、重ね合わせ濃度を積分することで濃度分布を求める手法である。

$$C = \frac{q'}{(2\pi)^{3/2}\sigma_x\sigma_y\sigma_z}\exp\left(-\frac{(x-x_0)^2}{2\sigma_x^2}-\frac{(y-y_0)^2}{2\sigma_y^2}\right)f_z$$

ここで、C は濃度、q'は排出物質量、σ_x, σ_y, σ_z はそれぞれ x,y,z 方向の拡散幅、x,y は排出源からの座標値であり、x_0,y_0 はパフの中心位置である。

また、f_Z は鉛直方向の濃度分布を表す関数で、地表面完全反射条件では次式となる。

$$f_z = \exp\left(-\frac{(H_e-z)^2}{2\sigma_z^2}\right)+\exp\left(-\frac{(H_e+z)^2}{2\sigma_z^2}\right)$$

また、混合層上端のリッドや接地逆転層による上空への拡散の停止が起こる場合、それらは上空にできる蓋としてモデル化されている。この場合、鉛直方向の濃度分布は次式で表される。

$$f_z = \sum_{n=-\infty}^{\infty}\left[\exp\left(-\frac{(z-H_e+2nh)^2}{2\sigma_z^2}\right)+\exp\left(-\frac{(z+H_e+2nh)^2}{2\sigma_z^2}\right)\right]$$

ここで、z は高度、H_e は排出源高さ、h はリッド高度、n は反射回数である。

②大気拡散モデルの再検討

パフモデルは、上記①で示した通り簡易性に優れており、ある場所から発生した物質の濃度計算において計算時間が少なくなる事から、多くの事例で利用されているモデルである。

この事から、パフモデルの利用を前提としてシステム構築のための基本設計を実施した。しかし、HNS には、流出後、短時間で蒸発等により海表面から消失する物質の他、ゆっくりと蒸発しながら長時間海表面を漂う物質も存在する。

そのため、この様な様々な物質の大気拡散予測を速やかに実行するシステムが必要となる。特に、長時間海表面を漂う物質の場合は、大気への蒸発地点が逐次変化するため多くの格子点を同時に計算する必要がある。この場合、パフモデルでは計算量が非常に多くなり、モデルの有する利点を失う事となる。

この事は、松梨ら（1993）が取りまとめている表（表 2.1-4）にも示されており、多くの格子点を同時に計算する場合は、計算格子を設定して拡散方程式を解く「格子モデル」が優れている事となる。また、表に記載されている事の他にも、格子モデルの長所として質量収支が保存される事も挙げられる。

表 2.1-4 モデルの特徴

	流跡線モデル※	格子モデル
長　所	特定地点での濃度計算労力が少なくてすむ。 上記理由もあり、複雑な化学反応、変質を考慮できる。	三次元分布をもつ風向・風速,拡散係数などのパラメータを考慮できる。 多くの格子点での濃度が同時に計算できる。
短　所	上層・下層で風速・風向が異なるときなどに追随できない。 多くの地点での同時濃度計算は計算量が増える。	疑似拡散が生じることがある。 格子間隔より小さなスケールの濃度変化を表現できない。

（松梨順三郎編著(1993)：環境流体汚染,森北出版株式会社,p319,表 3.5.5 を引用）

※ 松梨らが示す「流跡線モデル」にパフモデルが該当する。

以上の様に、大気拡散モデルのみに着目した場合は、簡易性そして計算時間の面で優れたパフモデルの利用が良いと判断されたが、海面を漂う HNS にも注目して検討すると、格子モデルの利用が妥当と考えるに至った。

次に本研究で利用する格子モデルの基礎方程式を示す。

③格子モデルの基礎方程式

格子モデルにおける基礎方程式は以下の通りである。

$$\frac{\partial C}{\partial t}+U\frac{\partial C}{\partial x}+V\frac{\partial C}{\partial y}+W\frac{\partial C}{\partial z}=\frac{\partial}{\partial x}\left(K_x\frac{\partial C}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_y\frac{\partial C}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_z\frac{\partial C}{\partial z}\right)+C0$$

ここで，

t	：時間
x，y，z	：座標軸（x,y：水平，z：鉛直）
U，V，W	：各座標軸方向の風速成分（W=0）
K_x,K_y,K_z	：各座標軸方向の HNS に対する渦動拡散係数
C	：HNS の濃度
C0	：海面の蒸発による単位時間当たりの付加濃度

（３）主な計算条件

海面拡散，大気拡散モデルによる計算を実施するための主な計算条件として、対象とするＨＮＳの物性に係わる値、そして大気拡散計算で結果に影響を及ぼす拡散係数などが挙げられる。

①海面拡散モデルの係数設定値

海面拡散モデルに用いている物性に係わる係数値等は、(社) 日本海難防止協会が平成 15 年度にとりまとめた「平成 15 年度　危険物の海上輸送時の事故対応策の研究報告書」のデータを利用する。また、報告書中にデータが存在しない係数については、参考とした「NRDAM/CME」モデルの設定値を利用する。

表 2.1-5 に海面拡散モデルに用いられている係数値等を示す。

表 2.1-5(1) 海面拡散モデルに用いられている係数値（その１）

記号	説　　明	単　位	設 定 値
K_1	海面拡散に係わる定数	/day	5×10^8
Sc	シュミット数	－	2.7
R	ガス定数	atm-m³/mole-°K	8.26×10^{-5}
K_3	混入に係わる定数	－	0.11
K_4	混入に係わる定数	－	100
Dz	鉛直方向の拡散係数	㎡/sec	0.0001
MW	キシレンのモル分子量	g/mol	106.170
	ベンゼンのモル分子量		78.120
	スチレンのモル分子量		104.150
	エタノールのモル分子量		46.070
	メチルエチルケトンのモル分子量		72.107
P_{VP}	キシレンの蒸気圧	atm (25℃)	0.00912
	ベンゼンの蒸気圧		0.14757
	スチレンの蒸気圧		0.00868
	エタノールの蒸気圧		0.08035
	メチルエチルケトンの蒸気圧		0.11539

表 2.1-5(2) 海面拡散モデルに用いられている係数値（その２）

記号	説　　　明	単　位	設 定 値
μ	キシレンの粘性	cP	1.128
	ベンゼンの粘性		0.6022
	スチレンの粘性		0.7033
	エタノールの粘性		0.4709
	メチルエチルケトンの粘性		0.4766
S	キシレンの溶解度	mg/l	115.5
	ベンゼンの溶解度		820.0
	スチレンの溶解度		300.0
	エタノールの溶解度		437100.0
	メチルエチルケトンの溶解度		181900.0

②大気拡散モデルの主な計算条件

格子モデルを利用しての予測計算のための主な計算条件を表 2.1-6 に示す。

表 2.1-6 大気拡散モデルの主な計算条件

項　目	設　　定	
計算格子	水平	1000m 格子
	鉛直	可変格子（７層）
渦動拡散係数		100 ㎡/s
風条件の取り扱い	水平	全層一様（時間変化考慮）
	鉛直	0m/s

２． ２ モデルの試作

本年度事業では本モデルの心臓部となる、海面拡散や大気拡散を予測するための計算プログラムの構築を実施する。そのため、次に示す様に計算対象日時を設定し、その時に流出した各物質の拡散予測計算を行う。そして、得られた結果を用いてプロトタイプモデルでの表示例を示し、本モデルの概要を述べる。

２． ２． １　流動および吹送流モデル

（１）計算対象日

潮流場や吹送流を計算する場合、流出事故発生を想定した日時を決める必要がある。

本研究で試作するプロトタイプモデルでは、解析結果例として事故発生後の海面上の拡散状況、そして蒸発後に大気中を拡散して海上から陸上へ拡がる様子を表示する。そこで、設定した事故発生地点（中の瀬航路北端部）から横浜～川崎方向へ向かう風（南東の風）が吹く日を想定する。

横浜気象台での最近の風向・風速観測結果より、南東方向からの風が数時間続く日を抽出すると 2005 年 4 月 2 日 15 時が該当し、この時の風速が 2.4m/s であった。また、潮位（東京：図 2.2-1）を見ると干潮になりつつあり、約２～３時間後からは上げ潮となる。そのため、海表面を拡散する HNS が湾奥海域へと流される事が予測される。

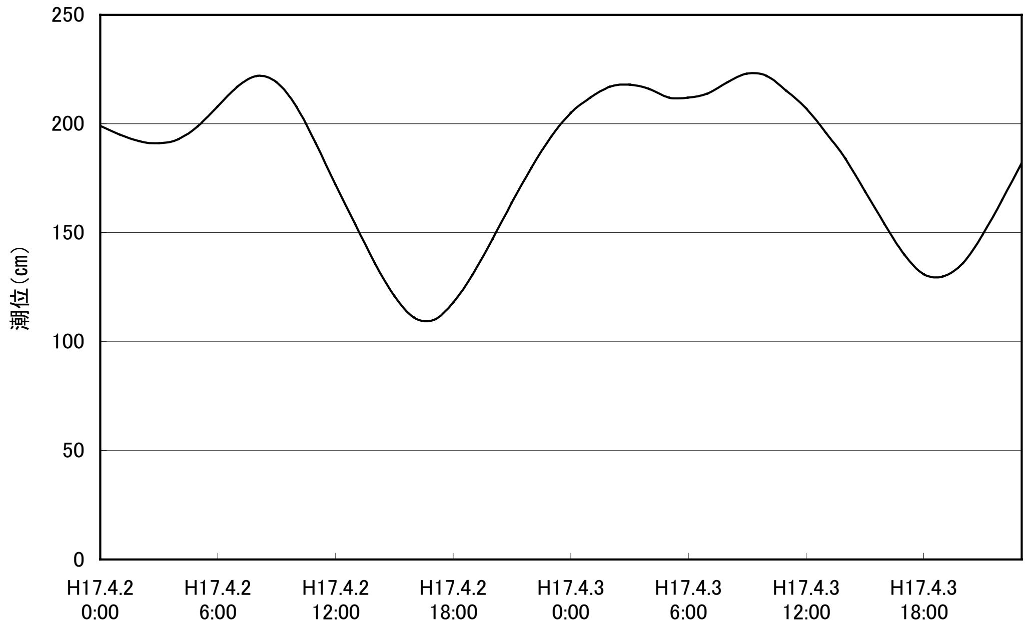


図 2.2-1 潮位変化図

（２）プロトタイプモデルの表示例

構築した流動および吹送流モデルを用いて計算した結果を、プロトタイプモデルの表示例を用いて図 2.2-2 に示す。なお、時間断面は、後で表示する海面・大気拡散計算結果に合わせて、3.5 時間後までを 0.5 時間毎に表示する。

【流動場の表示について】

①本モデルで流動場を表示する場合、『表示項目』のドロップダウンリストより「流動ベクトル（風あり）」「流動ベクトル（風なし）」の何れかを選択する。

②次に、『実行』ボタンを押す事で、事故発生日時からの流動場が表示される。（図 2.2-2 では「流動ベクトル（風なし）」の結果を表示する。）

③本年度事業では既に計算対象日時（事故発生日時）を指定しているが、実施した 4 潮時の潮流計算により調和定数のストックが可能となった。そのため、今後システムを高度化する事で任意に指定した日時からの潮流場が表示可能となる。

④また、風の条件についても、吹送流の計算は短時間で実施出来るため、任意に指定する事で風を考慮した流動場の表示も可能となる。ただし、風の指定方法については、統計値や予報値、任意に入力など幾つか考慮出来るため、個々に対応したシステムの高度化が必要となる。

⑤そして、流動場に影響する項目として残る河川水量に関しては、平水時のみを計算したが、他に出水時や豊水時、渇水時の計算も実施し、結果をストックする事で、事故発生時の状況に応じて河川水も考慮した流れ場の設定が可能となる。

〈流動ベクトルの選択〉

〈実行〉

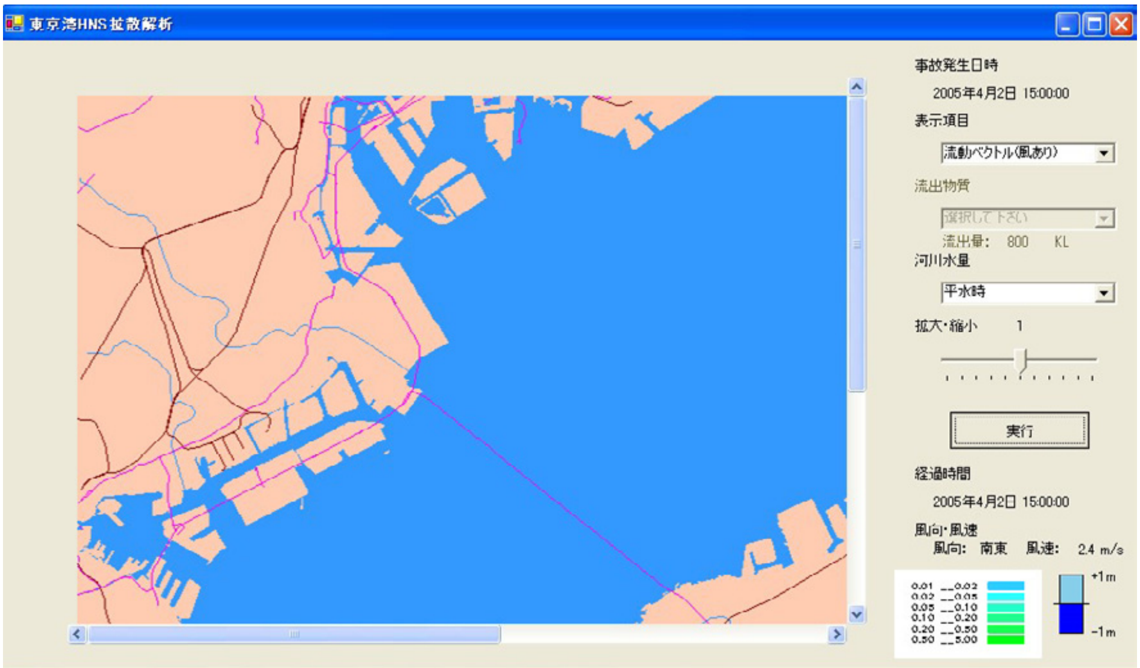
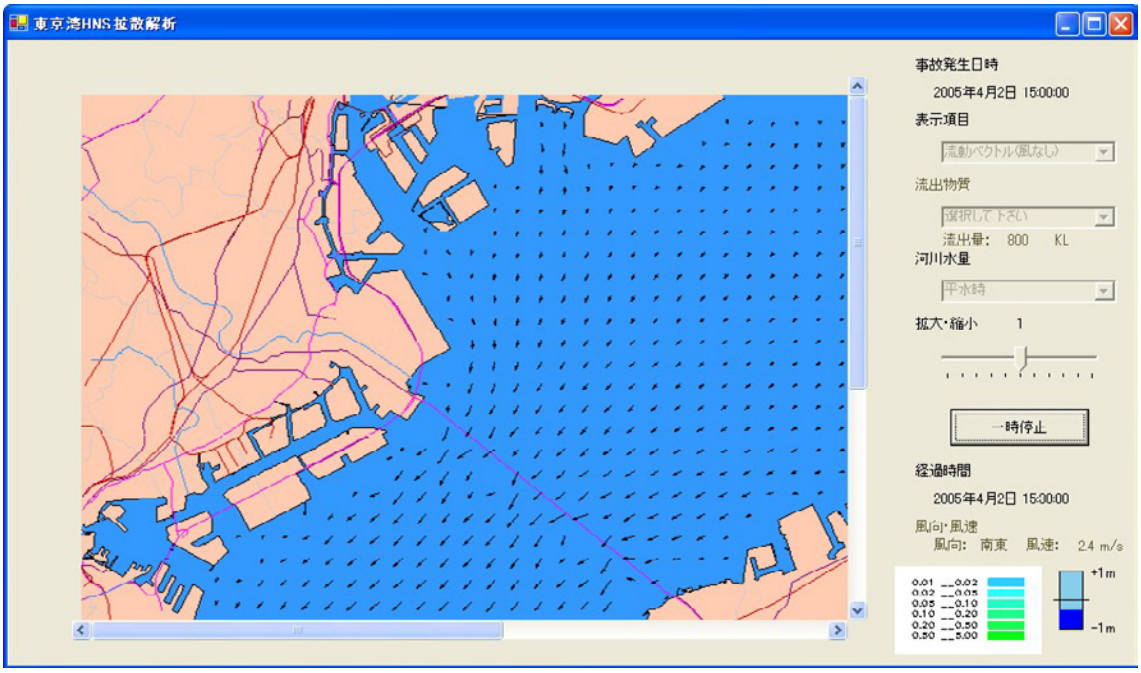


図 2.2-2(1)　流動場の表示手順

〈0.5 時間後〉



〈 1 時間後〉

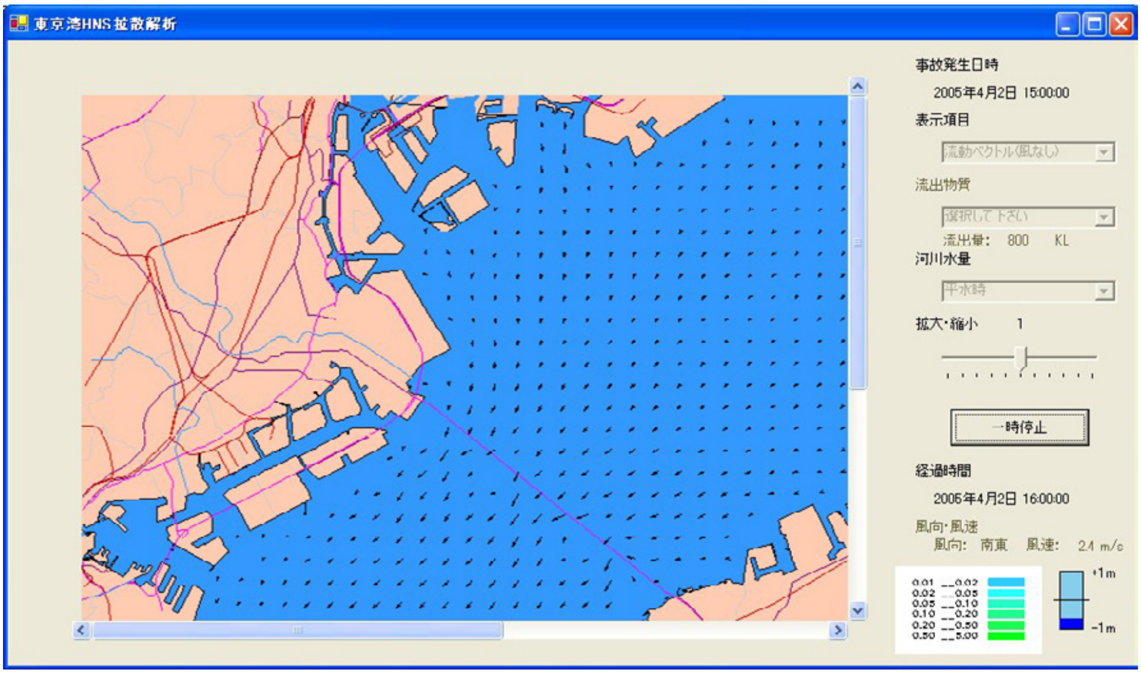
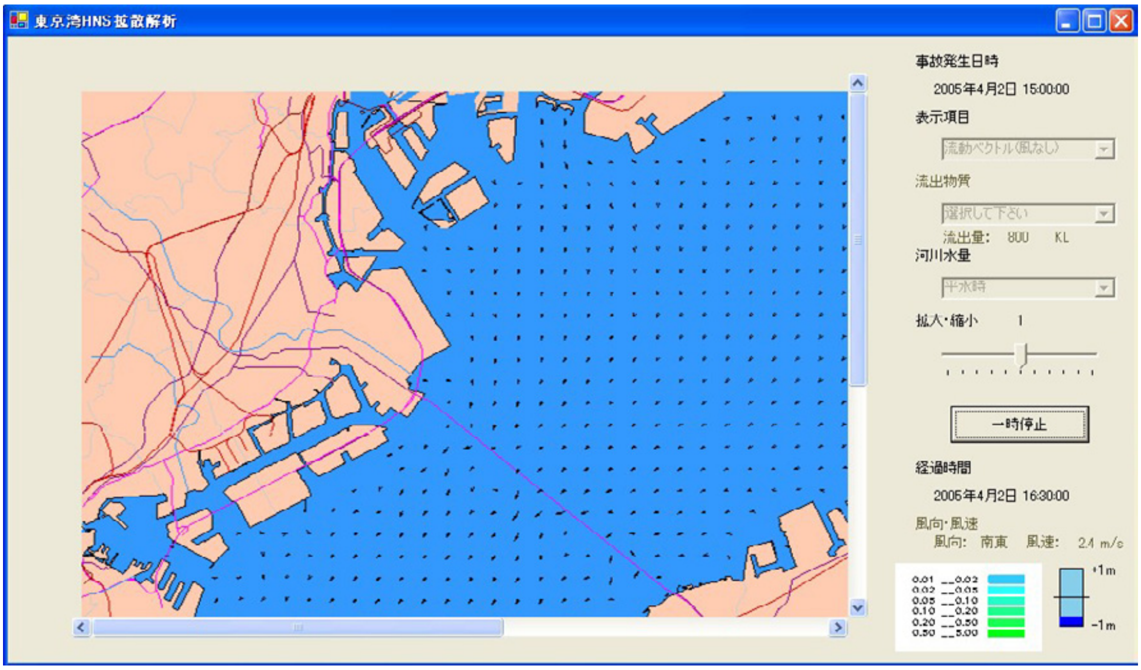


図 2.2-2(2) 流動計算結果（0.5 時間後、1 時間後）

〈1.5 時間後〉



〈２時間後〉

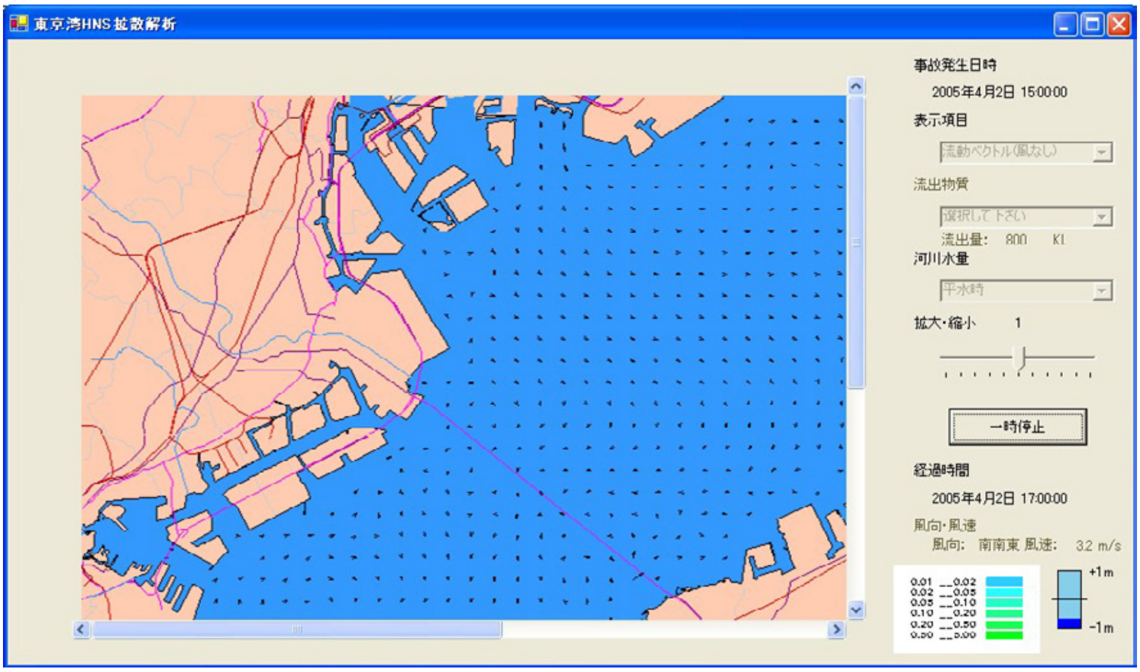
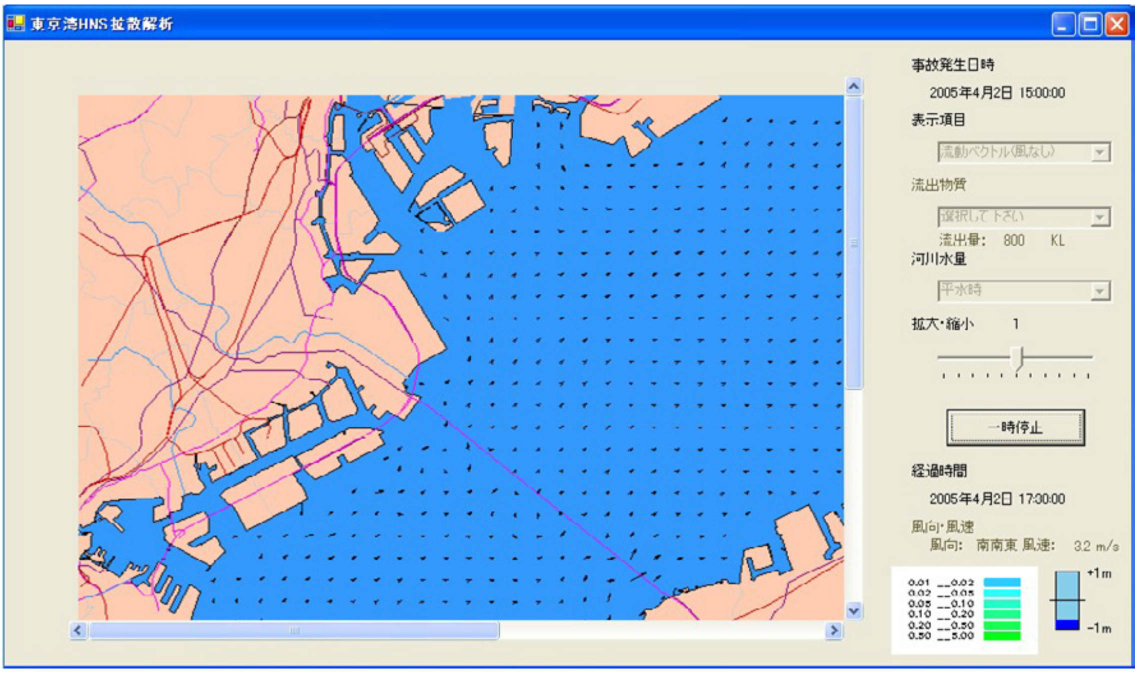


図 2.2-2(3)　流動計算結果（1.5 時間後、２時間後）

〈2.5 時間後〉



〈３時間後〉

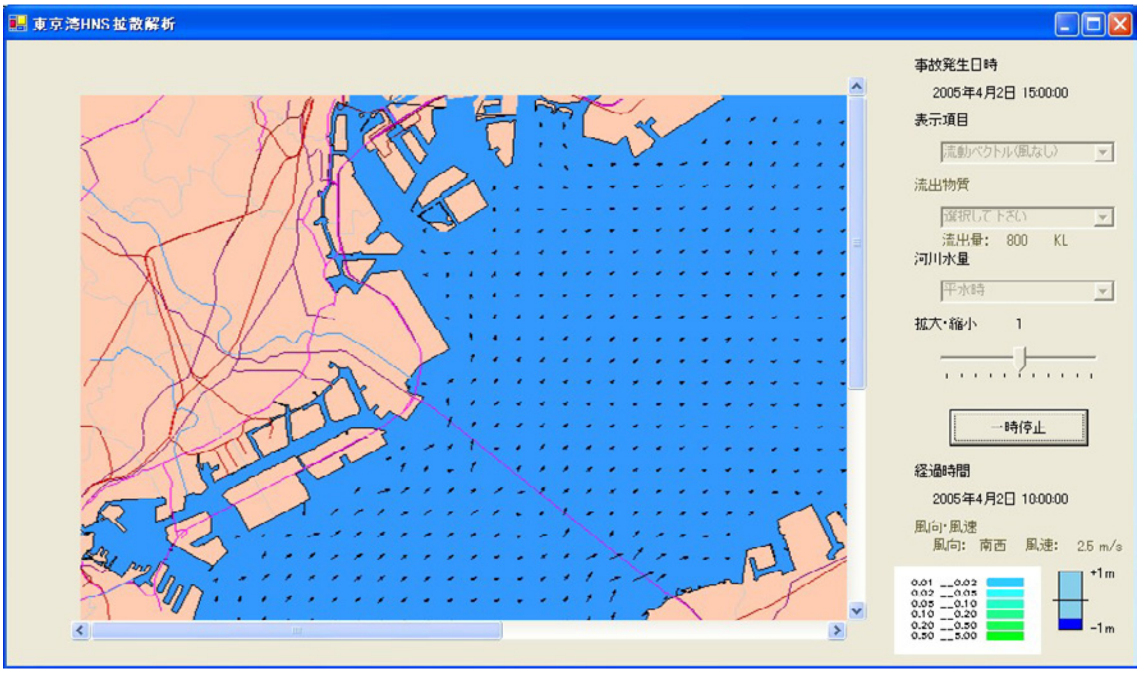


図 2.2-2(4)　流動計算結果（2.5 時間後、３時間後）

〈3.5 時間後〉

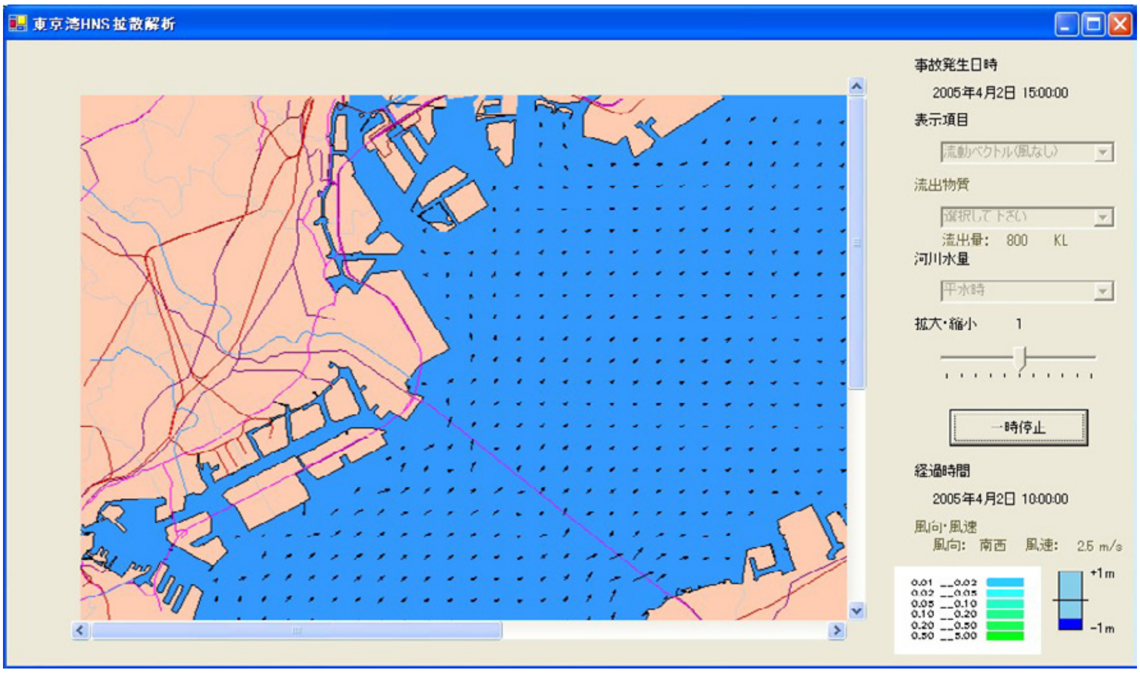


図 2.2-2(4)　流動計算結果（3.5 時間後）

２．２．２　海面・大気拡散モデル

海面拡散および大気拡散の計算は、先に風条件や河川条件を設定した流動場そして風向・風速をもとにして実施する。ここでは、対象物質としたキシレン，ベンゼン，スチレン，エタノール，メチルエチルケトンの計算結果をプロトタイプモデルの表示例を用いて図 2.2-3～図 2.2-7 に示す。

【海面・大気拡散の表示について】

①本モデルで流動場を表示する場合､『表示項目』のドロップダウンリストより「海面・大気拡散状況」を選択する。

②次に『対象物質』のドロップダウンリストより、表示する物質を選択する。

③そして『実行』ボタンを押す事で、事故発生日時からの毎時の海面拡散状況が表示される。

④本年度研究では既に流出地点及び流出量を指定しているが、今後システムを高度化する事で任意に指定した場所および流出量での拡散計算が可能となる。

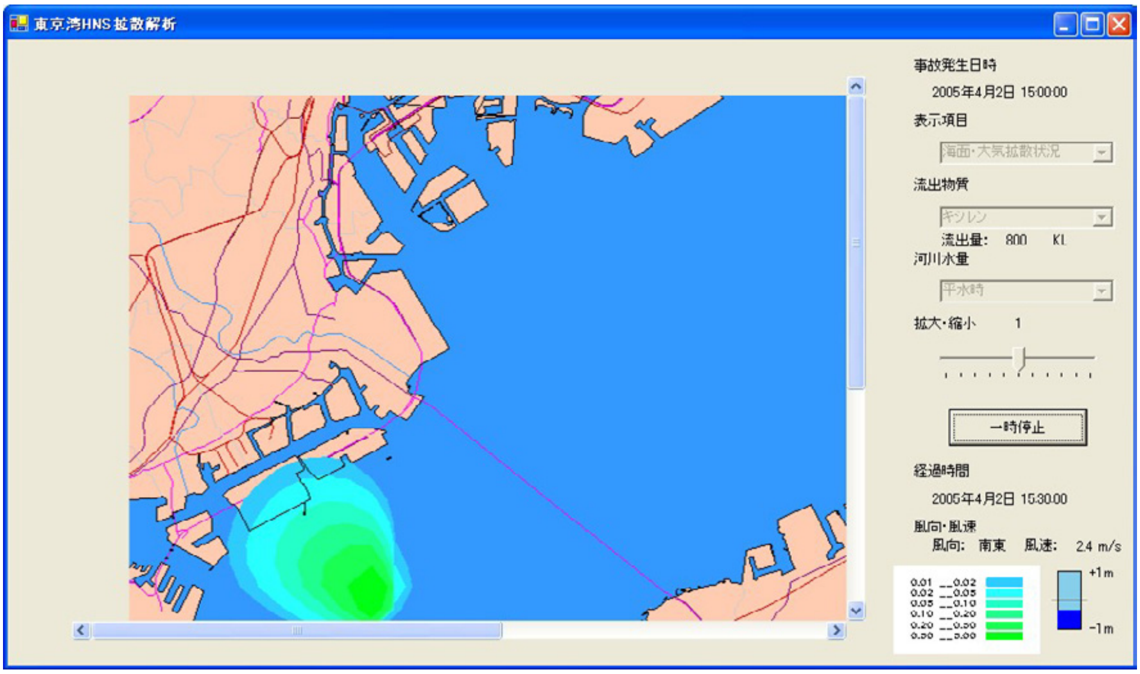
⑤また、物質の海面拡散モデルでは風速や気温そして水温を設定する必要がある。風速については先の流動モデルで設定した条件を利用出来るが、気温や水温についても統計値や予報値、任意に入力など個々に対応したシステムの構築が必要となる。

〈海面・大気拡散状況の選択〉

〈対象物質の選択〉

図 2.2-3(1)　海面大気拡散状況の表示手順

〈0.5 時間後〉



〈 1 時間後〉

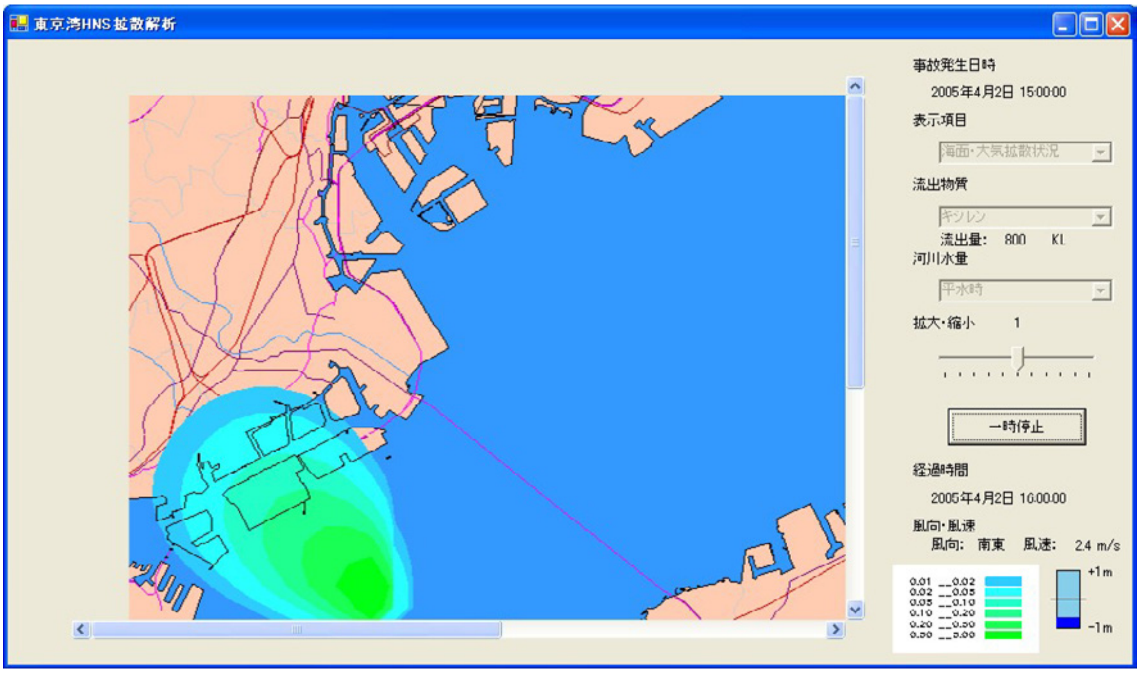
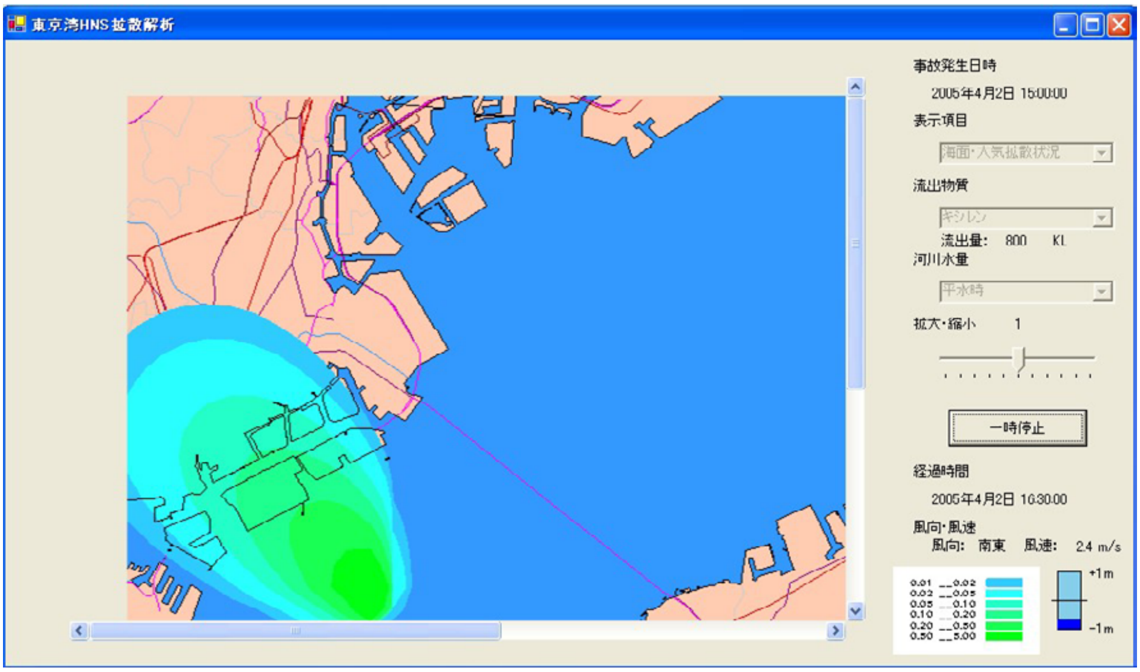


図 2.2-3(2) キシレンの海面大気拡散計算結果（0.5 時間後、１時間後）

〈1.5 時間後〉



〈２時間後〉

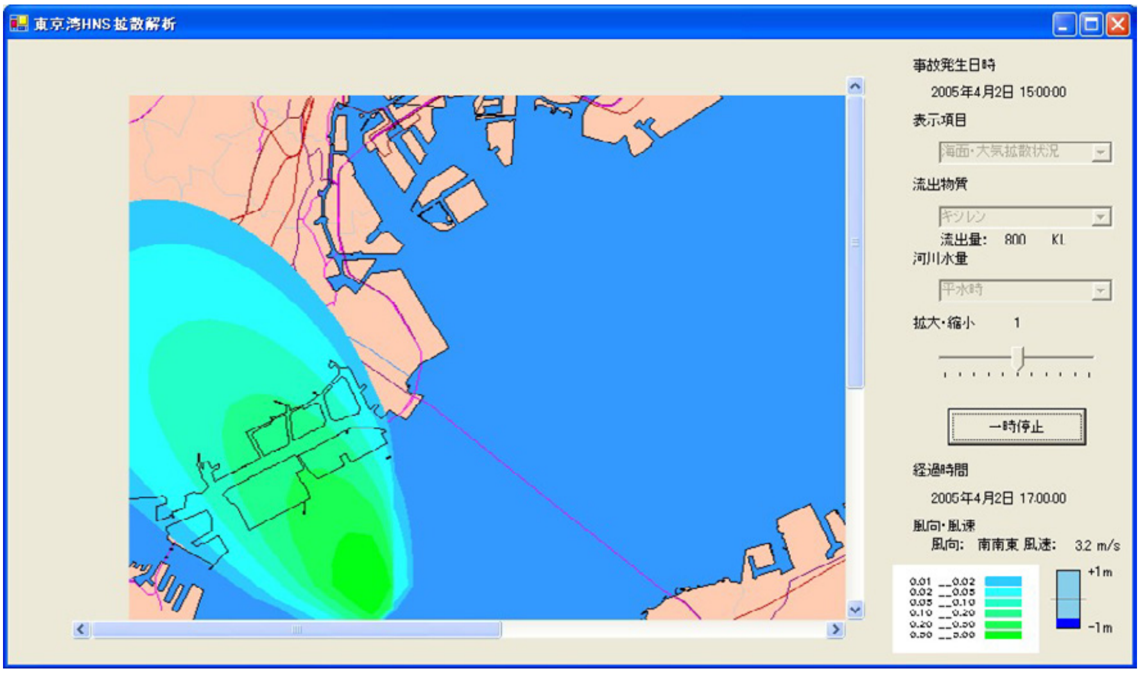
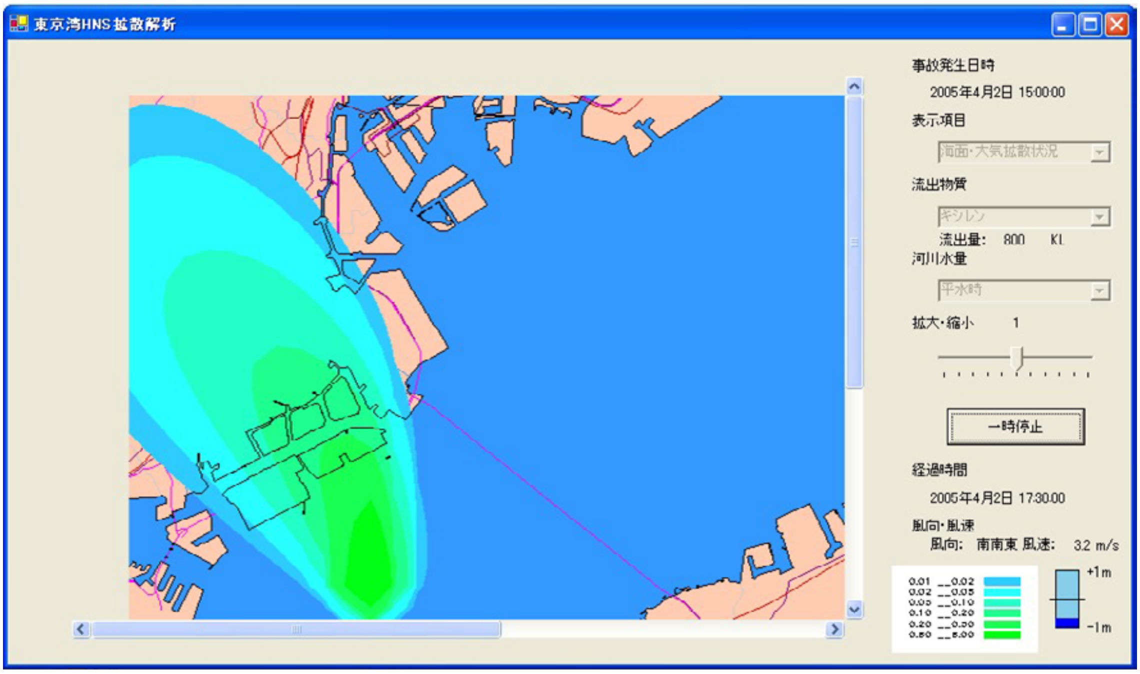


図 2.2-3(3)　キシレンの海面大気拡散計算結果（1.5 時間後、２時間後）

〈2.5 時間後〉



〈３時間後〉

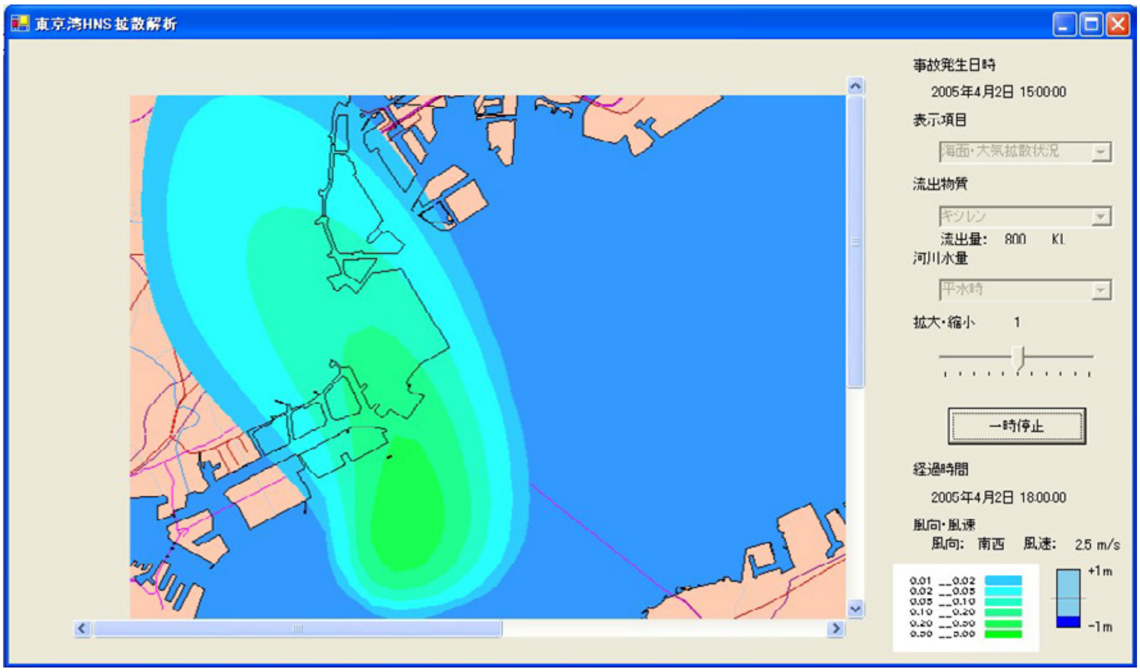


図 2.2-3(4)　キシレンの海面大気拡散計算結果（2.5 時間後、３時間後）

〈3.5 時間後〉

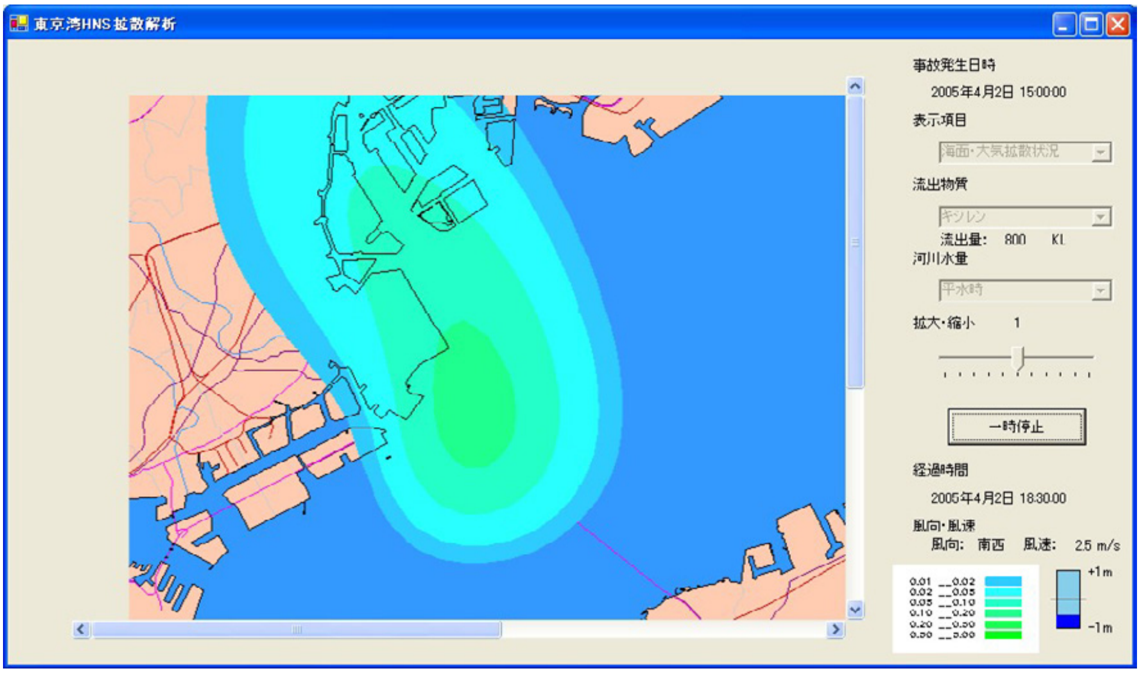
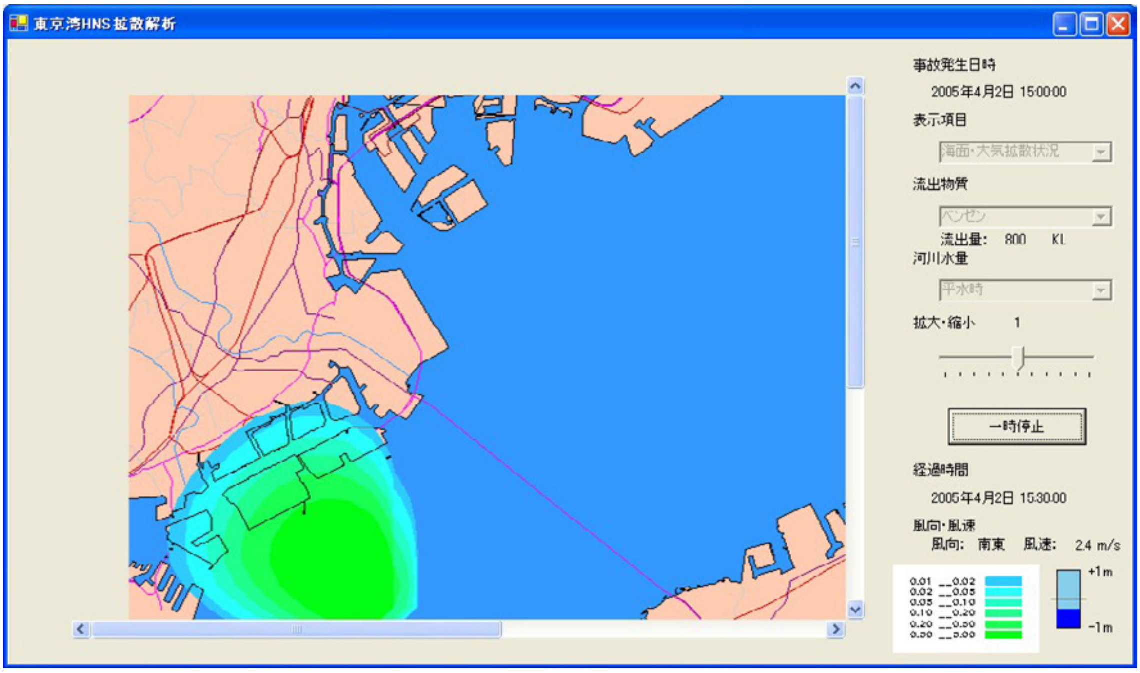


図 2.2-3(5)　キシレンの海面大気拡散計算結果（3.5 時間後）

〈0.5 時間後〉



〈1 時間後〉

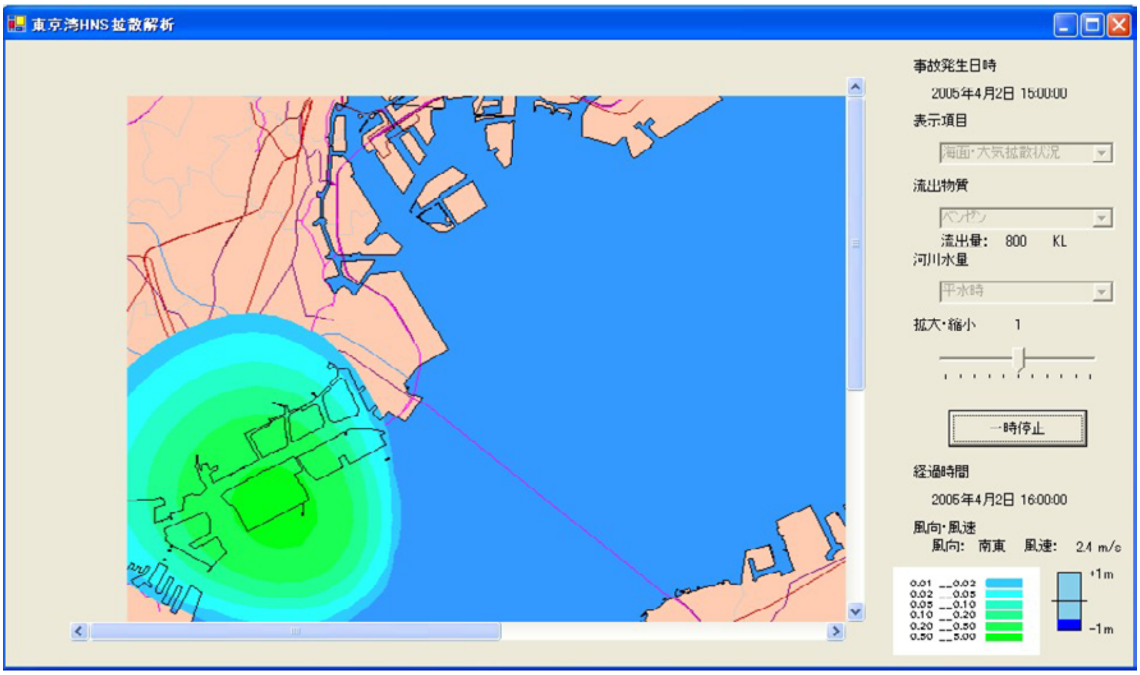
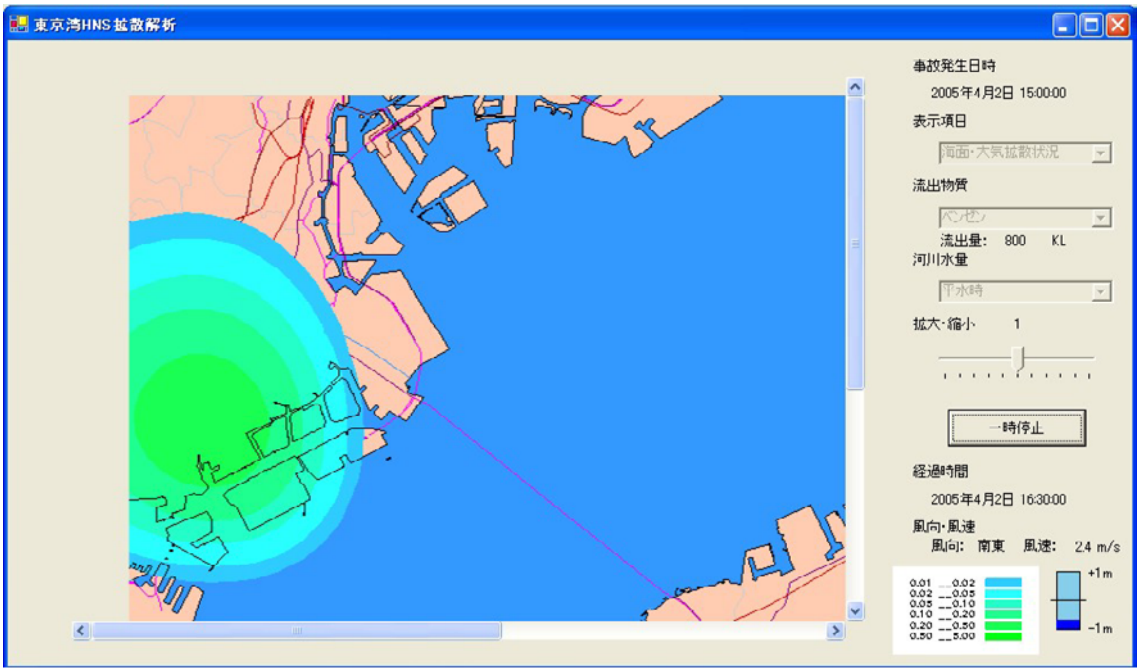


図 2.2-4(1)　ベンゼンの海面大気拡散計算結果（0.5 時間後、1 時間後）

〈1.5 時間後〉



〈２時間後〉

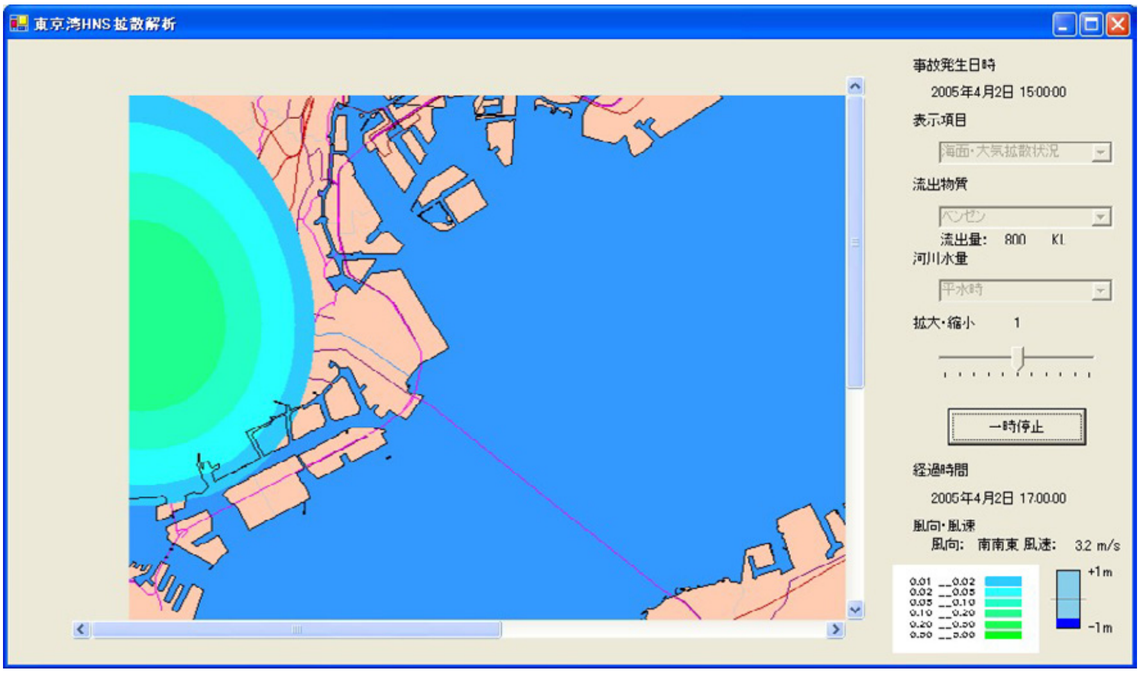
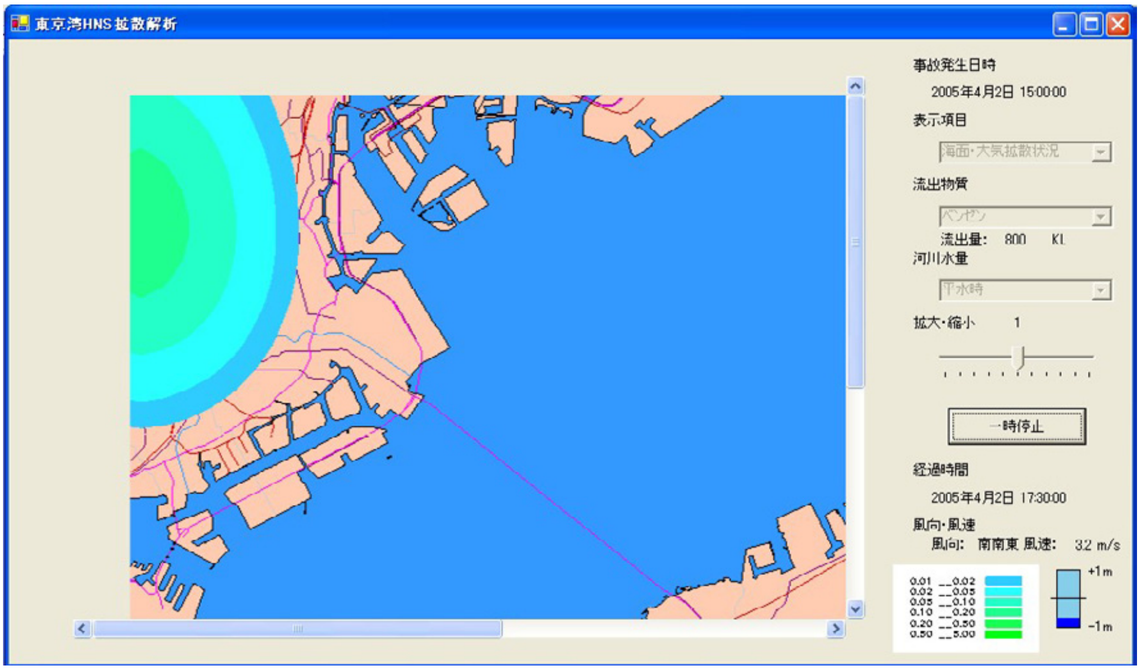


図 2.2-4(2)　ベンゼンの海面大気拡散計算結果（1.5 時間後、２時間後）

〈2.5 時間後〉



〈３時間後〉

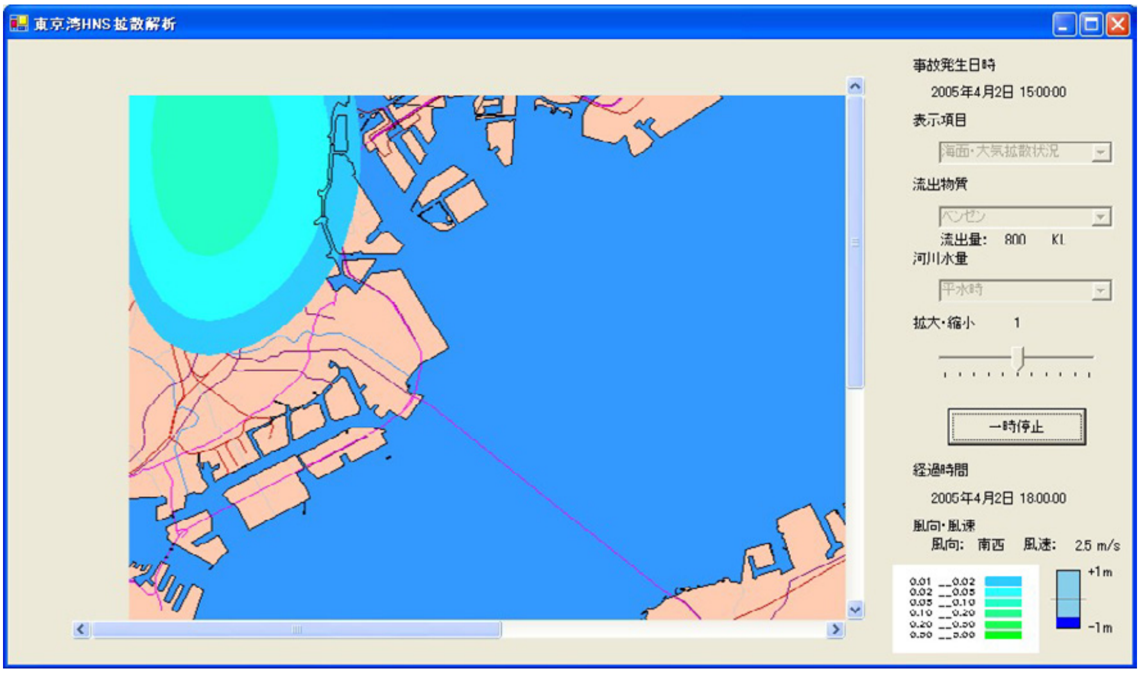


図 2.2-4(3)　ベンゼンの海面大気拡散計算結果（2.5 時間後、３時間後）

〈3.5 時間後〉

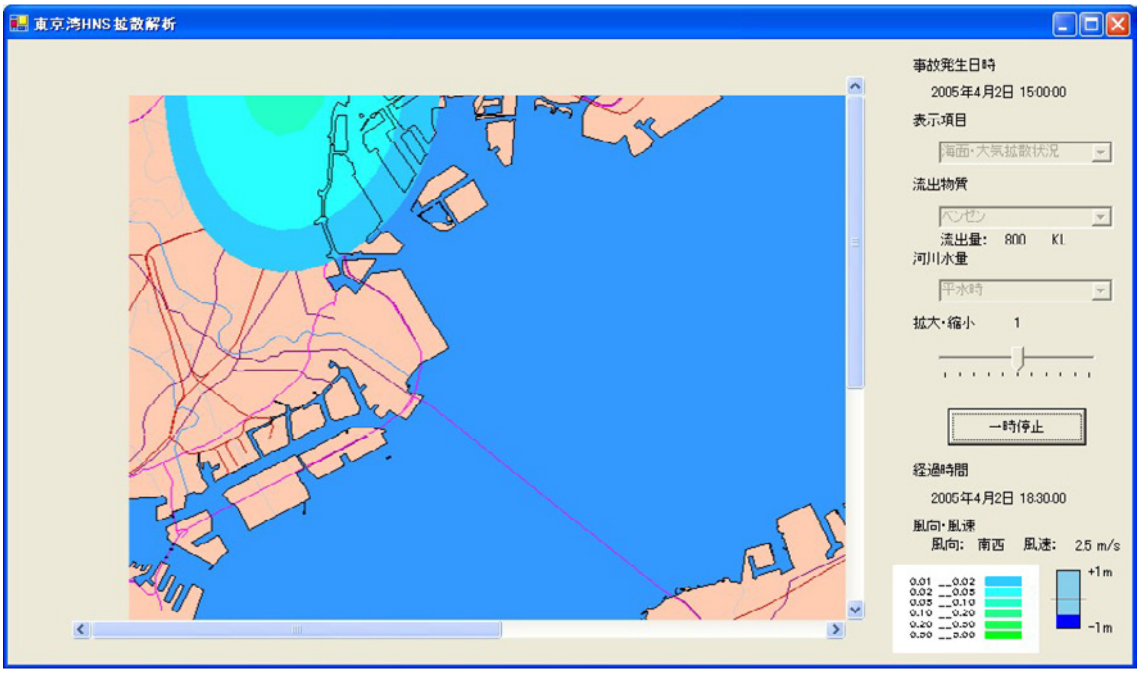
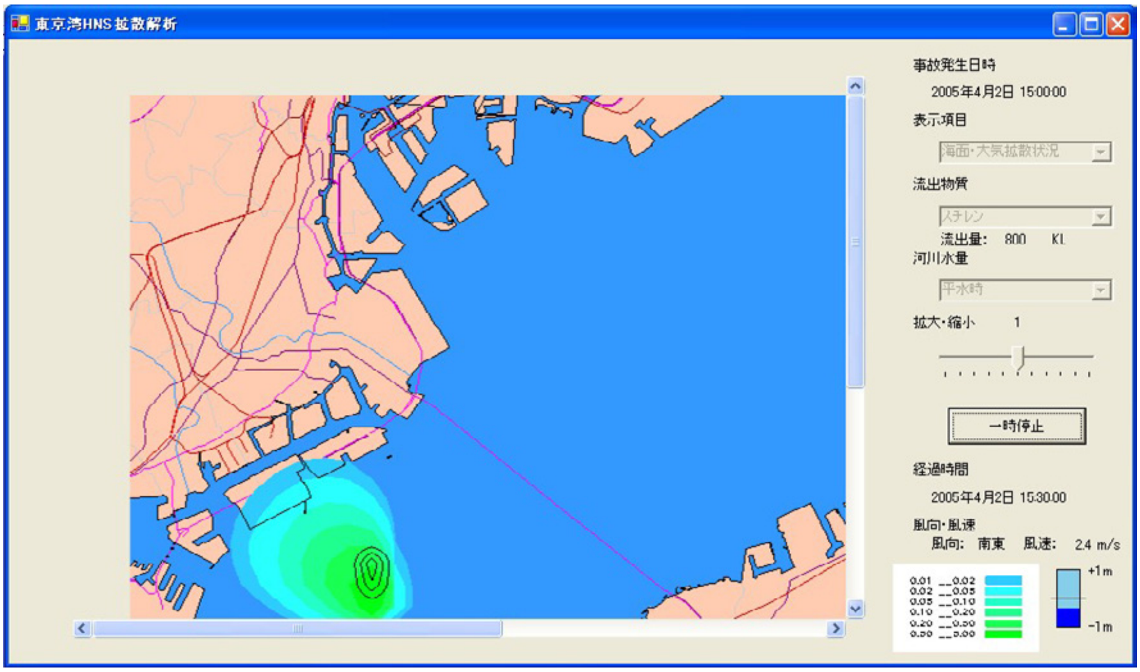


図 2.2-4(4)　ベンゼンの海面大気拡散計算結果（3.5 時間後）

〈0.5 時間後〉



〈1 時間後〉

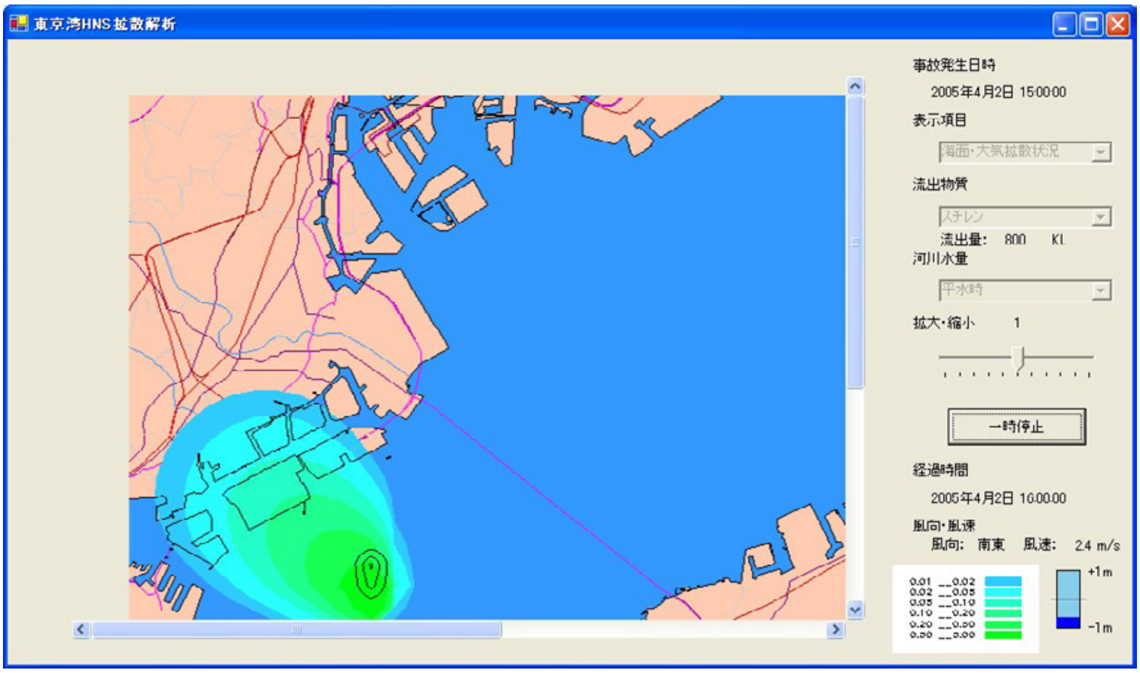
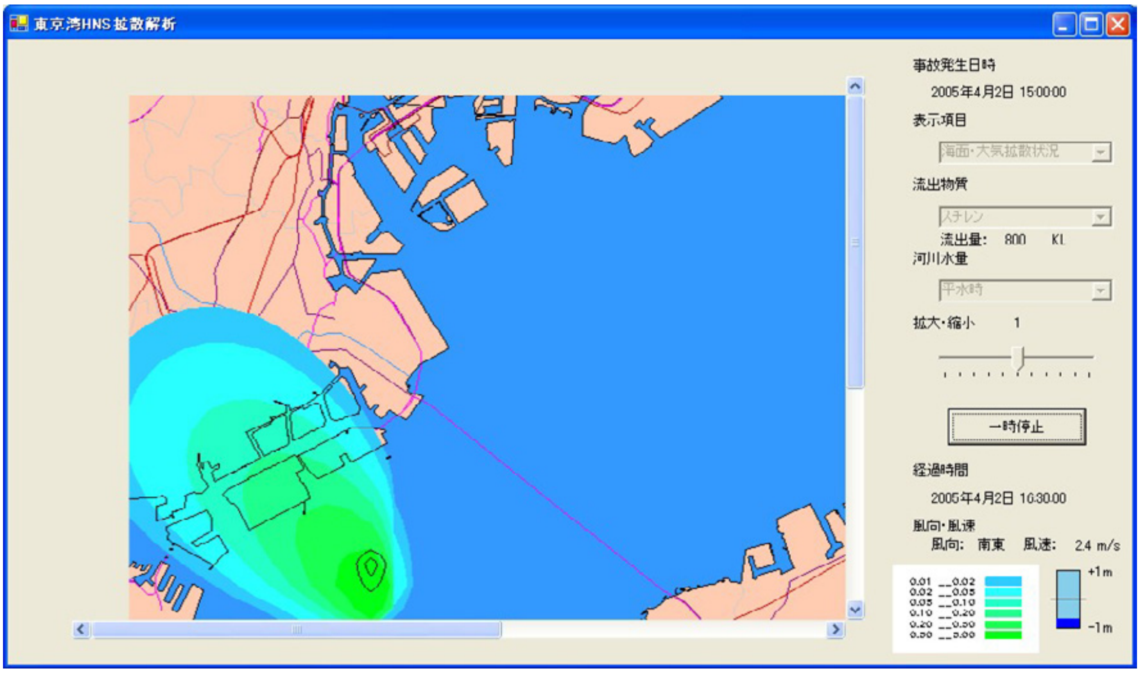


図 2.2-5(1) スチレンの海面大気拡散計算結果（0.5 時間後、1 時間後）

〈1.5 時間後〉



〈２時間後〉

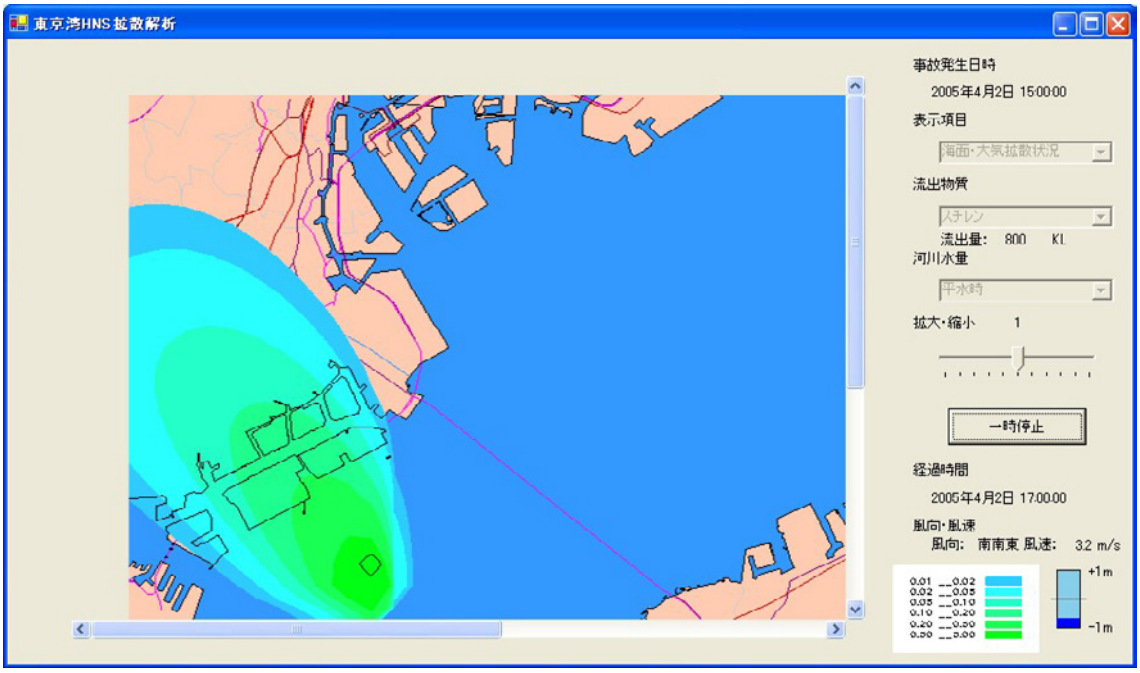
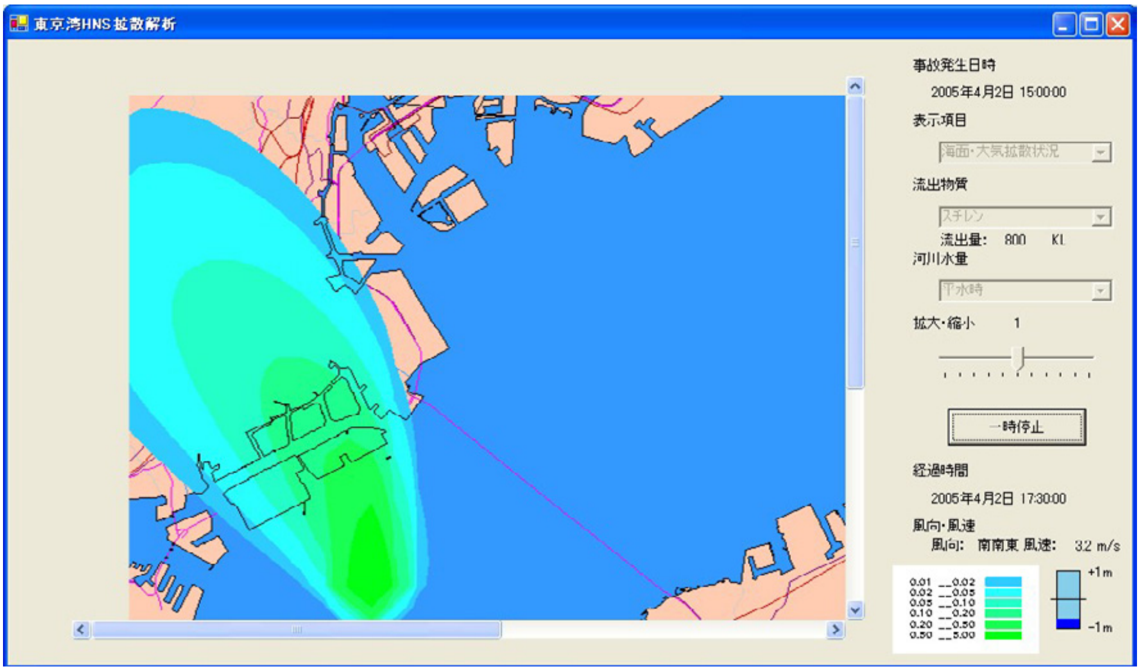


図 2.2-5(2)　スチレンの海面大気拡散計算結果（1.5 時間後、２時間後）

〈2.5 時間後〉



〈３時間後〉

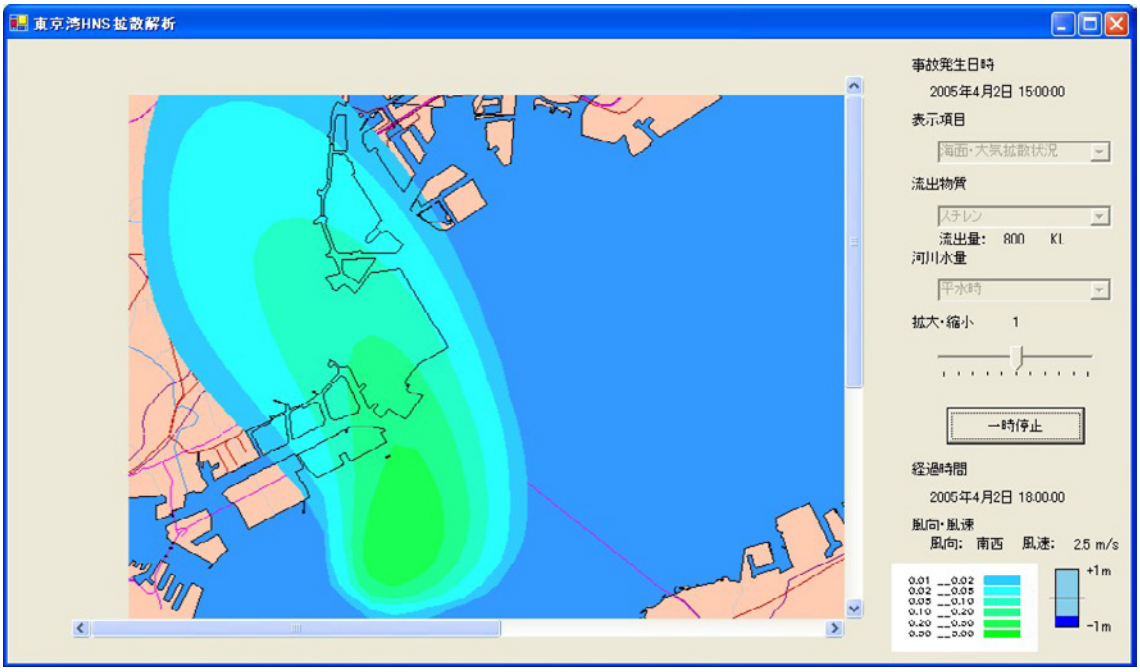


図 2.2-5(3) スチレンの海面大気拡散計算結果（2.5 時間後、３時間後）

〈3.5 時間後〉

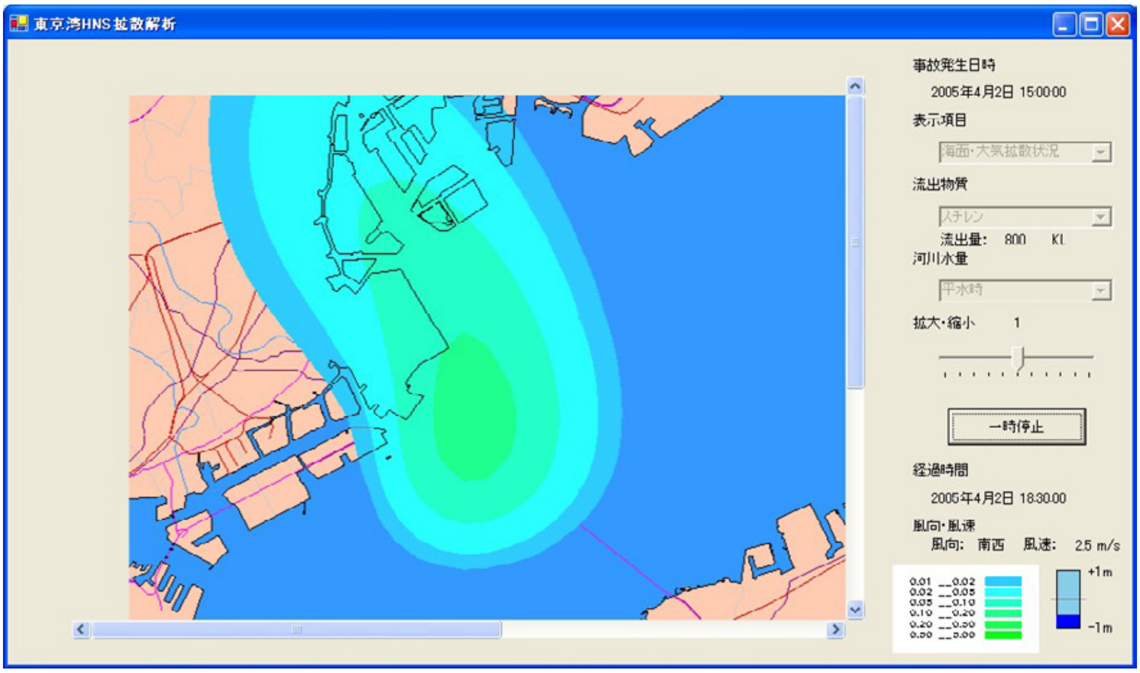
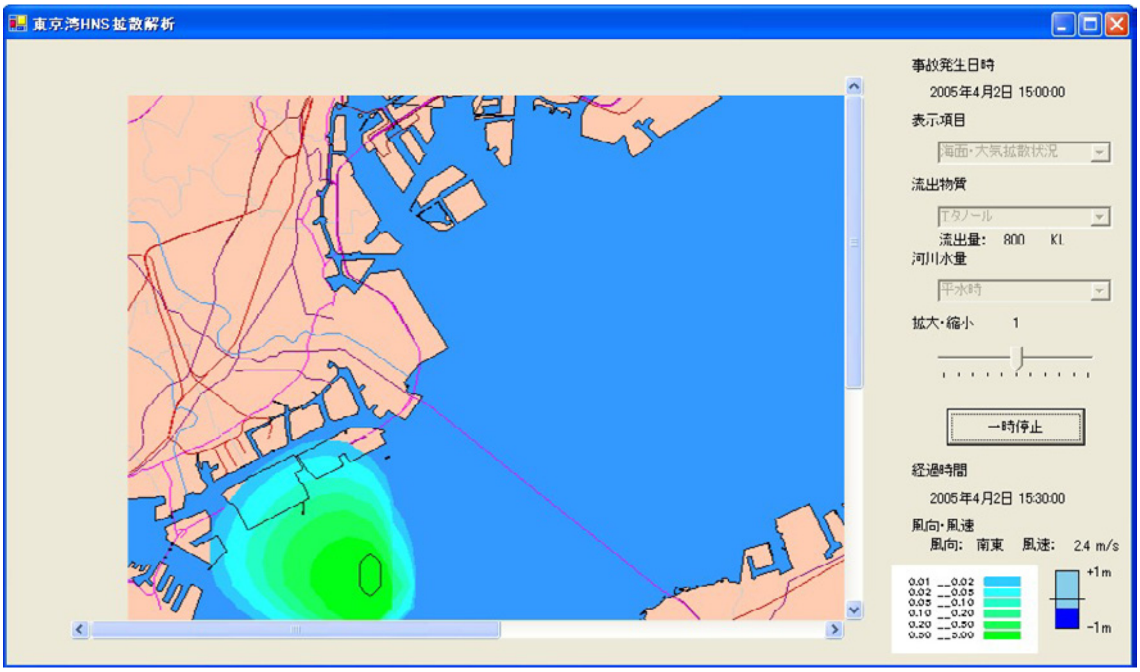


図 2.2-5(4)　スチレンの海面大気拡散計算結果（3.5 時間後）

〈0.5 時間後〉



〈1 時間後〉

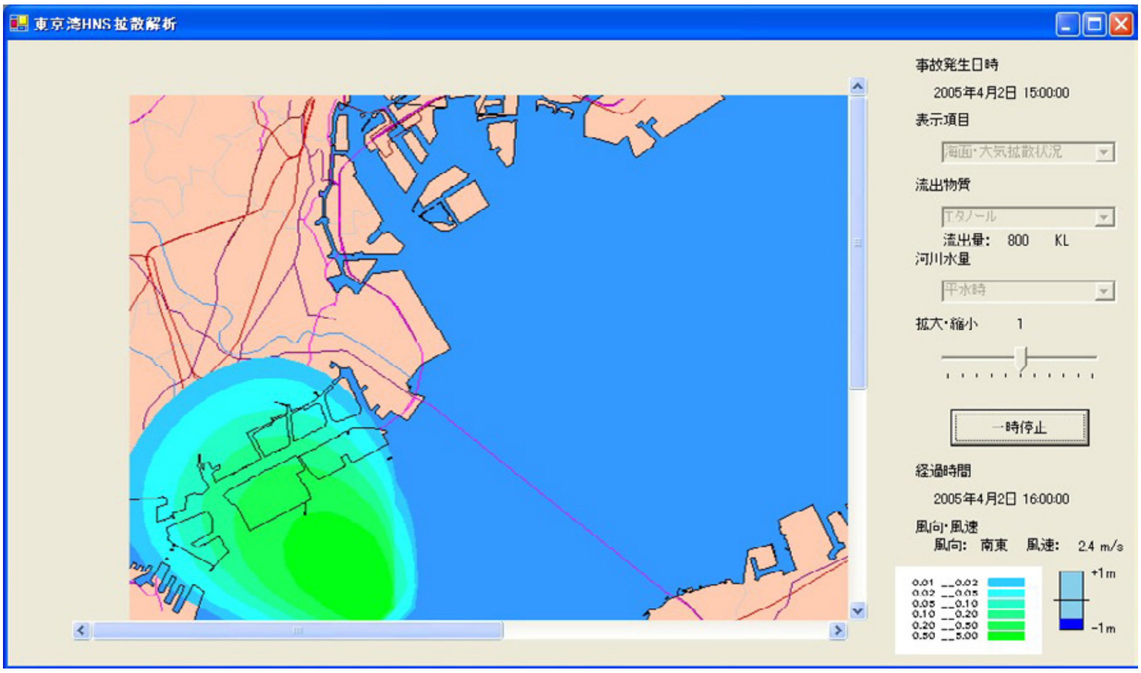
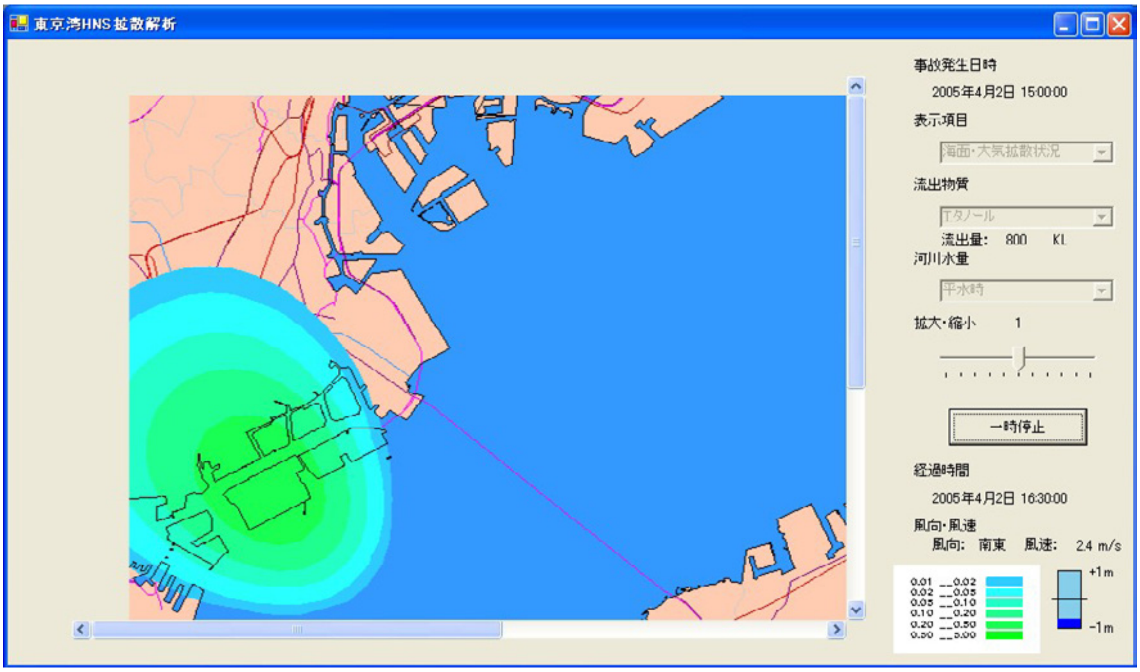


図 2.2-6(1)　エタノールの海面大気拡散計算結果（0.5 時間後、１時間後）

〈1.5 時間後〉



〈２時間後〉

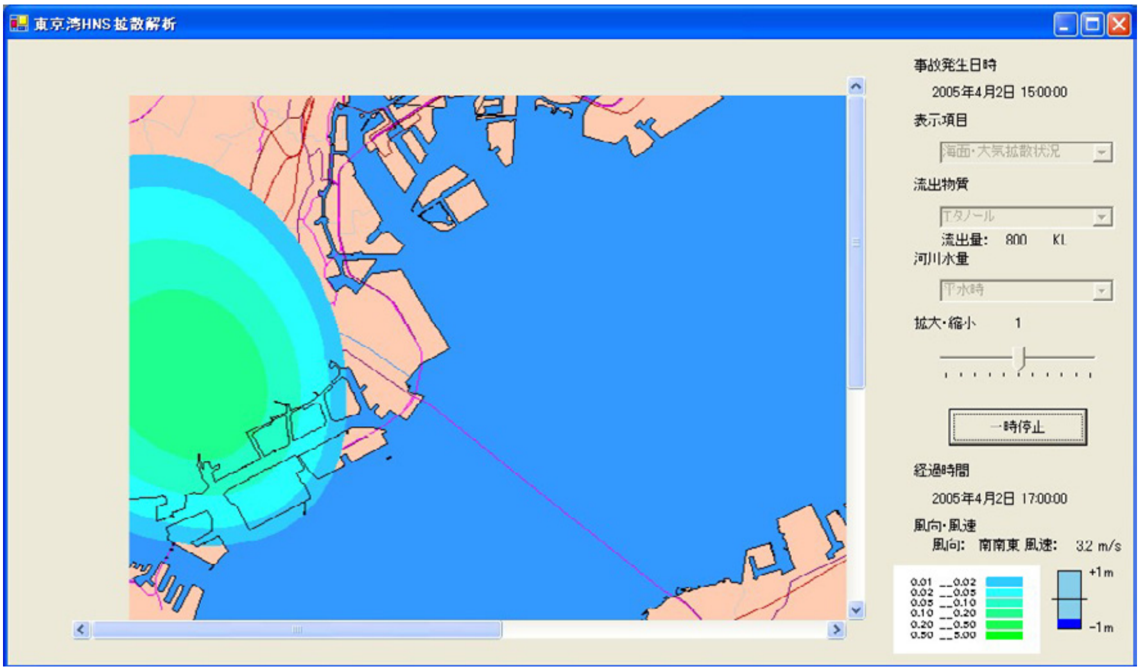
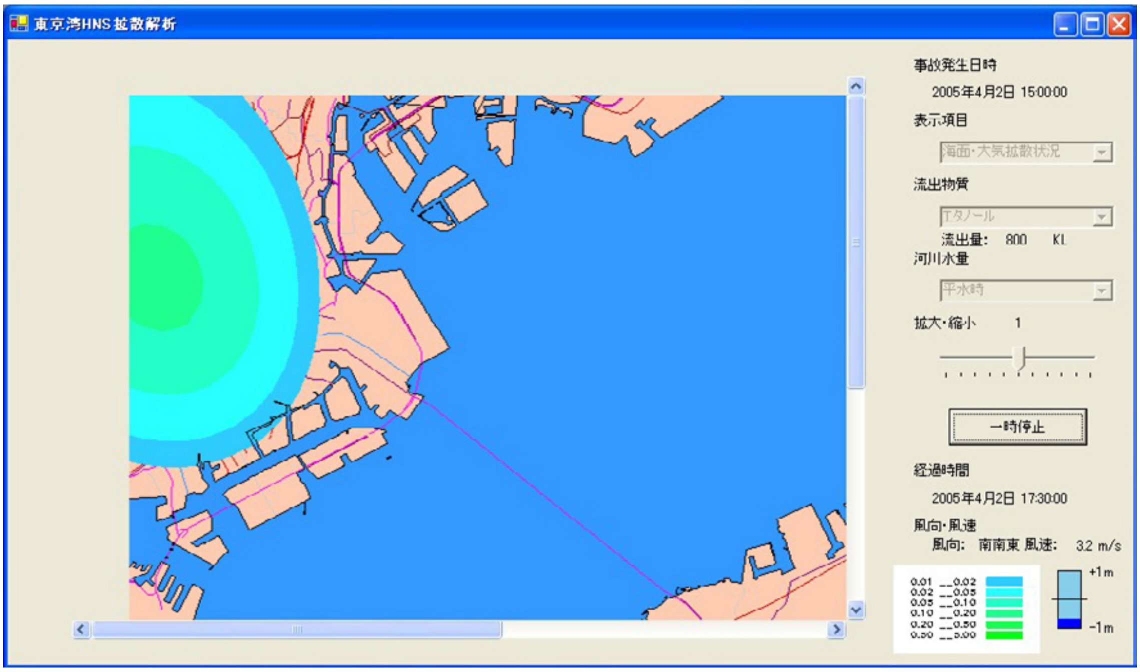


図 2.2-6(2)　エタノールの海面大気拡散計算結果（1.5 時間後、２時間後）

〈2.5 時間後〉



〈３時間後〉

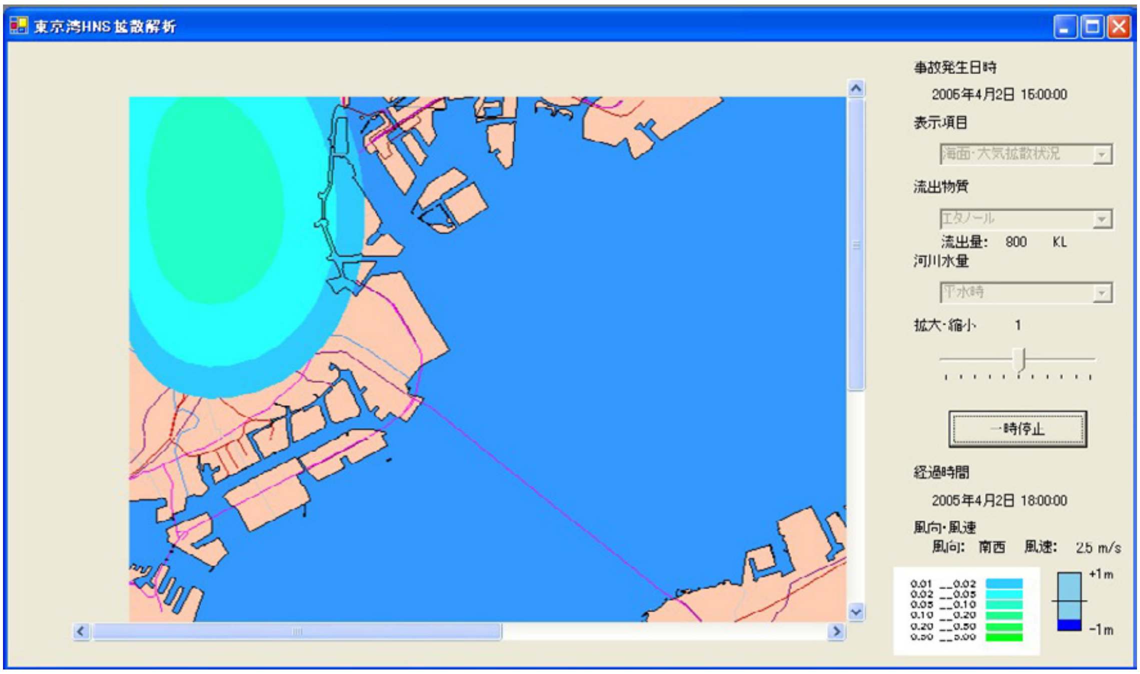


図 2.2-6(3)　エタノールの海面大気拡散計算結果（2.5 時間後、３時間後）

〈3.5 時間後〉

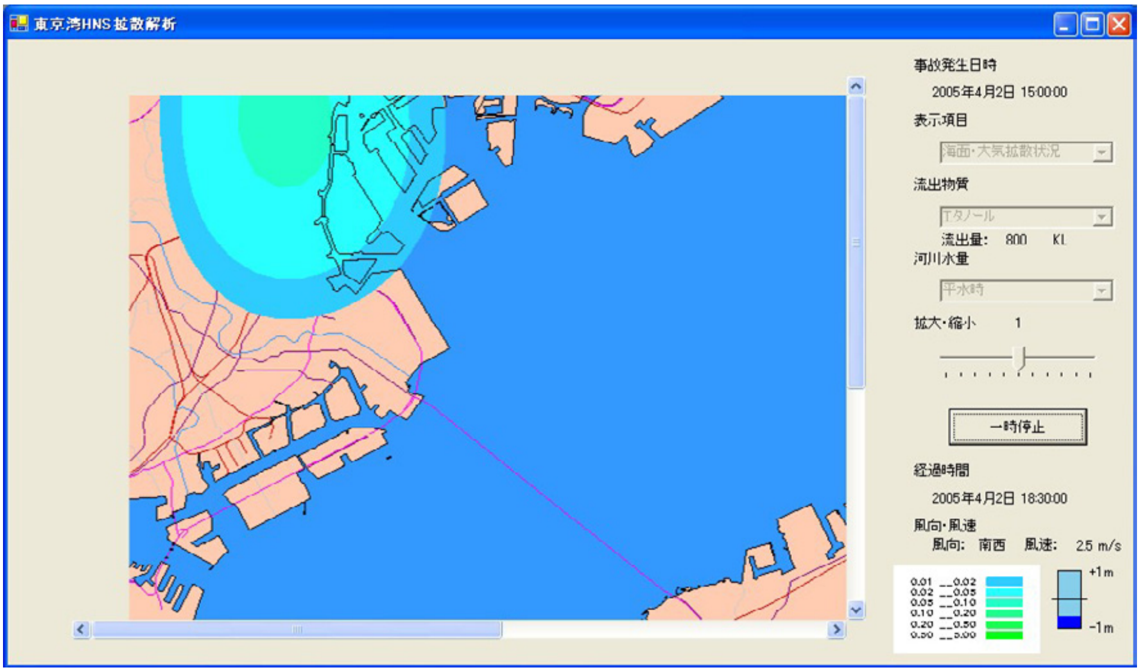
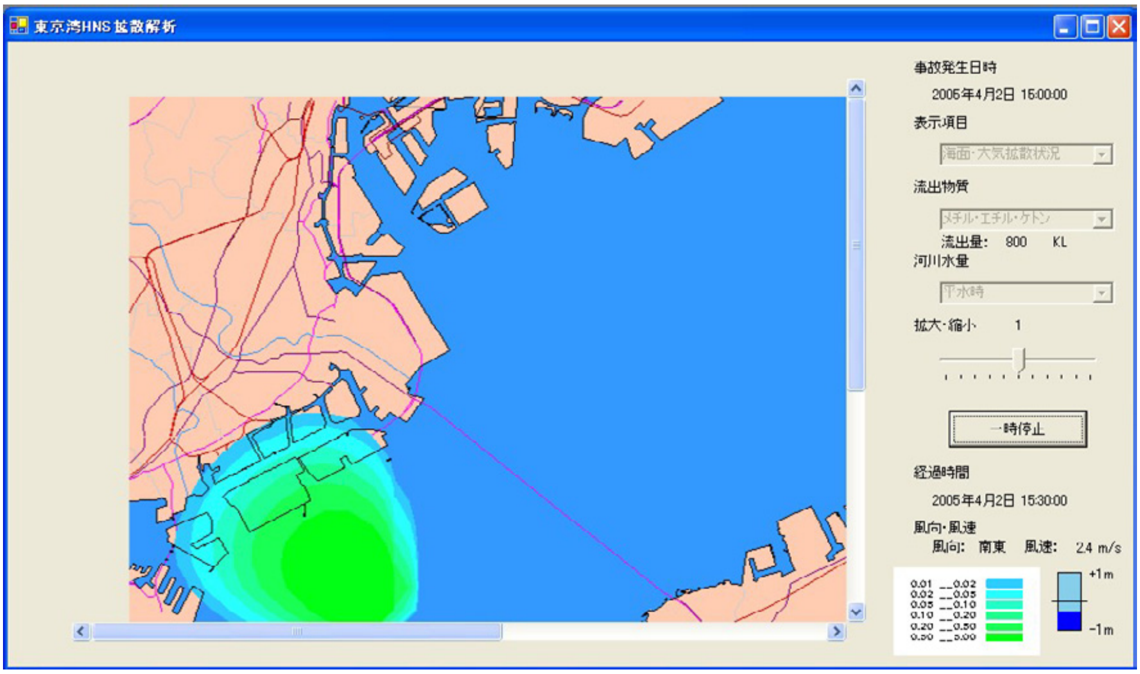


図 2.2-6(4) エタノールの海面大気拡散計算結果（3.5 時間後）

〈0.5 時間後〉



〈1 時間後〉

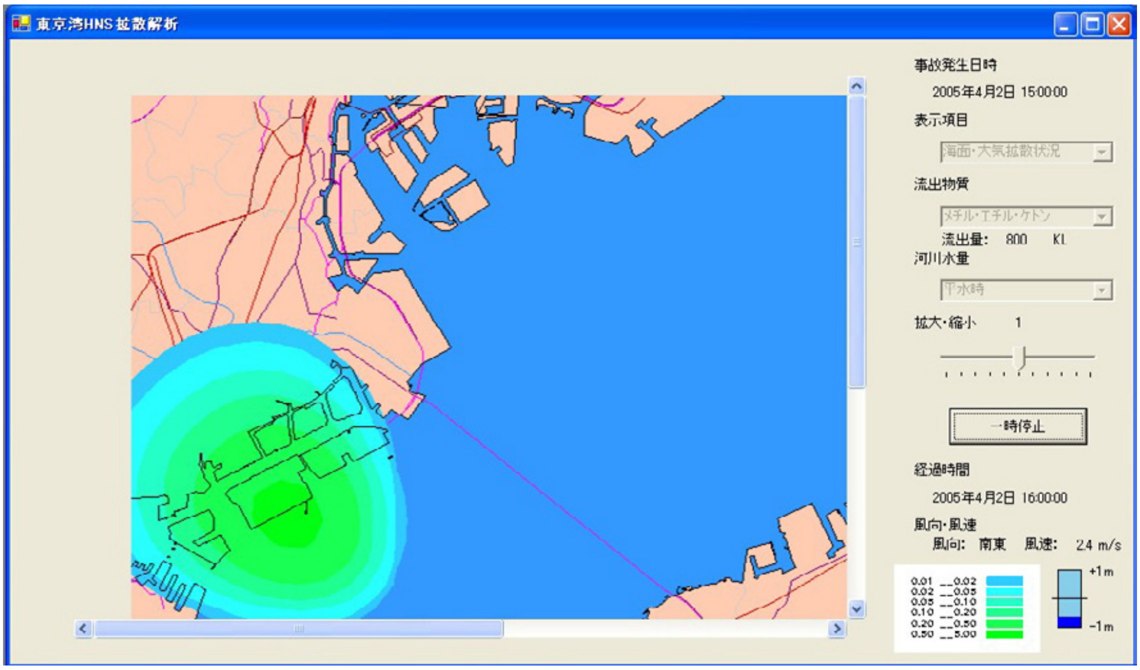
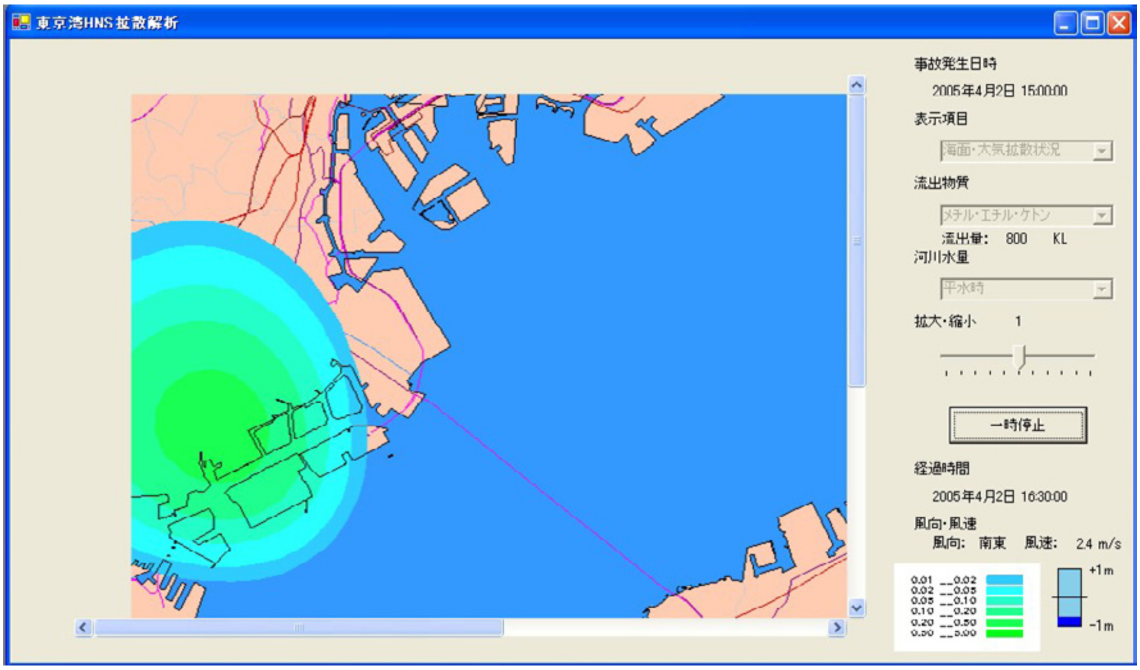


図 2.2-7(1)　メチルエチルケトン海面大気拡散計算結果（0.5 時間後、1 時間後）

〈1.5 時間後〉



〈２時間後〉

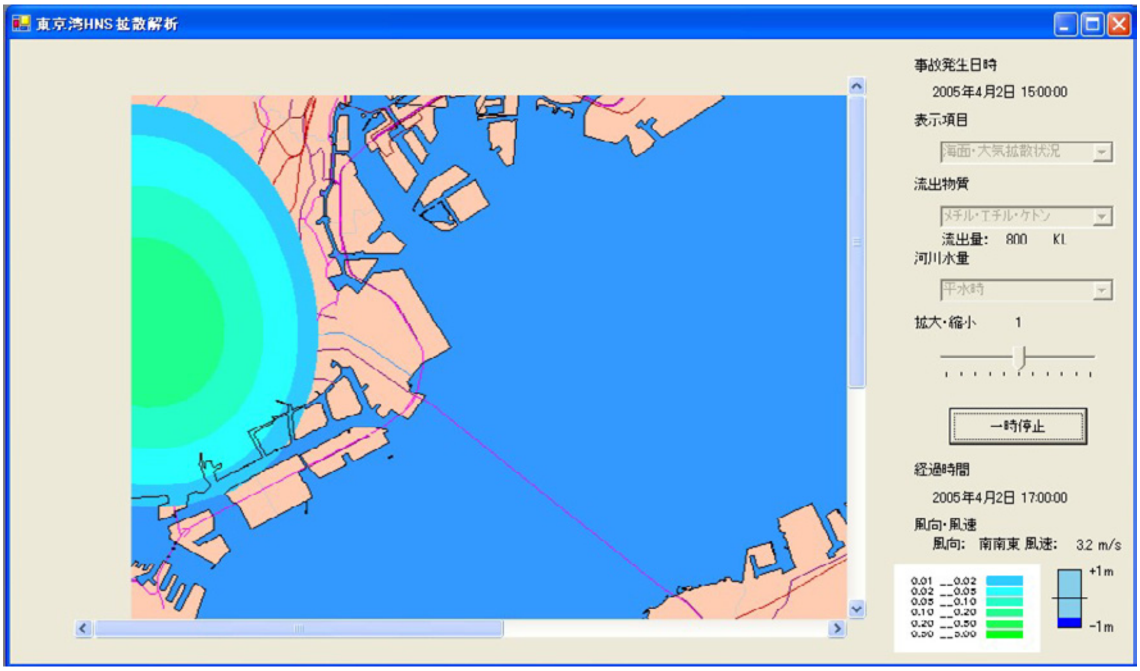
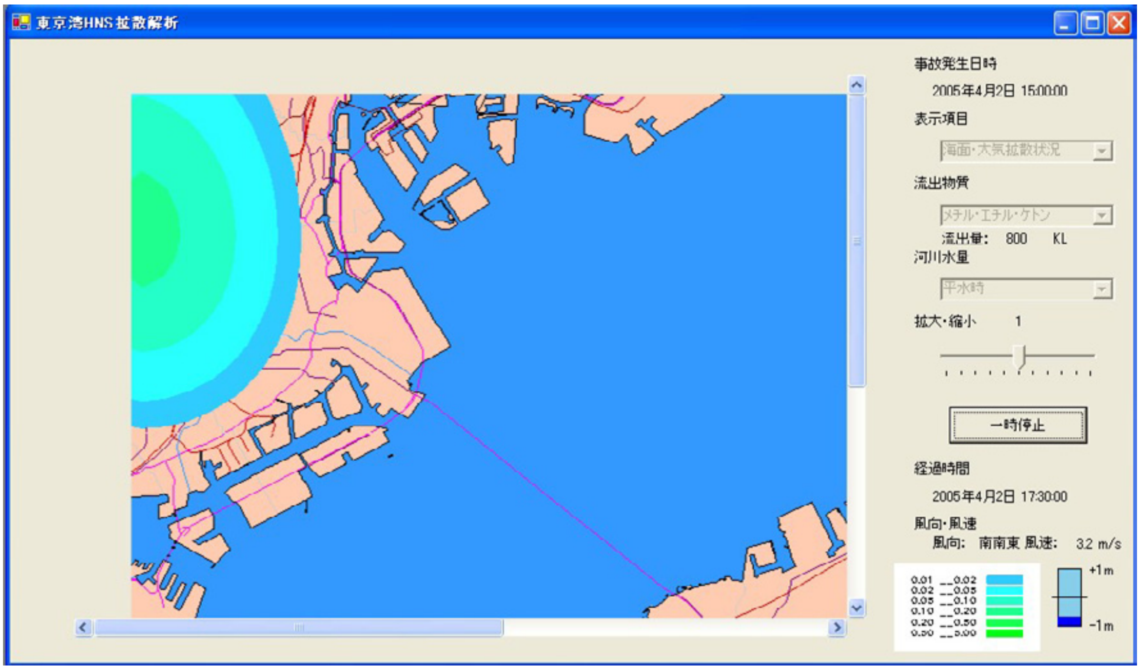


図 2.2-7(2) メチルエチルケトンの海面大気拡散計算結果(1.5 時間後、2 時間後)

〈2.5 時間後〉



〈３時間後〉

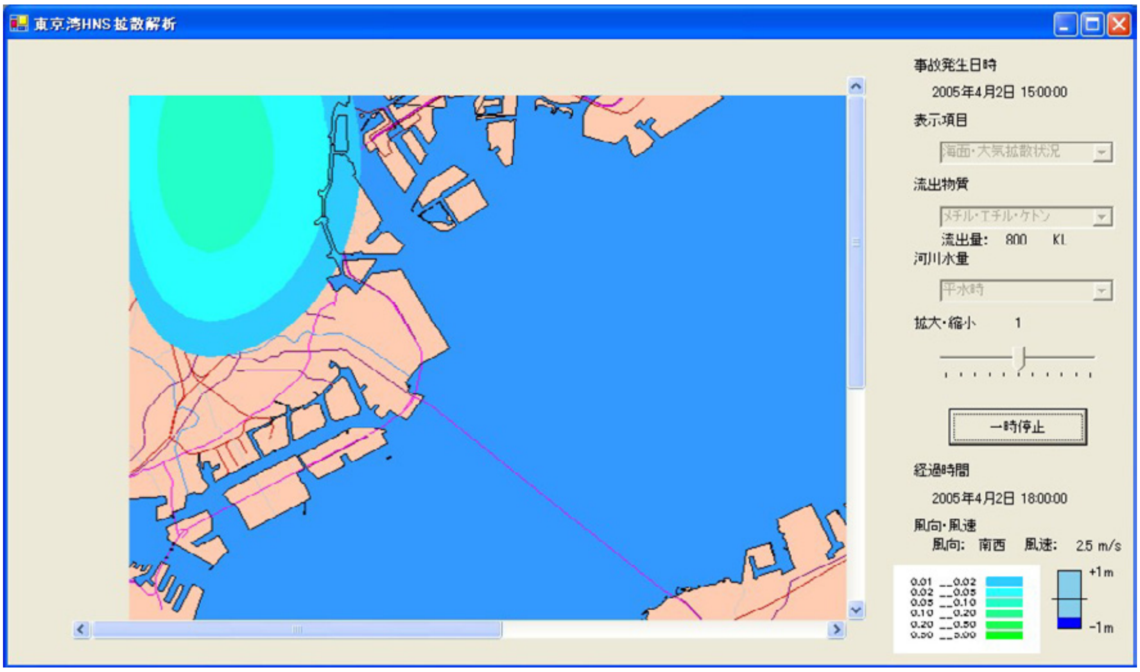


図 2.2-7(3) メチルエチルケトンの海面大気拡散計算結果(2.5 時間後、３時間後)

〈3.5 時間後〉

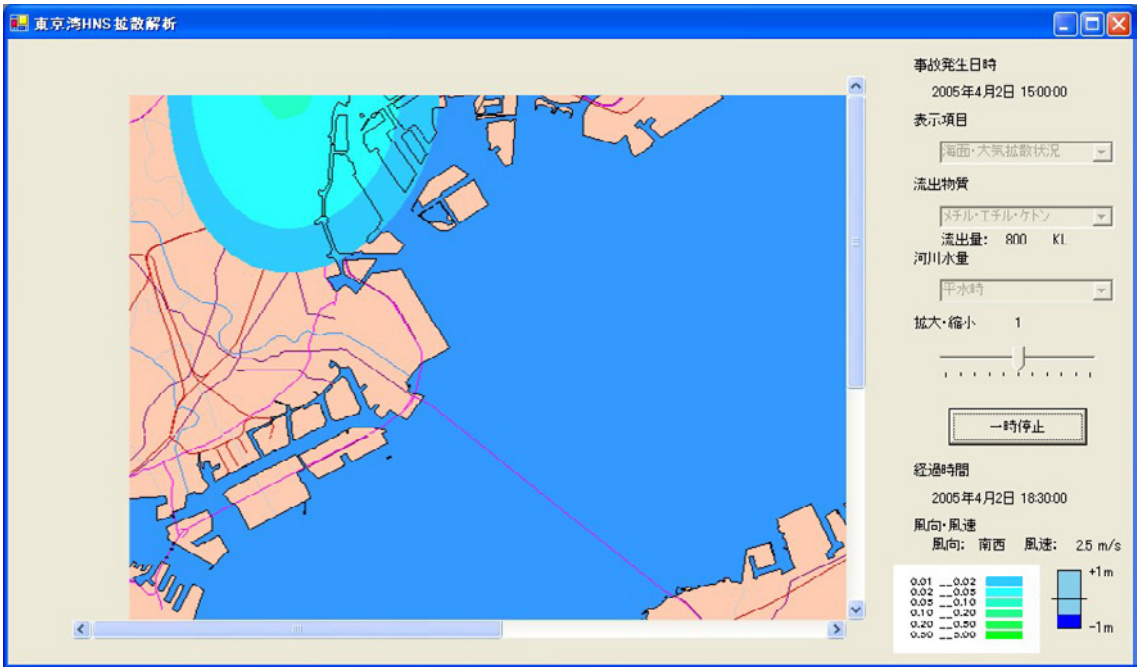
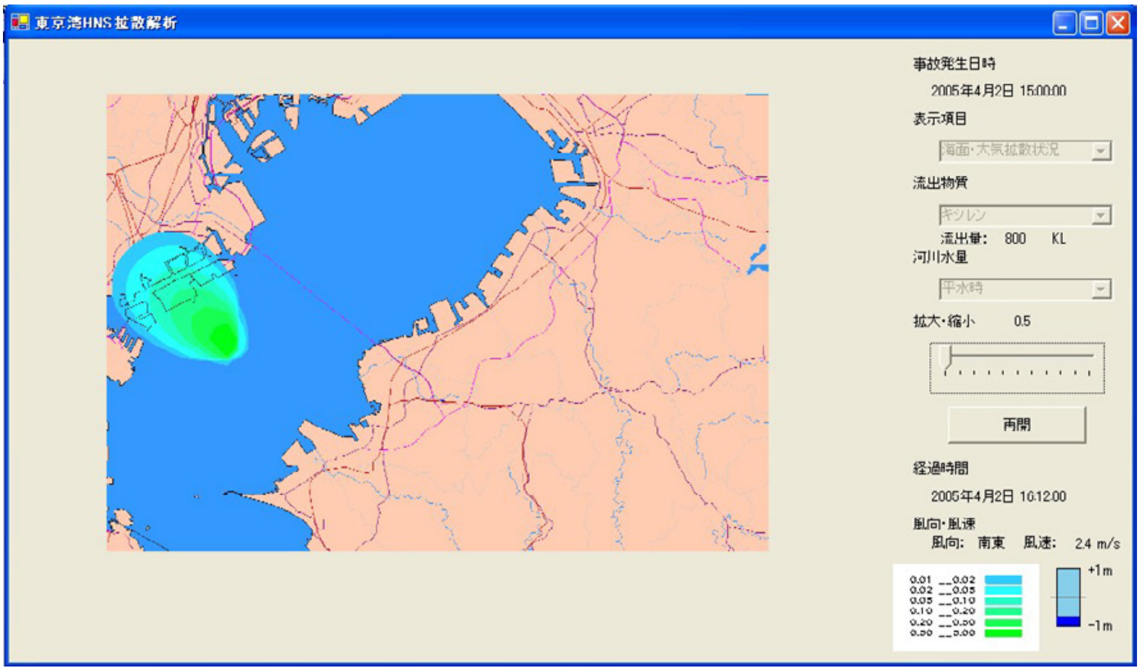


図 2.2-7(4) メチルエチルケトンの海面大気拡散計算結果（3.5 時間後）

２．２．３　その他

各シミュレーション結果を表示する際に、画面の縮小・拡大機能を設定している。設定倍率は 0.5 倍～2.0 倍までであり、実行ボタンの上にあるスライダを左右に移動させる事で倍率を設定する事ができる。（図 2.2-8）

〈0.5 倍時〉



〈2.0 倍時〉

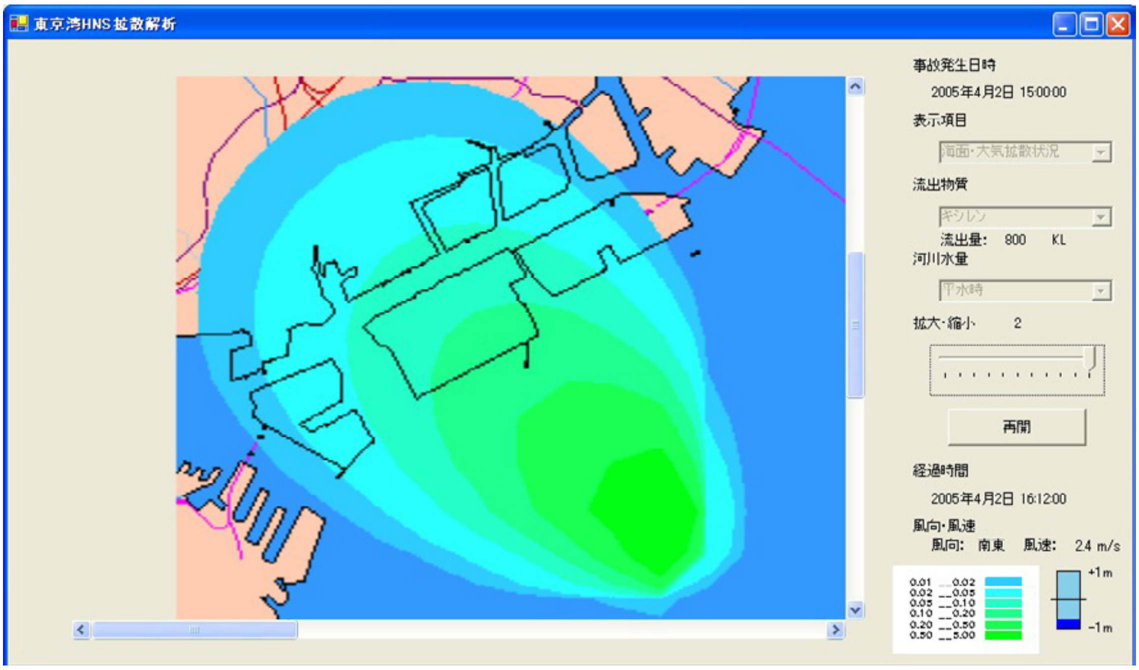


図 2.2-8　縮小・拡大表示例

２．３　プロトタイプモデルの試作のまとめ

プロトタイプモデルの試作のために、モデルの心臓部となる流動モデルや海面拡散モデル、大気拡散モデルを構築した。これにより、流動場や海面・大気拡散状況の予測計算が可能となった。

特に流動場については、主要４分潮の潮位条件のもと、東京湾全域の計算を実施しており、各格子点での調和定数を得る事が出来た。従って、今後のシステムの改良により、任意に設定した時間からの潮流場を、速やかに得る事が出来る。

次に、昨年度研究成果からの改良点として、大気拡散モデルを再検討した事が挙げられる。今回の検討そして改良により、海面上を長時間に渡って漂う物質から蒸発し、その蒸発物質が大気中を拡散する状況の予測計算を、パフモデルを利用するよりも短時間で実施出来る事となった。

また、プロトタイプモデルとして表示システムのベースも作成し、個々の計算結果のアニメーション表示が可能となった。

しかし、以下の課題も残されており、今後の高度化が必要とされる。

○ 流出事故発生場所の設定

○ 風条件など各種条件の設定

第３章　プロトタイプモデルの検証・評価

３．１　モデルの有効性

試作したプロトタイプモデルで用いている計算モデルのうち、流動モデルや大気拡散モデル（格子モデル）は流体力学の分野では古くから利用されており、対象とする事例毎に係数値等の設定が異なるものの、基本構造は本研究で構築したものと同様のものが多い。

そして、海面拡散モデルについては、前章で示した様に、有害物質の流出による沿岸域の被害額を評価し、損害賠償額を算定するための米国一律の標準手法として「1980 年の CERCLA（Comprehensive Environmental Response, Compensation, and Liability Act : 包括的環境対策・補償・責任法)」及び「1990 年の OPA（Oil Pollution Act : 油汚染法)」のもとでの使用を目的としたアセスメントツールである NRDAM/CME モデルを参考として、構築したものである。

従って、海面拡散モデルについても米国で十分に検証がなされたモデルであると考えられる。

このプロトタイプモデルにより、対象とした物質が流出した場合、「どの程度の時間で、どの水域まで拡がり、大気中をどの範囲まで拡散するか」の予測が可能となった。

これにより、事故発生時の防除対策や、近隣住民への警報および避難勧告など被害の極小化のための有効なツールとなる。

また、モデルを構築した事により、各対象物質の流出量や風速条件、そして温度条件を変化させた解析により、海面上に残存する時間の違いを検討する事が可能となり、有用なデータの蓄積にも利用できる。

そこで、次節で流出量と環境条件の違いによる蒸発終了迄の時間（海面残存時間）を検討する。

３．２　モデルを利用した有用データの蓄積

構築したモデルにより、各物質の流出量および風速条件別の蒸発終了迄の時間を計算した。その結果、流出した物質の蒸発終了迄の時間は流出量が多くなると僅かに長くなり、風速が速くなると急激に短くなった。

以下に、各物質の蒸発終了迄の時間について示す。

（１）キシレン

本研究で選択した５物質の中で、無風条件下で蒸発終了迄の時間が最も長く表 3.1 に示した結果を見ると、1000kl の流出で約 10 時間、2000kl の流出で約 12 時間となった。

しかし、風が 2m/s 吹くだけで、1000kl の流出で約 3.5 時間、2000kl の流出で約 4 時間となり、急激に時間が短くなる。

なお、風速 8m/ｓ時には、それぞれ約 2 時間となり、風速 2m/s 時より凡そ半減すると予測された。

表 3.1 キシレンの蒸発終了迄の時間

温度条件：20℃		風条件		
		無風条件	2m/s 条件	8m/s 条件
流出量	500kl	8.53 時間	2.90 時間	1.47 時間
	1000kl	10.03 時間	3.45 時間	1.75 時間
	1500kl	11.02 時間	3.82 時間	1.93 時間
	2000kl	11.77 時間	4.10 時間	2.08 時間

（２）ベンゼン

表 3.2 に示す様に、ベンゼンの蒸発終了迄の時間は短く、無風条件下の 1000kl の流出で約 3 時間、2000kl の流出で約 3.5 時間となった。

しかし、風が吹くと 1 時間以内に蒸発が終了する事が予測された。特に風速 8m/s 時には、流出量が 2000kl であっても蒸発終了迄の時間が約 20 分程度であり、流出量 500kl 時の時間と大差なく、流出後急速に蒸発し、海面上から消失すると予測された。

表 3.2 ベンゼンの蒸発終了迄の時間

温度条件：20℃		風条件		
		無風条件	2m/s 条件	8m/s 条件
流出量	500kl	2.42 時間	0.55 時間	0.27 時間
	1000kl	2.87 時間	0.67 時間	0.32 時間
	1500kl	3.17 時間	0.73 時間	0.35 時間
	2000kl	3.40 時間	0.78 時間	0.37 時間

（３）スチレン

表 3.3 に示す様に、スチレンの蒸発終了迄の時間は比較的長く、無風条件下の 1000kl の流出で約 8.5 時間、2000kl の流出で約 10 時間であった。

しかし、風が 2m/s 吹くと 1000kl の流出で約 3.5 時間、2000kl の流出で約 4 時間となり、急激に時間が短くなる。

なお、風速 8m/s 時には、それぞれ約 2 時間となり、風速 2m/s 時より凡そ半減すると予測された。

表 3.3 スチレンの蒸発終了迄の時間

温度条件：20℃		風条件		
		無風条件	2m/s 条件	8m/s 条件
流出量	500kl	7.27 時間	2.95 時間	1.53 時間
	1000kl	8.52 時間	3.50 時間	1.83 時間
	1500kl	9.33 時間	3.87 時間	2.03 時間
	2000kl	9.97 時間	4.13 時間	2.18 時間

（４）エタノール

表 3.4 に示す様に、エタノールの蒸発終了迄の時間は短く、無風条件下の 1000kl の流出で約 2.5 時間、2000kl の流出で約 3 時間となった。

しかし、2m/s の風が吹くと各流出量ともに約 1 時間で蒸発が終了する予測された。そして、さらに風速が増した 8m/ｓ時には、流出量が 2000kl であっても蒸発終了迄の時間が約 40 分程度であり、流出量 500kl 時の時間と大差なく、上述のベンゼン同様に流出後急速に蒸発し、海面上から消失すると予測された。

表 3.4 エタノールの蒸発終了迄の時間

温度条件：20℃		風条件		
		無風条件	2m/s 条件	8m/s 条件
流出量	500kl	2.17 時間	0.92 時間	0.48 時間
	1000kl	2.55 時間	1.08 時間	0.57 時間
	1500kl	2.78 時間	1.20 時間	0.63 時間
	2000kl	2.98 時間	1.30 時間	0.68 時間

（５）メチルエチルケトン

表 3.5 に示す様に、メチルエチルケトンの蒸発終了迄の時間は短く、無風条件下の 1000kl の流出で約 2.5 時間、2000kl の流出で約 3 時間となった。

しかし、2m/s の風が吹くと流出量 1000kl で約 40 分、2000kl で約 1 時間後に蒸発が終了する予測された。そして、さらに風速が増した 8m/ｓ時には、流出量が 2000kl であっても蒸発終了迄の時間が約 20 分程度であり、流出量 500kl 時の時間と大差なく、上述のベンゼンやエタノール同様に流出後急速に蒸発し、海面上から消失すると予測された。

表 3.5 メチルエチルケトンの蒸発終了迄の時間

温度条件：20℃		風条件		
		無風条件	2m/s 条件	8m/s 条件
流出量	500kl	2.02 時間	0.62 時間	0.30 時間
	1000kl	2.38 時間	0.73 時間	0.37 時間
	1500kl	2.63 時間	0.82 時間	0.40 時間
	2000kl	2.82 時間	0.87 時間	0.43 時間

３．３　プロトタイプモデルの検証・評価のまとめ

本研究において HNS 海面・大気拡散予測に係わるプロトタイプモデルを構築した事で、事故発生時の防除対策や、近隣住民への警報および避難勧告など被害の極小化のための有効なツールになり得る事が確認された。

また、モデルを用いて、流出した物質の蒸発終了迄の時間を計算し、有用なデータの蓄積にも利用出来る事が確認された。

前節「３．２」では、この時間を温度条件 20℃、流出量４種、風速条件３種で計算した。この結果、各物質ともに一定風速条件下では流出量の増加に伴い、残存時間の差が小さくなるものと推察される。また、温度条件によっても、結果が変わってくる可能性もあるため、次の条件で各物質の蒸発終了迄の時間を計算した。

○ 温度条件；５種（5℃，10℃，15℃，20℃，25℃）
○ 風速条件；５種（0m/s，2m/s，4m/s，6m/s，8m/s）
○ 流 出 量；200 種（2000kl まで 10kl 毎）

図 3.1 に示す結果の様に、各物質ともに温度条件による蒸発終了迄の時間の違いは殆ど見られない。しかし、風速条件の違いにより時間にも差が見られ、特に無風時と有風時の差が大きく現れている。また、風速の速い時ほど流出量の違いによる、蒸発終了迄の時間の差が小さくなる事が確認された。

以上、今年度研究において対象とした５種類の HNS について、蒸発終了迄の時間を検討した。今後、プロトタイプモデルを更に高度化し、他の HNS についても同様の解析を実施することで流出事故発生時の初動体制の強化など、有効性の向上が期待出来る。

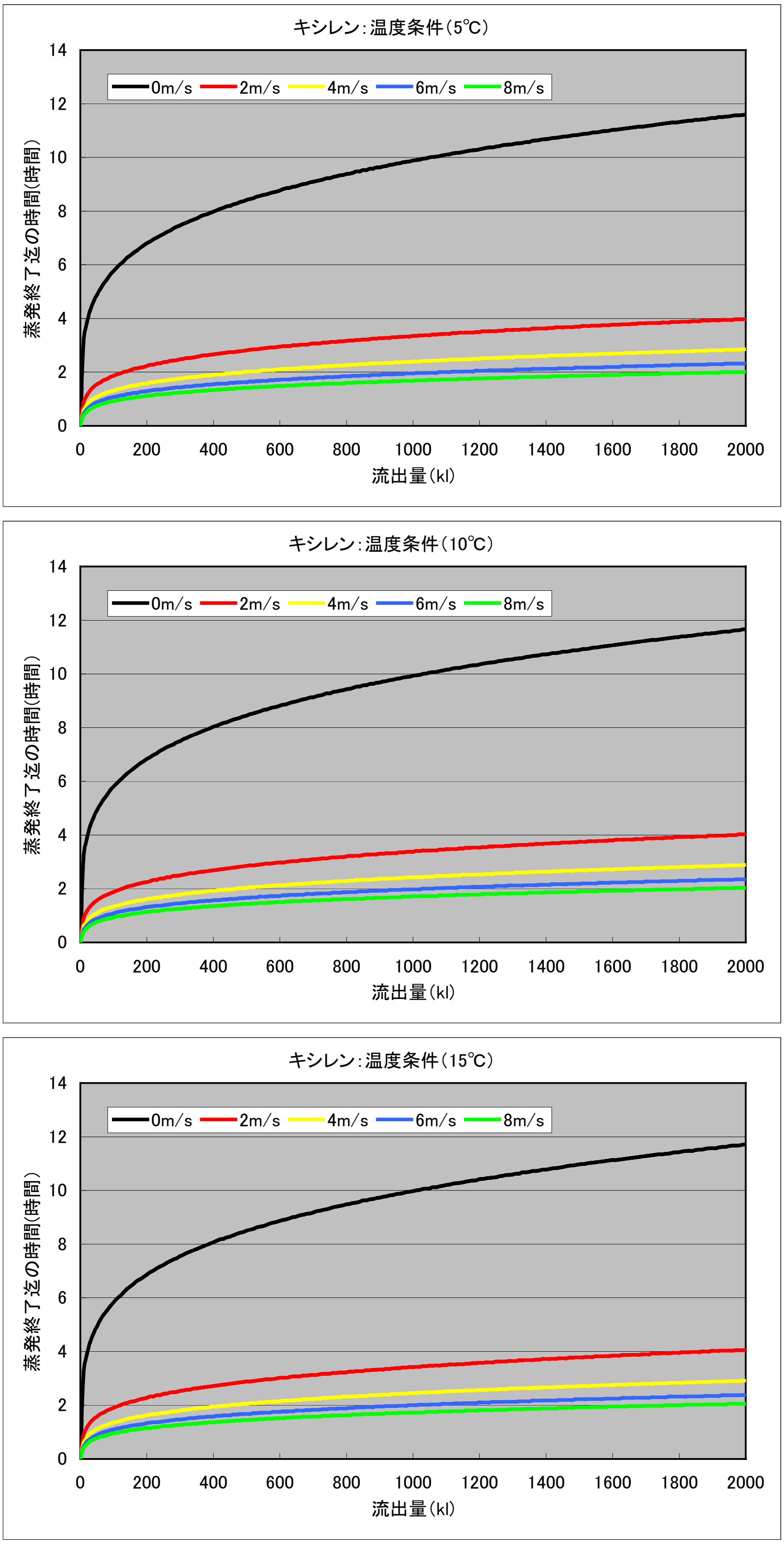
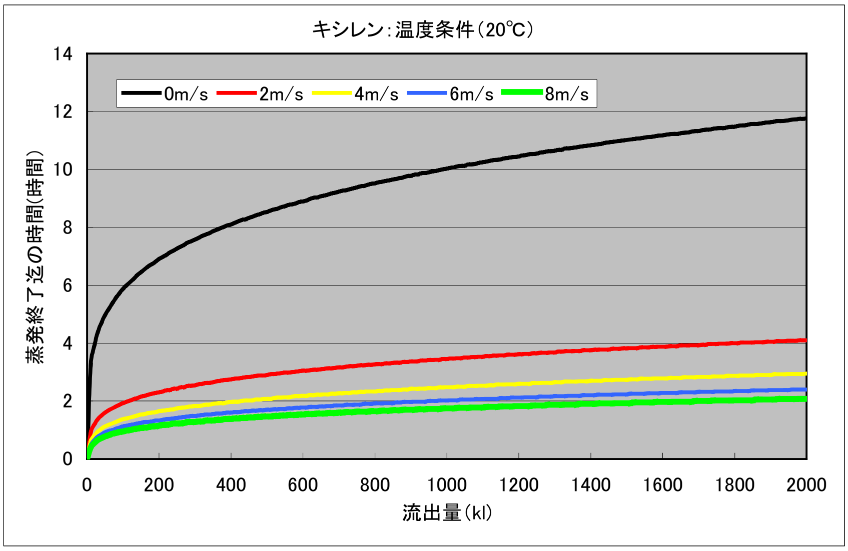


図 3.1(1)　環境条件別の蒸発終了迄の時間（キシレン：5,10,15℃条件）



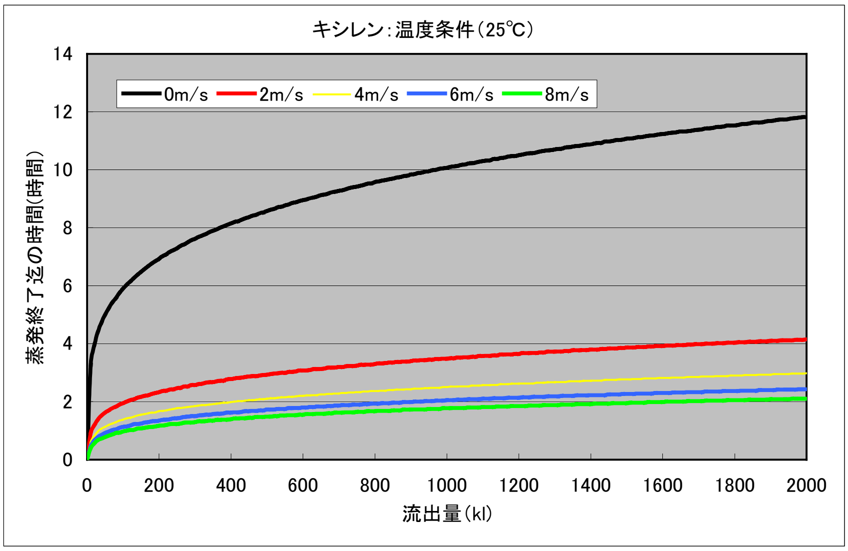
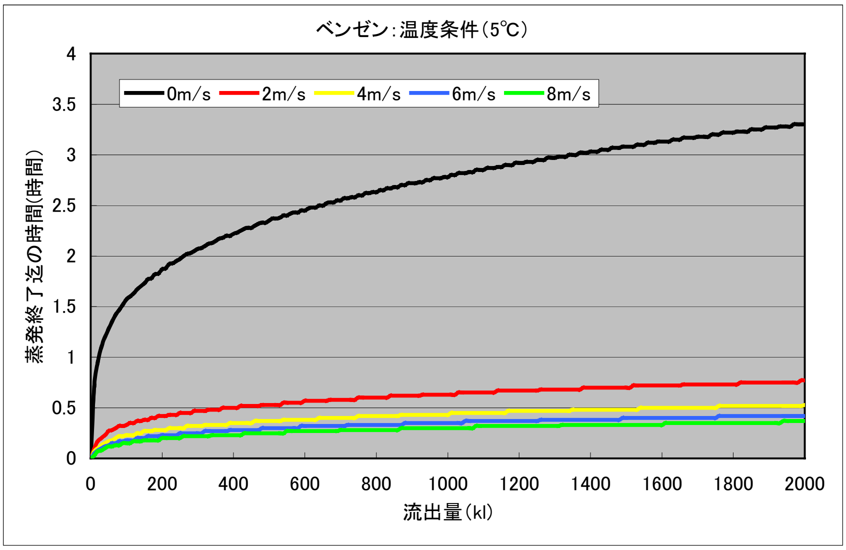
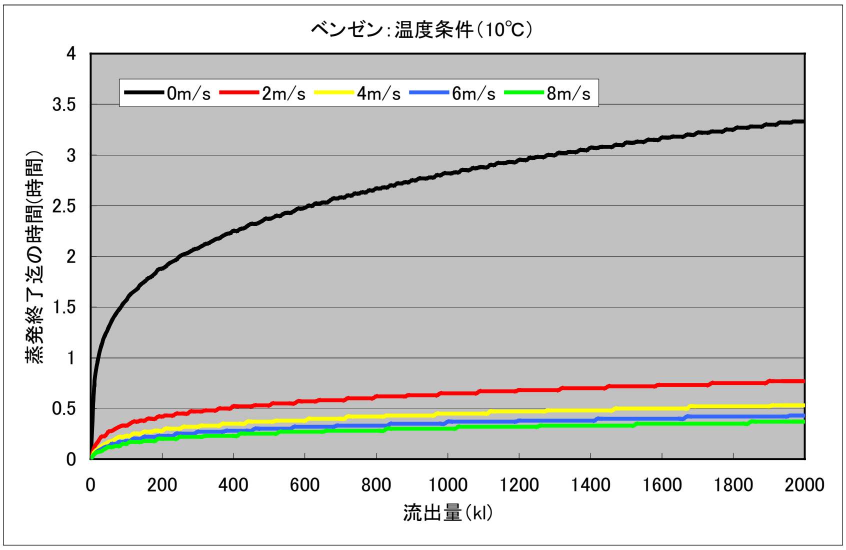


図 3.1(2)　環境条件別の蒸発終了迄の時間（キシレン：20,25℃条件）





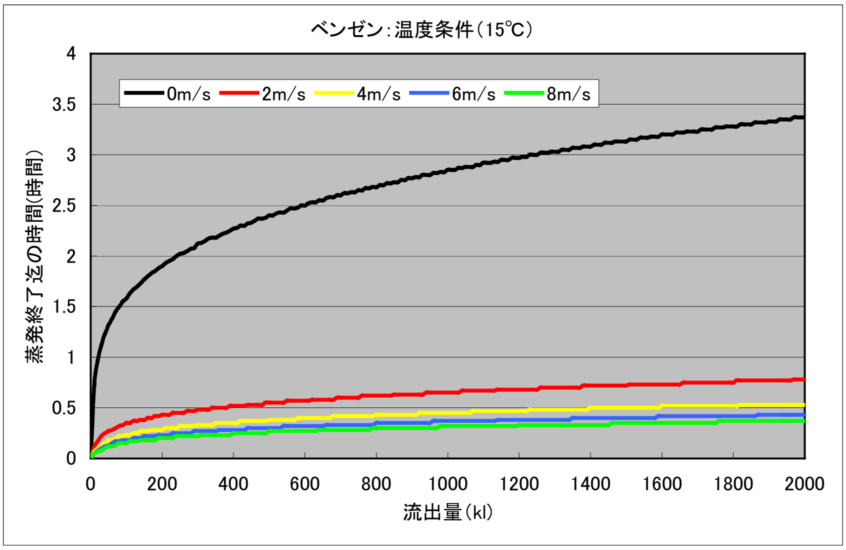
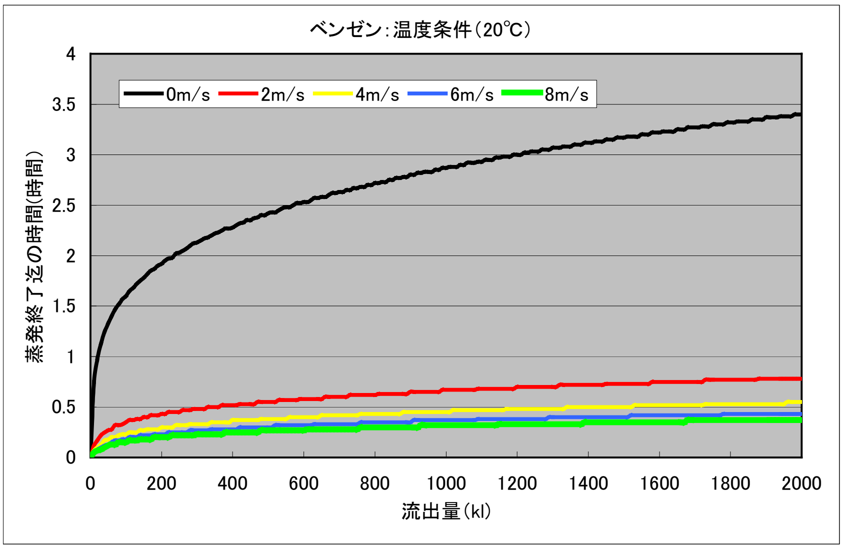


図 3.1(3)　環境条件別の蒸発終了迄の時間（ベンゼン：5,10,15℃条件）



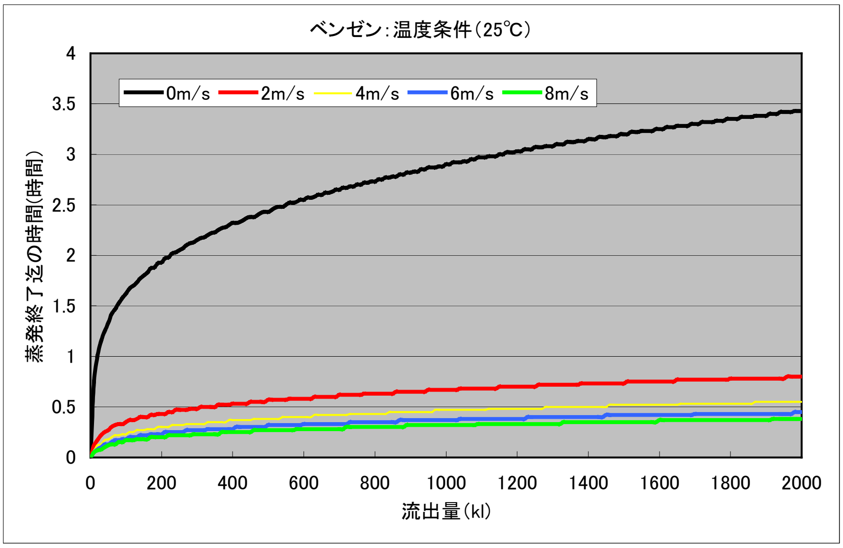
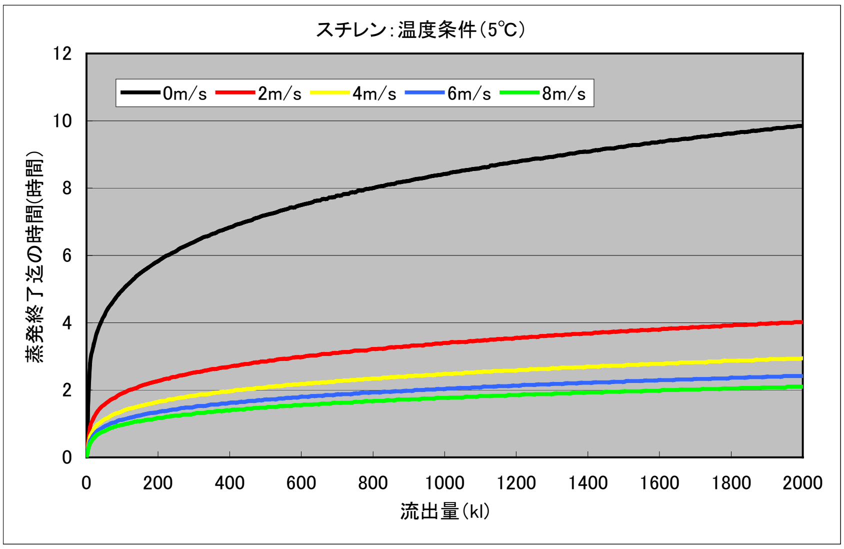
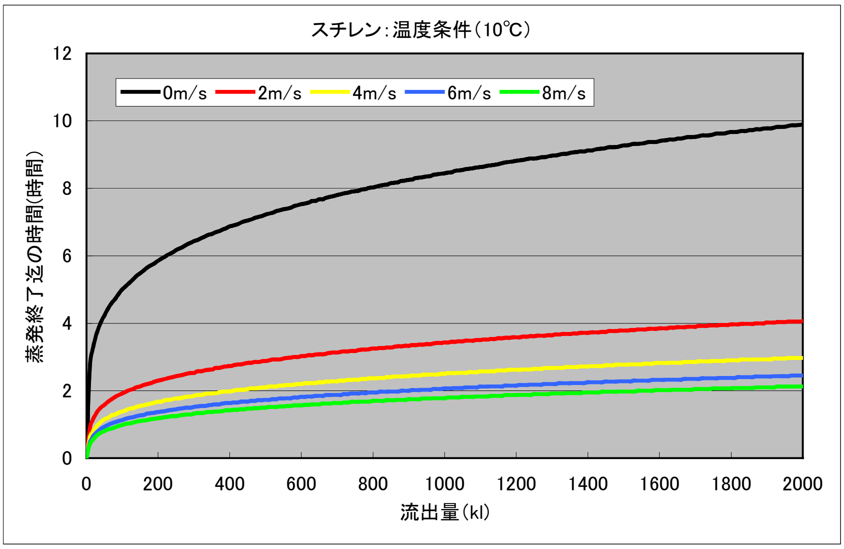


図 3.1(4)　環境条件別の蒸発終了迄の時間（ベンゼン：20,25℃条件）





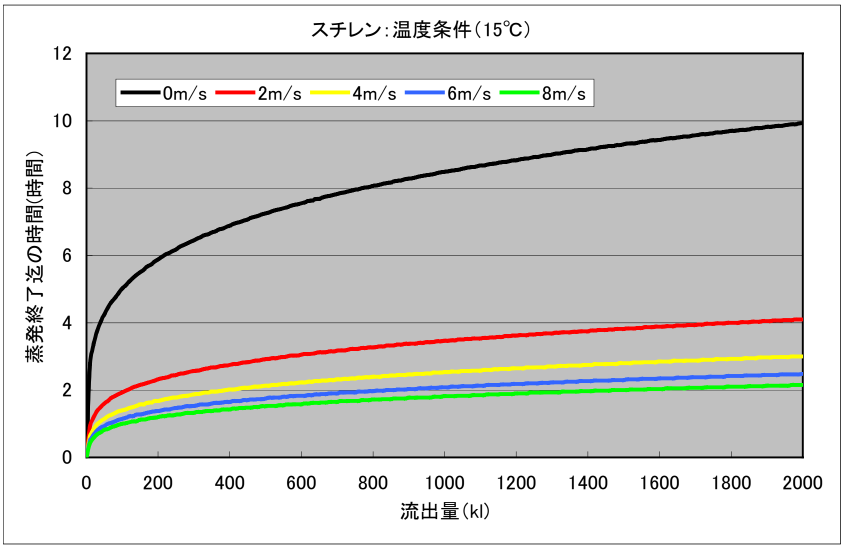


図 3.1(5)　環境条件別の蒸発終了迄の時間（スチレン：5,10,15℃条件）

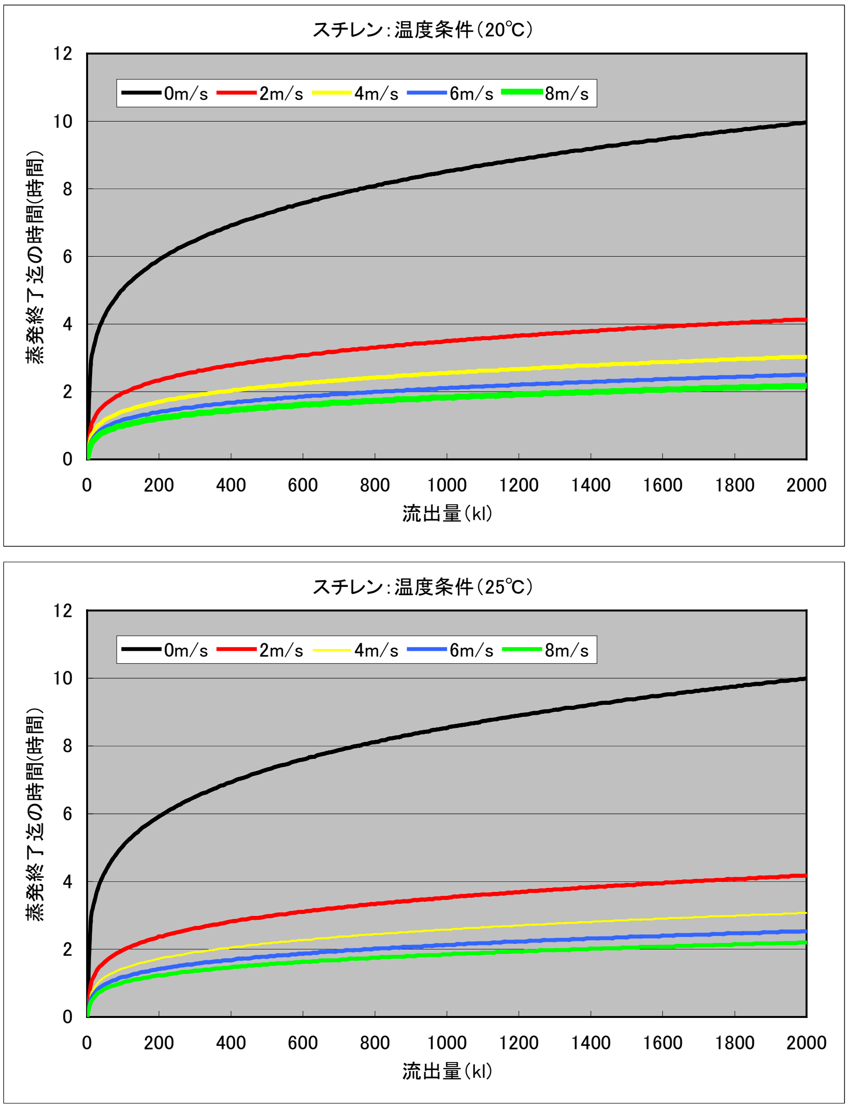


図 3.1(6)　環境条件別の蒸発終了迄の時間（スチレン：20,25℃条件）

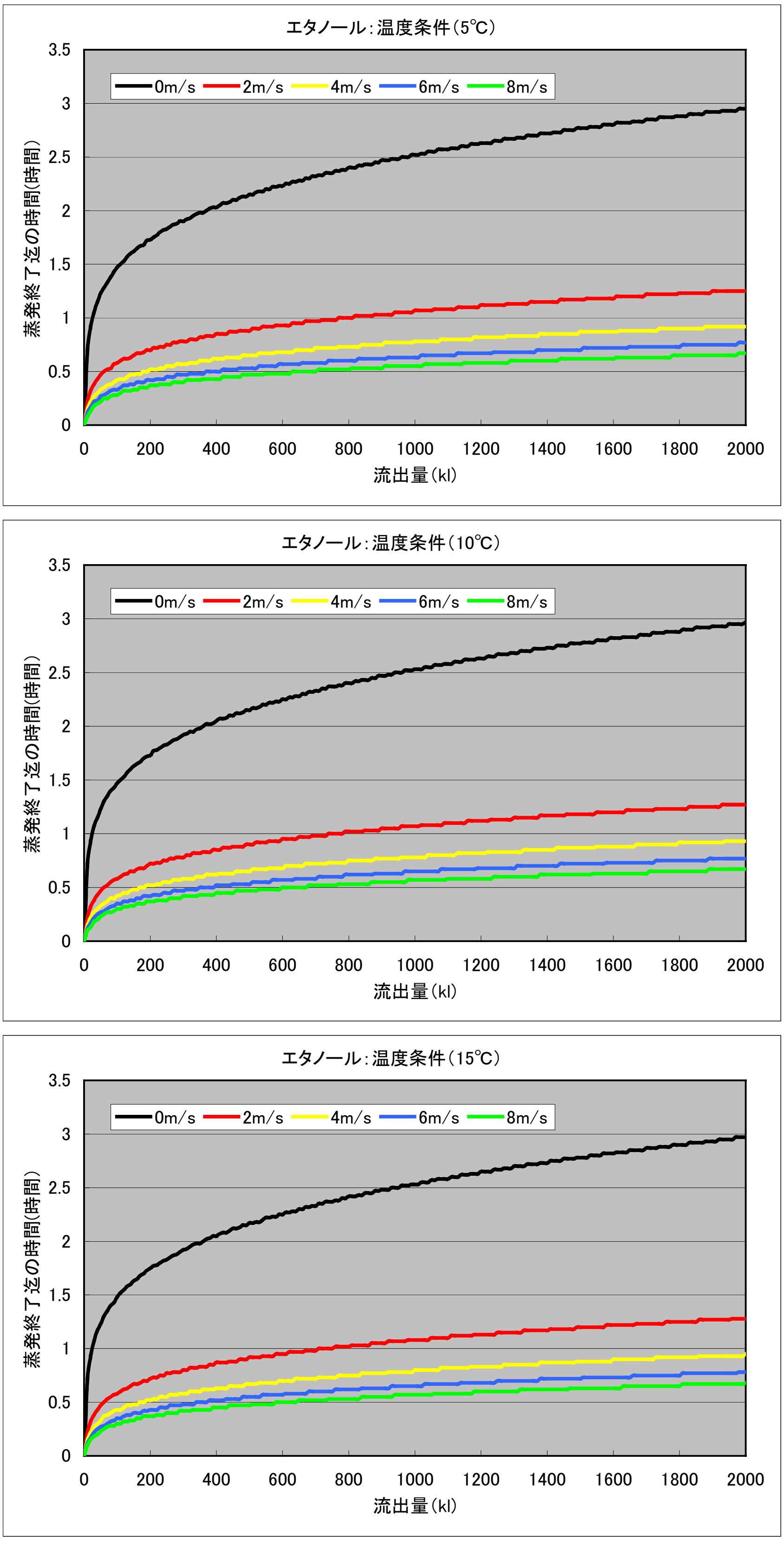
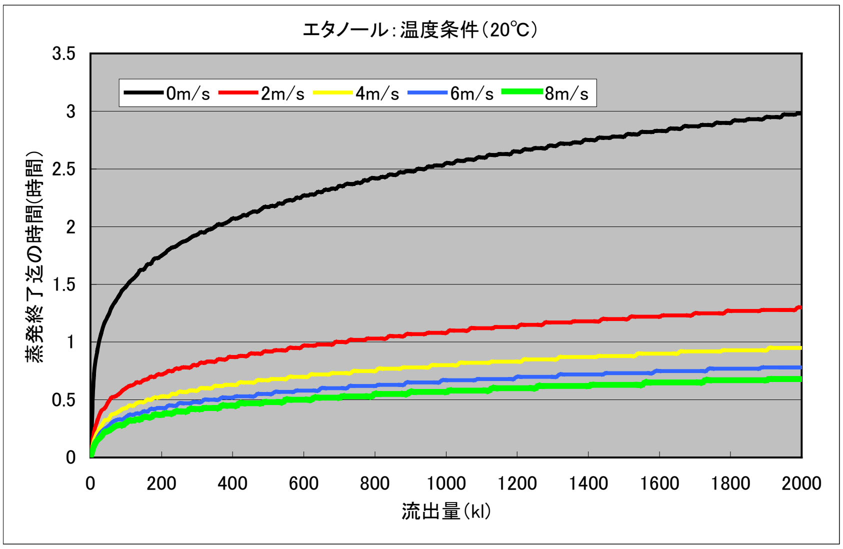


図 3.1(7)　環境条件別の蒸発終了迄の時間（エタノール：5,10,15℃条件）



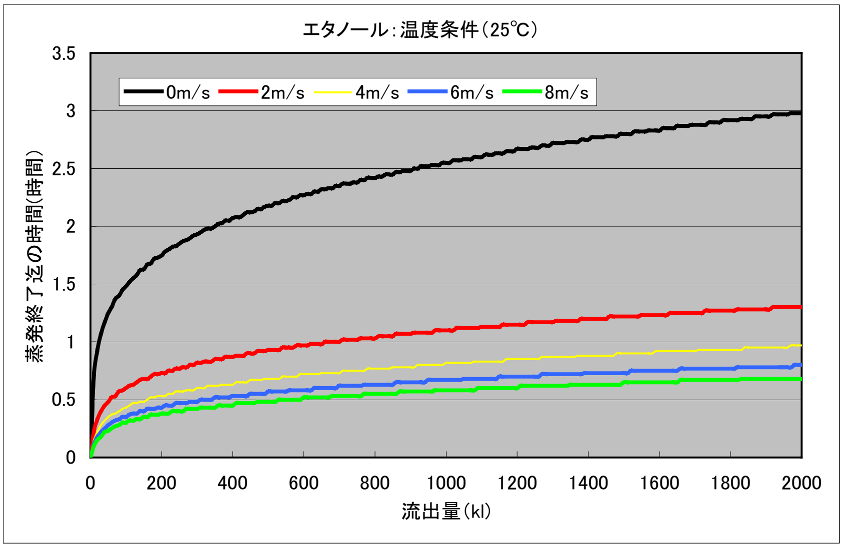
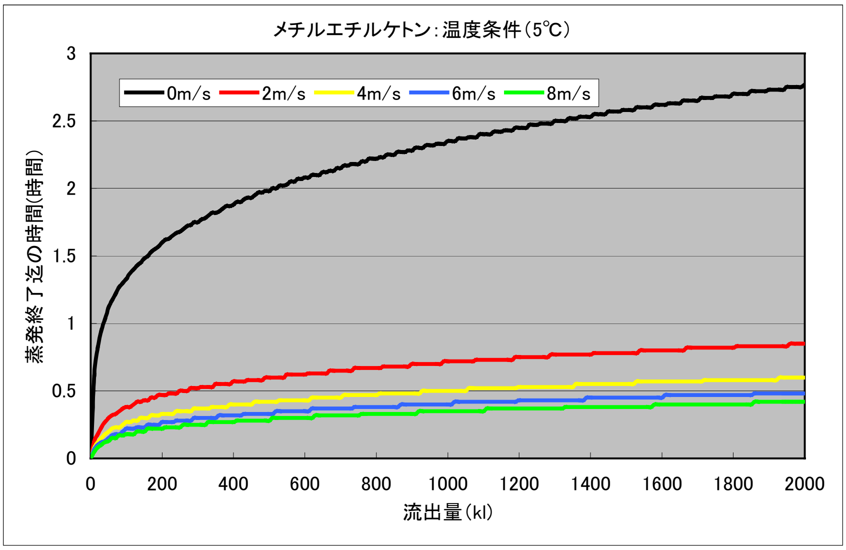
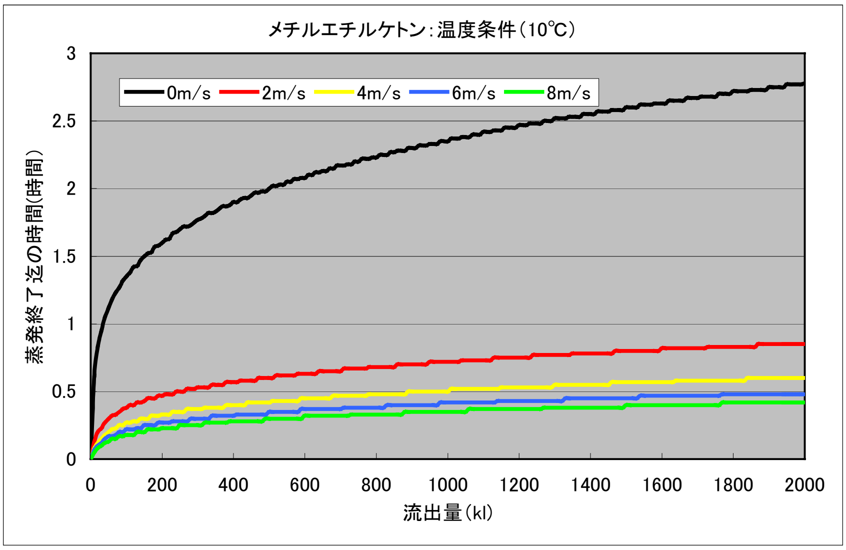


図 3.1(8)　環境条件別の蒸発終了迄の時間（エタノール：20,25℃条件）





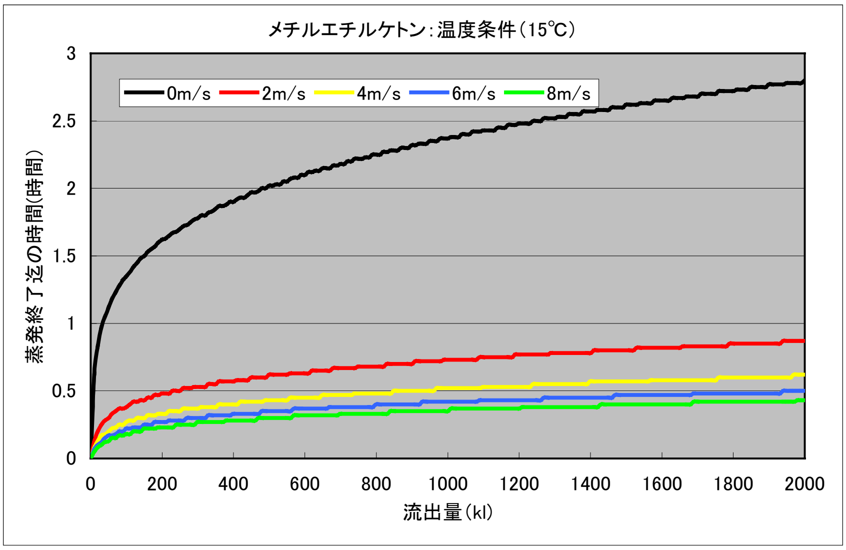
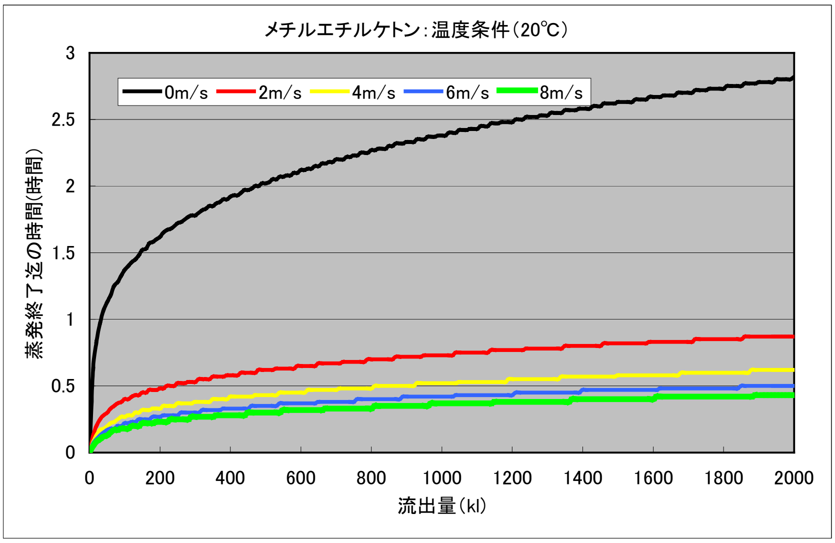


図 3.1(9) 環境条件別の蒸発終了迄の時間（メチルエチルケトン：5, 10, 15℃条件）



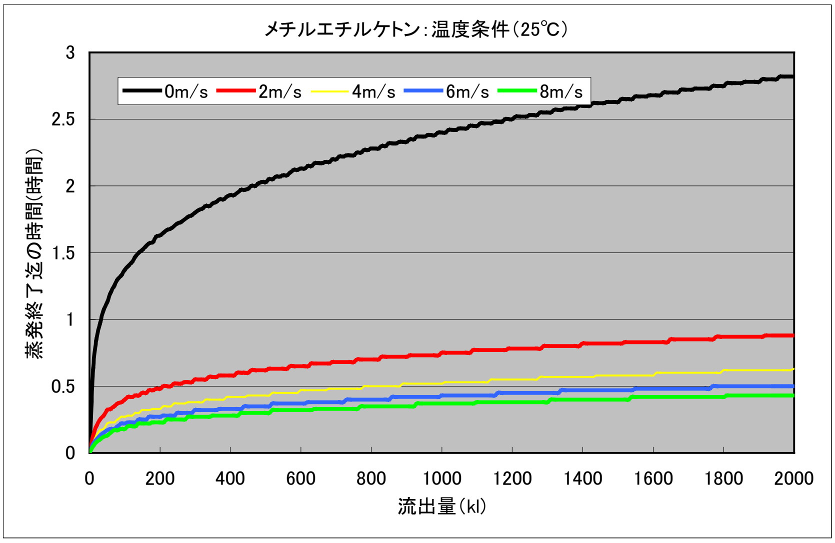


図 3.1(10) 環境条件別の蒸発終了迄の時間（メチルエチルケトン：20,25℃条件）

第４章　課題の整理

今年度調査により得られた今後の課題を抽出・整理した結果について、以下に記す。

今年度実施したプロトタイプモデルの試作では、モデルの心臓部となる流動や海面・大気拡散を計算する計算プログラムを構築し、表示システムの試作を行った。しかし、プロトタイプモデルでの計算条件の設定方法など、モデル高度化のために必要とされる課題も抽出された。

以下に、プロトタイプモデル高度化のための課題点を示す。

（１）対象とするHNS種の追加

今年度は5種類の HNS を対象にしてモデルを構築し、有用なデータの蓄積にも利用出来る事が確認された。

そこで、平成15年度調査で整理されたＨＮＳ92種について、モデルで設定している物性に係わる係数値を整理する事が必要と考えられる。

これにより、各物質の流出事故発生時の予測解析が可能となる。

（２）モデルの計算条件設定に関して

モデルを用いて、事故発生時以降の予測を迅速に実施するための課題点について、以下に示す。

① 流出事故発生場所および流出量の設定

各種計算プログラムが構築されたので、次いで事故発生場所をマウスによるプロットや、緯度経度の入力等により設定出来る様にシステム化する事が必要である。

② 流出事故発生日時の設定

東京湾の潮流計算を実施した事で、各計算格子での調和定数の利用が可能となった。この調和定数を利用して、指定日時からの潮流場を瞬時計算する様に、システムの高度化を行う事が必要である。

③ 気象条件の設定

海表面の流れや、流出物質の蒸発に大きく影響する風条件、そして気温の設定について、流出事故発生時の実測値の入力や統計値の利用、また気象予報値の取込などを選択し、設定出来るようにする事が必要である。

④ 河川水条件の設定

今年度調査では、河川水による流れは平水時条件１ケースのみの設定とした。しかし、河川水の流入は、海表面の流れ場に影響するため、出水時や

豊水時、渇水時の条件についても設定する事が必要である。

（３）モデルの計算対象の追加

今年度調査のプロトタイプモデルは、海面と大気中の拡散状況を予測する様に作成した。しかし、数多く存在する HNS の中には海水比重よりも重い物質も存在するため､水中への沈み込みや海底への堆積も予測出来るシステムが望まれる。

そのために、海中の挙動も考慮できるモデル開発が必要である。

資　料　編

〇NRDAM/CMEモデルのユーザーズマニュアル

The CERCLA Type A Natural Resource Damage Assessment Model for Coastal and Marine Environments (NRDAM/CME)

User's Manual
Volume II

for

Office of Environmental Policy and Compliance
U.S. Department of the Interior
Washington, DC 20240

Prepared by:

Applied Science Associates, Inc.
70 Dean Knauss Drive
Narragansett, RI 02882

April 1996

Revision I, dated October 1997

Contract No. 14-01-0001-91-C-11
ASA 90-53

TABLE OF CONTENTS

Page

1. INTRODUCTION

The Natural Resource Damage Assessment Model for Coastal and Marine Environments (NRDAM/CME) is a tool for assessing natural resource damages resulting from minor spills of hazardous substances or oils in coastal and marine environments. The NRDAM/CME is designed for use under the Comprehensive Environmental Response, Compensation, and Liability Act of 1980, as amended (CERCLA) and the Oil Pollution Act of 1990 (OPA). This document describes the installation and application of the NRDAM/CME.

CERCLA authorizes Federal, State, and Indian tribal natural resource trustees to assess and obtain compensation from parties responsible for hazardous substance releases that cause natural resource injuries. OPA provides trustees with similar authority in cases of oil discharges into navigable waters. The Department of the Interior has issued regulations for conducting natural resource damage assessments under CERCLA that are codified at 43 Code of Federal Regulations Part 11. The National Oceanic and Atmospheric Administration has issued regulations for assessments under OPA that are codified at 15 Code of Federal Regulations Part 990. Federal and State trustees who conduct hazardous substance assessments in accordance with the CERCLA regulations receive a rebuttable presumption if they proceed to litigation. Federal, State, and Indian tribal trustees receive a rebuttable presumption for oil assessments conducted in accordance with the OPA regulations.

The NRDAM/CME is one of the simplified "type A" procedures incorporated into the CERCLA regulations. The OPA regulations allow any type A procedure incorporated in the CERCLA regulations to be used for assessments under OPA as well, so long as certain criteria are met. This document is solely intended to provide users with a better understanding of how to install and apply the NRDAM/CME. Trustees who wish to use the NRDAM/CME and obtain a rebuttable presumption for hazardous substance assessments must consult the CERCLA regulations on the applicability of the model, the necessary and permissible user-supplied information, the required administrative process, and the conditions under which the model results can be supplemented with site-specific "type B" studies. Trustees who wish to obtain a rebuttable presumption for oil assessments, must consult the OPA regulations.

The computer model runs on an IBM-compatible personal or lap-top computer, and is contained on the diskettes distributed with this document. A graphical user interface allows easy access to the model, its databases and its output.

The NRDAM/CME system includes four main submodels, as follows:

Physical fates submodel

Biological effects submodel

Compensable value submodel

Restoration submodel.

The physical fates submodel determines what happens to a chemical which is spilled; where it goes, and how it is partitioned in the environment as time proceeds (on the water surface, in the water column, in the bottom sediments, or on the shoreline). The biological effects submodel determines what organisms are likely to come into contact with the spilled chemical and what the short and long term effects of that contact will be. The compensable value submodel calculates the value of lost or injured resources. The restoration submodel determines appropriate restoration actions and their costs.

In this volume the installation and use of the NRDAM/CME system are described. Section 2 defines the hardware and software required to run the model system and describes the installation procedure. Section 3 presents an overview of the graphical interface for the model system and describes the use of features common throughout the system. Section 4 details the steps involved in using the NRDAM/CME system to run a spill scenario and evaluate damages to the environment. Section 5 describes some of the file types used by the system and batch processing. An appendix presents formats for a variety of file types used by the model, and information on types of data available for winds and currents.

The NRDAM/CME system enables the user to simulate the trajectory of a hazardous material spill and its fate in the water, on the shore, and in the sediments. Associated damages to wildlife and fish and to recreational areas are assessed, and the cost of restoration or mitigation is calculated.

Embedded within the system are data for coastlines, bathymetry, and habitat types within each location. Use of the model system relies on an easy-to-use graphical interface to make the many complex components of the system accessible. Environmental data are user-selectable and include the ability to enter up-to-date wind data, tidal and background currents.

The physical fates submodel predicts the distribution (trajectory) and fate (mass balance) of a substance as a function of time for instantaneous or continuous spills from a surface source. A library of 459 hazardous materials, including pure chemicals, crude oils and petroleum products, is included with the model.

The first screen which appears when NRDAM/CME is run displays the main menu and a map of the last location for which the system was used. A hypothetical session to set up a new area proceeds as follows. The user starts the system by entering its acronym, NRDAMCME. A map appears on the screen, with the main menu across the top. The user pulls down the LOCATION item, then selects the coastal area encompassing the study area. The user may then select DATA from the Main Menu. Here the user can choose among a variety of options for creating or editing winds, currents, closures and habitats. Having created or modified the needed environmental data sets, the user can proceed to RUN MODEL and carry out simulations.

The RUN MODEL branch of the menu allows the user to select the physical fates submodel, or to run the biological effects and compensable value submodels using output from a previous fates simulation for a spill scenario. Model computations are displayed on the screen as the model runs; in addition, results are stored in files keyed to the scenario name, and are available for viewing or printing under the OUTPUT DATA branch, in both tabular and graphical form.

Under the SYSTEM branch, a variety of system management tools are available. These are designed to set the data path and measurement units, to manage space on the hard disk and to set map display colors.

2. INSTALLATION

2.1 Resources Required

The NRDAM/CME system is designed to be user-friendly and use clear, concise graphical procedures to enter data and display model output. The software is designed for an IBM-compatible PC running under DOS with a VGA color monitor. A wide array of printers may be used for printing graphics displays or text from within the NRDAM/CME system. Reports of model simulations may also be printed directly from the ASCII output files generated by the model on any text printer you normally use. See the discussion in Chapter 3 for details on printing.

An 80386, 80486 or 80586 machine with VGA color monitor is recommended; a Microsoft-compatible mouse and the keyboard are employed to control the program input, execution, and output.

The configuration required to run the model is as follows:

- IBM-compatible PC
- DOS 3.3 or higher
- 80386, 80486 or 80586 processor
- Math co-processor
- 1.4 MB 3.5" floppy disk drive
- 4 megabytes of RAM with 540 KB of conventional memory available
- Hard disk with 75 MB of available space
- VGA color monitor
- Microsoft-compatible mouse and mouse driver software

NRDAM/CME may be run from Windows version 3.1, however this reduces the amount of memory available to run the models and in some cases may result in unpredictable effects such as lockup. Thus, it is recommended that Windows not be open during operation of NRDAM/CME. Many simulations have been run from Windows 95 without any apparent problems.

2.2 Standard Installation

The NRDAM/CME system is designed to be run from a hard disk. Before proceeding further with the installation, check to make sure that your hard disk has a minimum of 75 MB available to load and run the model.

You must first install the model from the diskettes distributed with this manual, as follows:

Before installation, examine the set of diskettes. Notice that the model system is provided in several parts and that these parts are each installed separately. They include the model and database, the locations, and the documentation. Install the model first, then install the locations. Installation of the documentation is optional.

To install the model from floppy drive A:

Insert NRDAM/CME Model Disk 1 in drive A:

At the DOS prompt, type

A:INSTALL < Enter >

You may designate any floppy drive letter in the above procedure. For example, if you are installing the diskettes from drive B: you would type the following at the DOS prompt:

B:INSTALL < Enter >

At this point, a menu-driven procedure is initiated which installs the program. Follow the directions presented on the screen; when the installation menu appears, choose the first option: "Install NRDAM/CME" to install the program. The routine will display the amount of space needed for successful installation, and a list of all system drives. Drives in white have enough free space for a successful installation. Drives appearing in gray do not. Choose a drive with enough space and press <Enter>. Installation of the system will then proceed. You will be prompted to insert additional NRDAM/CME disks during the installation procedure. When all program files have been installed, repeat this installation procedure with the location data disks.

2.3 Directory Structure

A new sub-directory is created in your hard disk's top level (root) directory called NRDAMCME; this contains the model itself and the associated databases (see Figure 2.1). Note that the initial installation may not create all of these directories. The model will create them as they are required when appropriate new files are to be stored. About 50 MB of disk space is required for the installation of the model and locations; an additional 15 MB of disk space should be available to run applications of the model (the actual space required for model runs will vary considerably with the types of scenarios that are run). The documentation requires an additional 10 MB of disk space.

```
C:¥─┬─
    ├─NRDAMCME─┬─CONVERT
    │          ├─DOCUMENT──┬─VOL_1
               │           ├─VOL_2
               │           ├─VOL_3
               │           ├─VOL_4─────────TBL_IV-2
               │           ├─VOL_5
               │           ├─SOURCE
               │
               ├─DATA──────────┬─SYSTEM
               │           ├─HELP
               │           ├─MODEL
               │           ├─COMBINED
               │           ├─BIOSYS
               │           ├─CHEM
               │           ├─FONTS
               │           └─DATA_DEF
               │
               ├─LOC_DATA──┬─E_COAST──────┬─COAST
               │           │          ├─WINDS
               │           │          ├─KEYS
               │           │          ├─CURRENTS
               │           │          ├─MODELOUT
               │           │          ├─GRIDS
               │           │          ├─MODEL
               │           │          ├─DEPTHS
               │           │          ├─CASES
               │           │          ├─RESPONSE
               │           │          ├─CLOSURES
               │           │          └─BIOLOGY
               │           │
               │           ├─W_COAST──────┬─COAST
               │           │          ├─WINDS
               │           │          ├─
               │           │          ├─  etc.
               │           │
               │           ├─ALASKA───────┬─COAST
               │           │          ├─  etc.
               │           │
               │           ├─PACIF_IS───┬─  etc.
               │           │
               │           └─CARIBBEAN──┬─  etc.
               │
               └─PROVDATA─────NAMERICA───┬─BIOLOGY
                                  ├─GRIDS──────────┬─USER
                                  │           ├─ASCII
                                  │           └─ORIGINAL
                                  ├─ECONOMIC
                                  └─ENVIRON
```

Figure 2.1 NRDAM/CME directory tree.

2.4 Trouble Shooting

Questions that may arise:

Question:When I type in the installation command I get an error message similar to "Error reading drive A: Abort, Retry, Fail".

Solutions:Either you've gotten a defective disk or your floppy disk drive reads the wrong density of diskette. Diskettes come in two densities: Double density and high density. Because of the amount of data used, NRDAM/CME is only distributed on high density diskettes. If this is your problem you need to install NRDAM/CME on a system with high density disk drives and then use a commercially available backup or file transfer program to move the system from one PC to the other.

Question:I installed the NRDAM/CME system. The installation seemed to work just fine but problems developed when the system was started at the end of the installation procedure.

Solutions:If you get an error message which includes the phrase "floating point not loaded," or "floating point error" then your PC does not have a math co-processor or it is malfunctioning. The only solution for this is to install a functioning math co-processor in your PC or install and run the model on a 386 PC which has a functioning math co-processor; a 486 or 586 PC does not need a math co-processor as this is built into the machine's CPU chip.

Question:My PC just hangs when the NRDAM/CME command is given.

Solutions:It could be that you do not have a VGA monitor. If this is the case you will need to get a VGA graphics card and monitor in order to run NRDAM/CME.

Question:Most of the system seems to work just fine, but some modules either hang my PC or behave oddly.

Solutions:You might not have enough free memory (RAM) to run some of the submodels. Run CHKDSK to determine the amount of free memory you have and make sure you have at least 540 KB free. Some submodels do require this much memory to work properly. If you do not have enough free memory, try to make more by removing any memory resident programs (also known as TSRs) such as network drivers or loading them into high memory.

Question:My PC has additional memory (e.g. 1-8 MB) added to RAM. Can I use the extended RAM to run the model?

Solution:This version of the NRDAM/CME system is designed to take advantage of extended RAM.

3. COMMON USER INTERFACE FEATURES

Several user interface features are used throughout NRDAM/CME for such purposes as data display and file specification. The use of these features is described in detail in this section, and only briefly under the model options that use these features.

Either the keyboard or mouse may be used to select menu options and respond to queries. The mouse may be used to select the desired option or response as described in Section 3.1. The keyboard may also be used, either by moving to the desired option using the arrow keys then pressing ENTER, or by typing the highlighted letter of the menu option or response (e.g., press Y to select Yes).

In forms which have OK and ESCAPE (or ESC) buttons at the top, these actions can be initiated by using the mouse to click on the desired button. From the keyboard the OK option can be selected by pressing the CTRL and ENTER keys simultaneously. Press the ESC key to escape.

Most processes in NRDAM/CME can be halted by pressing the ESC key.

3.1 Using the Mouse

The mouse is used to select menu options and for many of the data entry procedures. To use the mouse move the arrow cursor which appears on the monitor to the desired position and click the left button of the mouse. The term `click' is used throughout the text to indicate the mouse button is to be depressed and released. If the option requires the button to be held down, it is specifically stated. The left button is always used unless otherwise noted.

The mouse may be used to move some objects by `dragging' them. To drag with the mouse, place the cursor over the object to be moved, depress the left mouse button and hold it down. Move the object to its new position, then release the mouse button.

3.2 Zoom

The model options which involve the display of spatial features (specifically, viewing environmental data, entering the spill location, and viewing model predictions) allow the capability to window areas of specific interest. To access this capability, click on the ZOOM option of the sidebar at the right of the monitor screen. The following sub-options will appear:

CREATE WINDOW makes a new window inside the current map. Use the mouse to describe a rectangle within the map window defining the new zoom window by clicking the lower left and upper right corners. Note that when viewing mapped data, the zoomed area displayed maintains a fixed aspect ratio. This is done to keep the shoreline and displayed data from becoming highly distorted, as would be the case if the height and width of the zoom box were widely different.

ZOOM IN allows you to return to previous zoom windows created using the "create window" command. Each time you select the ZOOM IN sub-option, the zoom window created at a larger scale than the one you are currently viewing will be displayed. You can zoom in to any number of previous zoom windows.

ZOOM OUT returns you to the previous smaller-scale zoom window created using the "create window" command.

UNZOOM ALL returns the map to its original (default) scale.

GLOBAL PAN allows you to move the zoom window to a different area on the base map. You must already be zoomed into an area before selecting this option. A display box will appear showing the entire base map with a box around the area you are zoomed into. Click on the ESC button to leave the GLOBAL PAN mode without changing your window. Click on the PAN button at the top of the box to move to a new zoom area. Use the mouse to reposition the box and click to set the location. Your map display will then change to the newly selected position, remaining at the same scale as the previous zoom window.

PAN PT → PT re-orients the zoom window. For example, if an island appears in the lower left corner of the map and you want it in the center of the map, use this option to re-orient the window maintaining the scale. Click first on the point you want to move (in this case, the island). Then move the mouse to where you want the island to appear on the new display (in this case, the center of the screen) and click. The new zoom window will be displayed.

REDRAW is used to redraw the map, erasing any data displayed. REDRAW does not modify the scale of the displayed map.

When NRDAM/CME draws a new view of a location for the first time, it creates a file of that view (a *.PCX file) so that it can redraw rapidly. Thus, each new view will take noticeably longer to draw than a previously-drawn image. This applies, for each location, to the opening screen, to the full map view and any zoomed-in windows, and to insets such as in the wind module or the map inset displayed by the F9 key. Whenever each view is drawn for the first time it may seem to draw slowly. This also applies when the map colors are changed. Because color is an element of the *.PCX file, old files are erased when map colors are changed and each new view will require that a new file be created.

3.3 Selecting a File or Item

At several points you will be asked to select a file or an item. Two standard formats are employed for selection. Menu sub-options are displayed as a number of push buttons. Click on the button of your choice to select a sub-option, or click outside of the buttons to escape from the selection process.

A second format used when a file name/item is required is a window which appears listing the files/items available with the default option highlighted. To select one of the options, either click the mouse on the name of the file/item you wish to use or use the up/down arrows of the keyboard to move through the list. To leave the list without selecting a file/item, click the mouse on the ESCAPE box at the top of the list or press the ESC key.

In some selection windows, more than one simultaneous choice is possible. Clicking on an option will turn a checkmark on or off next to that choice to indicate its selection or deselection. The ENTER key may also be used to toggle selections on or off. Use the OK button on the form or the CTRL and ENTER keys on the keyboard to complete the selection.

To search the list for a particular file name/item, click on the SEARCH box at the top of the list. You will then be asked for a text string to search for. The search feature is not case sensitive so you do not need to reproduce the case of the string for which you are searching. Enter up to 12 characters and press ENTER from the keyboard to start the search. If the characters are not found, a message will appear to that effect. The search only takes place forward of the present location in the list, so you should first press the HOME key if you wish to start the search at the top of the list.

If the list contains more files/items than fit in the window, an up/down scroll bar with arrows will appear to the right of the names. Use the arrows to move through the list by clicking the mouse on either the up or down arrow. Each click of the mouse will scroll the list by one file/item. Alternatively you can use the up/down arrows, and the PAGE UP, PAGE DOWN, HOME, and END keys on the keyboard to move through the list. Note that the elevator between the up and down arrows to the right of the list will move as you scroll through the list to show your relative position. By positioning the mouse on the elevator and holding the left button down, the elevator can be moved to the top or bottom of the list. When the mouse button is released, the displayed list of files/items will orient itself with the elevator's position. This feature allows you to rapidly move to the top or bottom of the list (or anywhere between). Clicking the mouse in the sidebar above or below the elevator will page up or page down the listing.

3.4 Selecting a Number

A standard format is used when it is necessary to specify a number (e.g., number of days, time increment). A box appears in which the default value is shown. To the right of the default value are up and down arrows. Click on the up/down arrows located to the right of the default value to increase or decrease the time increment shown. Click on the OK button when satisfied with the number shown in the box.

Alternatively, the keyboard may be used to change the default value. Enter the desired number from the keyboard and press ENTER (or click on OK) to continue.

Whenever the model asks you to select a number, there will be a range of possible values. You will not be able to use a value outside these limits. If you enter a value outside the range, the computer will sound a warning tone, and you must enter a new value.

3.5 Viewing Text

A text listing facility displays text (such as system help, and model output summaries). A maximum of 25 lines of text are displayed at one time. For longer documents you can scroll through pages of text by pressing the PAGE UP and PAGE DOWN keys from the keyboard. Alternatively the mouse can be used to move through the text by clicking within the vertical bar to the right of the text. A box (elevator) inside the vertical bar indicates your relative position in the text. By positioning the mouse cursor on the elevator and holding the mouse button down, you can move the elevator up/down to move toward the top/bottom of the text. When you release the mouse button, the text will reposition itself to align with the relative position of the elevator. You can also click on the up/down arrows above and below the vertical bar to scroll through the text line by line. The HOME and END keys from the keyboard will bring you to the top and bottom of the text file, respectively.

Three options are selectable at the top of the text display form. The ESCAPE option is used to stop viewing text and return to the menu. Click on SEARCH to scan (forward only) through the document for a particular word or phrase. If you are in the middle of the document and wish to search for earlier occurrences as well, press the HOME key before selecting SEARCH. After you select SEARCH, you will be asked to supply the text string to be located. Once the text has been entered, press ENTER from the keyboard and the first occurrence of the string will be highlighted. To find the next occurrence, click on SEARCH and then press ENTER again to keep the previous search sequence. If the word(s) for which you are searching are not in the document, a message will appear to that effect. The PRINT option is used to send the entire text to the printer.

3.6 Use of Function Keys and Control Keys

The keyboard function keys are used in various menus within NRDAM/CME. The function keys are not active in the main menu. The function key capabilities may be accessed either by pressing the desired function key or clicking on the function key box at the bottom of the monitor display. The uses of the function keys are as follows:

F1 -HELP: accesses a help document specific to the module you are working in. Use the scroll and text search features described in Section 3.5.

F2 -DISPLAY OPTIONS: changes the options displayed on maps. A listing will appear displaying the options available for the map you are viewing. A check mark next to the option indicates it is turned on. Click on the desired options to toggle them on/off. Click on the OK box at the top of the list to exit, saving all checked options as the new defaults.

F3 -DISTANCE: determines the distance between two points. When you select this option, the cursor will change to a cross. Position the cross over the first location and click the mouse to fix the point. Then move the cursor to the second location and click the mouse once more. The distance between the two points on which you clicked will be displayed. The units of the measured distance are set under the SYSTEM → SYSTEM PARAMETERS option of the main menu to be either kilometers or nautical miles. Click the mouse a third time to erase the distance display.

F4 -COLOR KEYS: to display a color key. Select the key to be displayed from the list which appears. A box outline will appear on the monitor. Use the mouse to move the box to the desired location, and click to set the position. A key explaining the significance of the colors used will be displayed. Color keys may be repositioned by dragging with the mouse cursor. They may be removed from the screen by clicking in the button at the upper right. The list may include the following keys:

HABITATS.KEY - A key of all the habitat types that have been used for the current location.

HABNAMES.KEY - A key of all the habitat types that are available for use by the model.

TRJCOLOR.KEY - a key of the colors selected to display chemical surface and subsurface trajectories in the model output. These colors may be modified under SET DISPLAY COLORS in the OPTIONS submenus of the model output.

DEPTHS.KEY - A key of colors used to display depths in the Depth Grids (Key F8).

CONC-PPB.KEY - a color key for the CONCENTRATION display of the model output in terms of the most recently run chemical.

SPILMASS.KEY - a key of spillet mass for color-code spillets (when that option is selected) in the trajectory output.

ICETYPE.KEY - A key of colors used to display percent ice cover.

F5 -POINT LOCATION: to get the latitude (N-S) and longitude (W) location of the point indicated by the mouse cursor. The latitude and longitude are displayed at the lower right of the screen. Press the F5 key to get the location displayed in degrees and decimal minutes. Press F5 again to get location in degrees, minutes and seconds; and again to get decimal degrees. Repeatedly pressing the F5 key will loop through this sequence of display units. The location shown is the position at the center of the cross-hair cursor. Move the mouse to change the cursor's position. Click the left mouse button to label a location in degrees and decimal minutes. Click the right button mouse or press the ESC key to leave the location mode.

F7 -WORKING LOCATION: to change the current location. Select the desired location from the list of available locations which appears.

F8 -DATA DISPLAY: to display environmental data on the base map. Select the type of data to be displayed from the options which appear.

F9 -WINDOW LOCATION: to display an inset showing the whole study area with a box around the area you are currently zoomed into. The box may be moved by dragging it with the mouse and its size may be altered by dragging the button in the lower right corner.

F10 -PRINT: to send the graphic display on the screen to the printer or to a .PCX file. If you select the printer, a form will appear asking you to specify the type of printer along with a variety of other options such as image width and resolution. Click on the printer field or press ENTER to get a list of printers. If you select the .PCX file, a box will appear in which you must enter the name of the file to be created. It will be given a .PCX extension and be placed in the ¥NRDAMCME directory. To quit from either print function without printing, press the ESC key. This function only prints a copy of the screen image. Do not use it to print text files. When viewing text files on the screen, they may be printed by clicking on the ‘print’ button at the top of the text viewing window (Section 3.5).

These function keys are not enabled in some modules. If a function key is not enabled for the particular module in which you are working, you will get no response when pressing that key.

Ctrl Keys - Certain keys when pressed simultaneously with the Ctrl key allow the user to switch between NRDAM/CME modules or to view environmental data. Pressing Ctrl-F1 or Ctrl-H will list the functions accessible from these special keys. Note: Not all Ctrl key functions are applicable in all NRDAM/CME modules. If a ctrl key is not enabled for the particular module in which you are working, you will get no response when pressing that key.

CTRL-H	Listing of ctrl key functions (or CTRL-F1)
CTRL-G	Grids (habitat grids)
CTRL-D	Depths (depth grids)
CTRL-T	Tides & Currents
CTRL-M	Map Coastline
CTRL-K	Color Keys
CTRL-W	Switch to a different NRDAM/CME module.
CTRL-E	Text Editor
CTRL-X	Exit to DOS

3.7 Data Entry Forms

Throughout NRDAM/CME, forms are used for data entry. The forms contain a number of fields for entering data. Move between fields by pressing the ENTER or TAB keys to move sequentially. Click the mouse on a specific field to jump around the form. When you fill in one field, you will automatically be moved to the next sequential field.

Four types of data can be entered on forms, but each field accepts only one type of data. The four types are:

- character - any combination of letters, numbers and characters.
- whole number - any non-decimal number (integer).
- decimal number - any number (real), the decimal point is not necessary unless the value is less than 1.0.
- multiple choice - the choices are listed for you. Multiple choice fields are designated by a button with a triangle in it which appears only when you're in the field. When you click or press ENTER, a list of available choices will be presented, either as a scroll-bar list (Section 3.3) or as a push-button list.

Number fields have a range of acceptable values associated with them. If you enter a number outside the acceptable range, a beep will sound and you must enter a valid number before leaving the field.

At the bottom of the form, a help message will appear specific to the field you are in. It will prompt you for the type of data required. In some forms, additional information or help about the data expected will be available by either pressing the F1 key on the keyboard when the cursor is in the appropriate field or by pointing to the field with the mouse and clicking the right mouse button.

To exit from a form, saving all the data entered, click on the OK box at the top of the form, or press the CTRL and ENTER keys simultaneously. To exit the form without saving any of the data entered (i.e., keeping the initial values), click on the ESC box or press the ESC key.

3.8 Information Messages

Two methods are used to provide information about your status or deliver help or warning messages.

The bar across the bottom of the NRDAM/CME display generally shows buttons that allow you to access the function keys using the mouse. In some modules, however, the bar is used to inform you of the status of calculations or to prompt you through a process.

Information bars also appear within the map display to signal the end of calculations or to deliver warning messages. These bars will remain in place until you click the mouse or press any key to clear them.

3.9 Data Display Box

A data display box is used in many modules for displaying data additional to what is shown on the underlying map. The GLOBAL PAN and the F9 key window location option are examples.

Both the size and location of the box may be manipulated. To change its size, position the mouse cursor over the lower right corner button. Holding the mouse button down, drag the corner to expand or contract the size of the box. Release the button when the desired size is reached. To move the box, position the mouse cursor on the edge of the box. Holding the mouse button down, drag the box to its new location, and release the button.

To erase the data display box, click on the ESC button inside the box, click the mouse anywhere outside the box, or press the ESC key. Other buttons besides ESC may be used in the data display box depending on its application. If so, their functions are discussed elsewhere in this manual.

4. USING THE NRDAM/CME SYSTEM

4.1 Preparation

A series of steps are involved in setting up the model to simulate a spill and evaluate the resulting damages. Aside from actually installing the model system on a suitable computer, the first step a user must undertake is to collect all the relevant data to be used to set up the model. Important data inputs include a variety of types of information which will come from diverse sources:

(1)spill location

(2)date and time of the spill

(3)chemical spilled, including its physical and chemical form

(4)Volume and time-course of release

(5)Wind speed and direction during and following the spill

(6)Tidal and background currents over the area affected

(7)Amounts, times and locations of spilled substance removed from the water surface and shoreline by cleanup efforts

(8)Time of high tide nearest the spill time and the tidal range

(9)Air and water temperature

(10)Suspended sediment concentration and settling rate in the water column

(11)Implicit price deflator for the gross natural product for the data of the spill

(12)Closures of fishing, shellfishing, hunting, and beaches

(13)Ecological habitats throughout the area affected

While it is essential for the user to provide values for many of these inputs, the model system will function without some of them and will provide default values for some others. Users who wish to obtain a rebuttable presumption must consult 43 CFR 11 to determine the required extent and format of user-supplied information. With data in hand, it is now possible to start NRDAM/CME, create or edit the appropriate data files needed, and run a simulation.

To run NRDAM/CME go to the top level directory ¥NRDAMCME and type NRDAMCME. The main menu will be displayed across the top of the monitor display. Access to all the features of NRDAM/CME is provided through the main menu. A brief summary of the main menu is provided below; detailed information is given in the appropriate sections which follow. Options may be selected by clicking on the desired option, by pressing the highlighted letter, or by using the left/right arrow keys to move to the option (up/down arrows to move to a sub-option) and then pressing ENTER.

4.2 The Main Menu

NRDAMCME gives a brief overview of the NRDAM/CME system.

LOCATION allows you to select the area to which the model will be applied.

DATA handles data entry and editing. Data such as the model grid, currents, etc. can be displayed by pressing the F8 key while in most map modules.

RUN MODEL controls model setup and execution. Sub-options allow you to select the mode you want to run.

OUTPUT DATA controls viewing previously run spill simulations.

SYSTEM allows the user to specify system-wide parameters, and perform file operations.

HELP provides help on common user interface features and functions found throughout the NRDAM/CME system.

QUIT leaves the model system and returns to DOS

Each of these options is described in more detail in section 4.4. It will help to understand these details by first briefly describing the whole process of setting up and running a model simulation.

4.3 Setting up and Running a Case

The system is up and running, with the main menu on the screen. Click on LOCATION and select the geographic region where you will be working. Most data types in the model are location-specific, so it is important to carry out all work in NRDAM/CME with the location in which the spill occurs selected. For instance, a wind file created in the east coast location will not be available for use in a west coast simulation.

A simulation cannot be run without a time-specific wind file. The next step is to select WINDS from the DATA submenu. Here, select **New wind**, and enter the wind data for the time of the spill event, creating a file that will be selected later when setting up a spill scenario. The details of this process are discussed below in Section 4.4.3.1. Creation of a wind file is quite simple once the data are acquired. If the wind record is very long, it may be more practical to enter the wind data into an ASCII text file and use the wind module to import the data into the format used by the model. This is discussed in Section 4.4.3.1 and the format for the ASCII file is shown in Appendix A.

The next step will be to create a file containing background and/or tidal current data for the area of the spill. This is not essential (i.e., the model will run without a current file) but if there are any known currents in the area, they should be included as part of the simulation. Select MAIN MENU, then DATA → CURRENTS, and the currents utility will be loaded. You may now proceed to create a current grid and enter the current data. This is discussed in Section 4.4.3.2. Creating good current files may require more practice than wind file creation. A number of steps are involved and more care needs to be taken to ensure that data are being entered accurately. Note also that in most cases, good current data will be much more difficult to acquire than wind data. Sources of current data are discussed in Appendix C of this manual.

If specific data were collected on closures of fishing, shellfishing or hunting areas, or beaches, then now is the time to enter that data. From the main menu, select DATA → CLOSURES. This is a simple procedure, clearly directed by prompts in the closures module. It is described in Section 4.4.3.3.

NRDAM/CME contains an extensive habitat database which is contained in a grid system, which is the actual domain within which the model runs. There is no need for the user to enter habitat data to run a model simulation as the data has already been entered, however, many assumptions were made in the creation of the existing database. Users with detailed local knowledge of an area may find that specific areas of the habitat database do not agree with the known habitat distribution. In that case, it will be desirable to edit the habitat grids. In any module except winds, zoom in to the area where the spill occurred (Section 3.2) and view the habitat grids using the F8 key: Habitat Grids → Habitat Data. The habitats will be displayed with each distinct habitat type represented by a different color/pattern. Display the key to habitat types using the F4 key: HABITATS.KEY. If the habitats in the existing database are inconsistent with the known habitats in the area, use the habitat editor to edit them. From the main menu, select DATA → HABITAT EDITOR. Refer to Section 4.4.3.4 for a description of the use of the habitat editor.

All of the remaining setup steps take place in the Run Model module. From the main menu, select RUN MODEL → PHYSICAL FATES. If cleanup efforts removed significant quantities of spilled chemical from the water surface or the shoreline, then the next step is to create a cleanup file. Select **set Cleanup** from the side menu and proceed to enter cleanup data following the instructions in Section 4.4.1 under CLEANUP. Cleanup files will store information separately for water surface and shoreline cleanup and specific to as many time and space windows as the user specifies, so it is useful to get detailed cleanup information.

The next step is to tell the model system where the spill occurred. There are two ways to do this: locate the spill site with the mouse, or enter the latitude and longitude into the Spill Information form. Where the spill is close to land, enter the spill location using the mouse. Display the habitat grids (F8: Habitat Grids → Habitat Data) while zoomed into the area of the spill. The model sees these grid cells, not the shoreline that is normally viewed to help orient the user. Thus, the spill site should be located relative to the habitats rather than to the shoreline. (It is sometimes helpful to draw the shoreline over the habitat grids, using CTRL-M, to help make clear what is represented by the grids.) Select **spill Locat.**, place the cursor over the spill site and click.

The final steps will be to fill out the Spill Information form and start the model. Select **set Scenario** and the Spill Information form appears. Enter a name for the scenario to be run, enter the latitude and longitude of the spill site if it has not already been selected with the mouse and enter the data and time of the spill. Also, give a description of the scenario, the volume spilled (and units) and the duration of the spill, and the time of high tide. From lists, select the chemical spilled, the winds, currents and cleanup files to be used and whether or not to use ice and to follow the fates (trajectory and weathering) calculations with the biological calculations. Section 4.4.4 describes this in more detail. When the form is filled out, click on OK or press CTRL-ENTER and the Environmental Variables form appears. Enter temperature and suspended sediment data (use the defaults if these data are lacking) and click on OK (CTRL-ENTER). Enter the appropriate economic factor in the next form, then indicate whether or not to use a closure file and select the file if so. When asked if you wish to run the model now, click yes (press Y) and the fates calculations will start. These calculations vary widely in the time to completion. Runs from beginning to end, with biology and economics, may take as little as 10 minutes or in excess of 24 hours. This will depend on your hardware, the chemical spilled, and the size and location of the spill. When calculations are complete, the program will return to the NRDAM/CME graphic interface in the OUTPUT DATA module.

To view the output from the model run, from the main menu select OUTPUT DATA → MODEL RESULTS (if you are not already there). Click on **Scenarios** and select the scenario name of the case to be viewed. Zoom into the area of interest. To view displays of the distribution and fate of the spilled chemical, select **Fates**. Here, you can view graphical displays of the spillet (surface slick) trajectory, dissolved chemical distribution, or concentrations of dissolved chemical in the water column or the sediment. There is also a text record of the results of the fates calculations over the time course of the spill. To view text files of the biological impact and damages calculations, with or without restoration, select **Bio/Econ**. Several levels of detail are available. Select **Assessmnt Rpt** to create a file containing all the inputs and text summaries of the model run which serves as a documentation of the scenario.

The above discussion only gives a quick overview of the steps involved in setting up, running and viewing a spill scenario. There are many details and options along the way which have not been discussed. Section 4.4 describes each of the Main Menu options in more detail with instructions on how to use each of the modules and how to manipulate the graphical interface to provide the most information of interest to the user.

4.4 The MAIN MENU Options

4.4.1 NRDAMCME

Selecting the option NRDAMCME → ABOUT NRDAMCME will bring up a text file giving a brief overview of NRDAM/CME's capabilities and usage. View the text as described in Section 3.5.

4.4.2 Location

Select the option LOCATION → CHANGE LOCATION to get a list of the geographic areas provided with NRDAM/CME. The location you are currently working in will be highlighted. Select a new location as explained in Section 3.3 or ESCAPE to leave your location unchanged.

4.4.3 Data: Entering and Viewing Data

Several sub-options are available under DATA which allow you to enter or modify environmental data for model simulations. These options are described in the sections below. To view other data used by NRDAM/CME such as the grids, currents, or habitat types, press the F8 key from any map module. You will then be asked to select the type of data to be displayed and for the file containing the data.

4.4.3.1 Winds

To access the wind entry option select DATA → WINDS from the main menu. The wind entry option allows you to view existing wind data in text or stick plot form, create new wind records, and revise existing wind records. Wind data are saved specific to the location you were in when selecting the DATA → WINDS option. This location can be changed with the use of the F7 key when you are in the winds option. The location is shown in the lower right corner of the monitor. Appendix B contains a brief listing of sources of wind data.

Note:

1)Winds are specified using the meteorological convention. That is, you specify the direction from which the wind is coming. The wind series is displayed in a stick plot form at the bottom of the screen in an oceanographic convention, showing the direction toward which the wind is blowing. The color of the stick plot vectors is purple for winds with any westerly component, yellow for winds with any easterly component, and red for winds due north or south.

2)A wind is time-specific; the time period covered by the wind record must correspond with that of the spill in which it is used. If a wind record ends before a spill scenario is complete, the model will calculate stochastic winds to complete the scenario using values from the environmental database as mean values for speed and direction.

There are two basic approaches to creating new wind files. One is to enter data line by line within the wind entry screen. The other is to enter the data into an ASCII text file and use the wind entry module to import that file into the format used by NRDAM/CME. The latter approach will be discussed later with the other ASCII options.

To enter wind data using the wind entry screen, select **New wind**.

- Complete the form which appears, specifying the wind file name, a description (optional), and the date the wind record starts. Help is given at the bottom of the form. Click on OK to continue, or on ESC (or press the ESC key) to return to the sidebar options.

  - In the next form, choose the hourly time increment: each wind value you input is assumed to occur for this length of time. The allowed increment range is between 1 and 48 hours. Enter the desired value as explained in Section 3.4. Click on OK or press ENTER to select the value shown.

Notice that the buttons on the side menu have changed and that the fields in the windows on the left side of the screen now contain values. You may enter wind data using either the mousse or the keyboard:

  - Mouse - move the cursor arrow inside the sectored circle. Wind direction may be specified in 10 degree increments. Wind speeds of 0 to 30 knots are specified as the mouse is moved away from the center of the circle. The speed and direction corresponding to the arrow's position on the circle are shown in the upper left corner of the monitor. When you have the desired speed and direction, click the left mouse button to enter the data into the wind file.

  - Keyboard - select **Manual** from the sidebar. A form will appear on which to enter the direction and magnitude of the wind. Direction is input in degrees from North. The wind speed is in knots (maximum 120 kts). Use the keyboard to enter the desired values in each field; click on OK or press the CTRL and ENTER keys simultaneously to add the entered data to the wind file for one time increment.

Continue this procedure until all the wind data have been entered. Then click on ESC or press the ESC key to return to the sidebar options.

Note that, for either entry method, the current time in the wind file is updated in the lower left window and a stick plot representation (in oceanographic convention) is displayed at the bottom of the screen. When all the wind data are entered, click on **Save** to save the file and return to the opening wind menu. You may at any time click on **No save** to escape from data entry to return to the opening wind menu. All data entered will be lost.

While entering wind data, several editing options exist as buttons on the side bar:

Increment - to change the time increment for entering data. This may be changed at any time during an editing session, whether entering or altering wind data.

Rewind - to move backwards in time in the wind file. Each selection of the REWIND button will erase the last record in the wind file spanning the selected time increment.

Forward - to move forward in time in the wind file. Each selection of the FORWARD button will move forward in the wind record by the selected time increment. You may only move forward as far as you have rewound.

Append wind - to add data from an existing wind file to the end of the working file. Select the desired wind file. Only data in the selected file which fall after the last time entered in the working file will be appended to the working file.

In order to edit an existing wind file, select **Edit wind** from the opening wind menu.

- Select the wind file you wish to view/edit from the list of existing wind files which appears. Click the mouse on the desired file name. The wind data in the file will be displayed in stick plot form.

- Choose the time increment (1 - 48 hrs) in which data are to be input and edited.

You can now add to or revise the data in the file either using the mouse or manually, as described above. All of the options described above, such as **Rewind**, **Forward**, and **Append wind** may be used to edit the wind file. When done editing, click **Save** to save the edited file or **No save** to escape without altering the file.

The final option available on the opening wind menu allows a variety of ASCII manipulations of wind files. Select **Ascii** and the following four options appear in a submenu:

List Wind File - allows you to list a wind file in text form. When LIST WIND is selected, a window containing the available wind files appears. Click on the desired file. A window will then appear which displays the wind data in text form. View the data as explained in Section 3.5.

Import Ascii Wind - puts external wind data in ASCII format into the binary format used by NRDAM/CME. The ASCII file of wind data must have an .ASW extension and be in the NRDAMCME¥CONVERT directory. The expected format of the file is given in Appendix A.

To use the **Import Ascii** option, select the ASCII wind file name (.ASW extension) from the list that appears. Once you have selected a file, a window will display some information about the file, including the first few lines of wind data. You will be asked to select the units in which the wind data were entered into the ASCII file. The ASCII file utility can import wind data in units of knots, miles per hour or meters per second. A new form then appears on which to specify the filename that will be stored in NRDAM/CME format. The name can be any 8 character name, including the same name as the ASCII wind file. The new file will automatically be given a .WND extension.

Export Ascii Wind - puts the internal binary wind data format into ASCII format for export from the NRDAM/CME system. Winds can be exported in units of knots, miles per hour or meters per second. The file will not be labeled with units so it is important to remember and note which units were selected.

Edit Ascii Wind - provides a listing of an ASCII wind data file and allows modification of the data.

The following function keys are operable in the DATA/WINDS menu, and function as described in Section 3.6:

F1 -displays help text.

F7 -changes the current application area. Wind data are stored specific to the location for which they were created.

F10 -allows you to print the screen display on your selected printer.

4.4.3.2 Currents

From this module you can enter currents to be used by NRDAM/CME's spill trajectory and fates submodel. Both tidal currents (time varying) and background currents (time invariant) may be input. All currents are spatially variable. The process of current file creation has been simplified to allow rapid, easy generation of a current field without requiring the user to understand all the underlying details. Sufficient flexibility is incorporated into this process to allow the user to modify the resulting files as needed to obtain the required final result. On-screen prompts guide the user through this process. Refer to Appendix C in this volume for a discussion of sources of current data.

The steps involved in creation of a currents file are:

1) Set up the screen image.
2) Create a grid and edit it, if needed.
3) Draw background current vectors, and edit them
4) Draw tidal current vectors, and edit them.

On the Main Menu, under DATA, select CURRENTS. The data entry process will be simpler if, at this point, the screen and display options are set up appropriately. Use the **Zoom** option to set the screen view to the area of interest, a view just large enough to encompass the entire current field. Next, under **Options**, select **Set Scaling/Color** and adjust the scaling of the current vectors to a size consistent with the magnitudes of the currents to be entered. This may be different for background and tidal currents; adjust the scaling for whichever you will be entering first. The procedure for adjusting vector scaling is given later in this section. Making the above adjustments requires a certain amount of experience with the use of the current generation utility and with the effects of currents on the results of spill simulations.

To initiate the process, select **Grids** from the side menu and select **New Current Grid**. Provide an appropriate file name at the prompt (up to eight characters, no extension). You will then be prompted to outline the grid frame, i.e., the area within which currents will be created. Click on the opposite corners of this area with the mouse. The area will then be divided up into grid cells of appropriate size, with the land and water areas defined. Land cells will be dark red and water cells will be blue (or whatever water color is selected under SYSTEM/SET MAP COLORS). This process may take several seconds to a minute or more. Check to see that water and land are properly identified. It is normal for the shoreline edges to be somewhat ragged, and that will be fixed during editing, discussed below. If very large areas of water are displayed as land or land as water at this point, however, it may be simpler to define the grid area again as a new grid.

The appropriate area of coverage for the current grid should be as small as possible to get the best grid resolution (and minimize the work invested in creating it) but it must cover the entire area affected by a spill. If a spill's trajectory carries it across the edge of the current grid, you will get anomalous results at the edge.

Note that the gridding process defaults to a minimum grid cell size, of the order of the habitat grid cell size. If too small an area is outlined as the grid frame, the process may have difficulty defining the shoreline and distinguishing land and water. If you only get a small number of grid cells, with odd or incomplete gridding, and typically accompanied by the message "Current grid generation not complete," then restart the grid generation process, this time selecting a larger grid frame.

When the initial grid generation process is complete, you may wish to edit the grid. If land or water cells appear inappropriately represented, select **Grids--Edit Current Grid**. It will probably be helpful to zoom in first to better see details. If there are narrow passages in your gridded area, you will probably need to edit the grid. Simple shorelines may not require it. If you elect to edit the grid, click the mouse crosshair in each grid cell you wish to change. Clicking in a grid cell toggles its status between land and water, and may be changed repeatedly. Press escape, or click the right mouse button when done editing. You will then be prompted to save the edited grid; just click on YES (or press Y). The process of saving the edited grid applies whether or not current vectors have been entered. For this reason, you will get messages on the screen that currents are being spread and saved even though you have not yet entered any currents.

When you have completed grid preparation, select **Vectors** to enter current data. Select **Background Currents** or **Tidal Currents**; you will be given the option to choose the other later. You are now ready to enter current data into the grid. See Appendix C for suggested current data sources.

BACKGROUND CURRENTS are time invariant. Use the mouse to position the cross-hair at the base of the current vector and click. As you position the mouse, the latitude and longitude of the cursor will be displayed at the bottom of the screen. The first click, at the base of the vector, is the actual location of the current being entered and will be located at the center of whatever grid cell the cursor is in when clicked. Move the mouse to set the direction and magnitude of the vector. Values of U (east-west component), V (north-south component), magnitude and direction (degrees from north) are shown on the information bar at the bottom of the screen. Enter as many vectors as desired in this fashion. Click the RIGHT button or press the ESC key to stop entering vectors. Magnitudes are given in units of cm/sec or knots (nautical miles per hour), depending on the setting of the units in SYSTEM → SYSTEM PARAMETERS. Units may also be reset in CURRENTS by selecting **Options → Display Options**. Because magnitudes are given to one decimal place in either unit, it is recommended that current entries always be in metric units (converting from knots if the data source is in knots) in order to take advantage of the better resolution.

TIDAL CURRENTS are time varying with a tidal period. Select the type of tide. Typically either M_2 (semi-diurnal, 2 high tides a day) or K_1 (diurnal, 1 high tide a day) is dominant. Draw the current vectors at maximum flood tide as described above. The tide is assumed to be rectilinear (ebb is opposite of flood).

As with grid editing, it may be easier to enter current vectors if you zoom into smaller areas first. With both of these functions, you can zoom in, edit or enter data then move the zoom window to another area and recommence data entry. It is necessary to leave the data entry/edit mode to move around, but no data is lost in the process.

When you are done entering, the currents you have entered will be spread over the whole water area of the grid, interpolating and extrapolating from the vectors you have entered. The "user vectors" you have entered will remain in red, or whatever color has been set under **Options** → **Set Scaling/Color**, while the spread vectors will be green. If the current field does not now appear as you would like it, or you wish to enter more vectors in another zoom window, simply reselect the current component you have been entering (background or tidal; it will be indicated on an inset on the **Vectors** button) and add to or alter the vectors present. Spreading of the current field will occur each time you escape and files will be saved whenever you leave edit mode.

Spreading occurs from the user vectors you enter. For a simple uniform current field, a single user vector will suffice. Spreading will not occur across land. For more complex current fields, it will be necessary to enter more vectors, and to view and edit the results of the spreading to ensure that a sensible current field results.

When you have completed entering data for the first current component, select the other component under **Vectors** if you wish to represent both current types in your simulation. As before, when done, press escape or the right mouse button, and the currents will be spread and saved under the same file name as the first component.

The current creation process is now complete. The currents file created may be used in a model run. If the results of the first file prove unsatisfactory for any reason, the file may be re-edited at a later time. When selecting water **Grids**, however, select **Old Current Grid** , then select the file name from the list that appears. You may then proceed to edit background and/or tidal vectors just as was done in the original data entry. If you wish to keep the old current field, but use it as a template for a new current field, then select **Save as** after opening the old file. Give a new filename at the prompt, then proceed to edit the new file. See the description of **Options**, below, for a description of some tools for making large overall changes to existing current fields.

Note that when you edit a file, vectors will be drawn in two colors. The vectors input by the user will be drawn in the color selected under **Options** → **set scaling/color**. The vectors that spread over the remaining grid are drawn in green. It is useful to set the user-input vectors to a color that contrasts well with green, and with the background, so that they are easily distinguished.

When you edit a file, you may both alter previously entered vectors and enter new vectors. In any event, only the user-input vectors are spread to create the remaining interpolated vectors each time you complete data entry.

OPTIONS - The Options button offers four choices for modifying the display or the current data.

Rescale all vectors - Allows you to uniformly increase or decrease all the vectors in a grid by a fixed factor. This affects all the vectors of the displayed component only; it does not alter the other component. This will overwrite your old data. If you wish to use this option to create new currents without losing the old data, you must first select **Save as** to save the old currents, and create a new file name for editing. At any time, the name of the file you are currently working on is given on the inset in the **Grids** button.

Reset all to zero - Allows you to clear all the vectors, both user-entered and spread, in the currently displayed field to zero. It does not affect the other component, It is useful for starting over with new currents on an old grid. It will not prompt you for a new file name. Be sure to save the old grid under a new file name first if you wish to keep the old data.

Set scaling/color - This option affects the display of current vectors, not their values. The form which comes up on the screen has three fields. "Current Vector" is the current speed, in cm/sec, represented by the scale vector displayed on the lower right of the screen. "Corresponding Pixel Length" is the length of the scale vector that will be displayed (in pixels). All current vectors will be displayed using this ratio of pixels to current speed. It will be useful to alter this ratio for entering and/or displaying widely varying current speeds. It may be altered any time some other function is not active, so you may change scaling between selecting a file and editing it, or between entering tidal vectors and background vectors, but not in the middle of a data entry operation. The "Current Vector Color" entry affects the color of the scale current vectors and the user-entered vectors, but not the spread vectors. It will be most useful to set this to a color which contrasts with the green spread vectors and with the map color.

Display Options - This option offers two display choices: UNITS allows you to select whether display units should be in nautical miles (and knots) or kilometers (and cm/sec). DISPLAY HYDRO GRID allows you to turn off the grid display to view currents, once created, without the background grid. Creating a new grid will automatically reset the grid display to ON.

4.4.3.3 Closures

This utility allows you to create a file which records closures of harvest and recreational areas. You will be led through a simple series of lists and forms as described in the following section. You will first be asked to provide a name for the scenario for later selection when running the model.

Closures are indicated for each province separately. A list of all provinces in the present location will be displayed. Check all the provinces which had closures by clicking on them. A check will appear after all the chosen provinces. Then click on OK. You will be given the chance to indicate closures for each chosen province in turn. Notice that the province you are currently working on is written at the top of the screen.

Closures may occur in one or more of the following types of areas:

harvest areas:
- Seaward fish habitat
- Landward fish habitat
- Structured fish habitat
- Seaward shellfish habitat
- Landward shellfish habitat
- Structured shellfish habitat
- Waterfowl hunting area
- Mammal hunting or trapping area

recreation areas:
- National beaches
- State beaches

Habitat types are defined in Section 6 of Volume I. Refer to that text for clarification of how the different types are distinguished. Definition of beach types is discussed in Section 10 of Volume I.

Harvest area closures are entered in days and square kilometers. The model will use these inputs to calculate a product of the km^2-days. When the amount of closed harvest area varies from one day to the next, the user should calculate the total km^2-days of closure, enter this value in the area field and enter one (1) day.

For each chosen recreation area, choose all the months in which there were closures, and a form will appear for each chosen month. Beach closures are given in days and linear kilometers of beach. The model simply calculates the product of km-days from the user-input data. For situations in which variable lengths of beach are closed within a month, the user should calculate the total km-days of closure for that month, enter that value into the kilometers field and enter one (1) day.

When closure data entry is completed, you will be given the opportunity to view the closure file. Select YES and verify the entries that are recorded in the closure file. You may also view any closure file at a later date by selecting **View files**. There is no utility for editing a closure file within NRDAM/CME. If entries are not correct, or you wish to modify them, create a new file and reenter all the data.

4.4.3.4 Habitat Editor

NRDAM/CME includes a complete habitat database for the area of coverage of the model system. This database consists of a system of 100 x 100 cell grids in which each of the cells has been assigned a habitat type. These habitats may be viewed in most modules of the model by selecting F8:DATA → Habitat Grids → Habitat Data. It is recommended you zoom into the area of interest first; drawing all the habitat data for a large area may take a long time. With the exception of a few areas in which good quality habitat data were available and were entered into the habitat database, most of the habitats were created using a variety of default assumptions. Thus, it is likely that most users of the NRDAM/CME will find it useful to be able to edit habitat grids in the areas of interest in order to produce a more realistic environmental setting for the model runs. The habitat editor allows the user to alter the habitat designation for the grid cells in the model according to user inputs. It is accessed as a sub-option under DATA in the Main Menu.

The logic of the habitat editor is as follows. The NRDAM/CME reads habitat grid files located in the NRDAM/CME directory structure as:

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥P#.HAB

where # = a four digit identifier for the grid, the first two digits being the province number and the second two digits being the grid number within the province.

With the habitat editor, the user may change the habitat assignments in these habitat grid files. The user may also restore the original habitat grid, as distributed with NRDAM/CME. The original files are located in the directory:

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥ORIGINAL

These original files are not altered by the habitat editing tool. However, the working files that the model reads (P#.HAB) will be whatever the user has currently saved. If edits are made and saved, these edits will be used by the model. If the original grid is selected and saved by the user, the model will use the original habitat data. Care should be taken to be sure of the version (i.e., the user's edited grid or the original grid) that the model is currently using. The habitat editor offers the user the option of creating and saving several different copies of a grid which may then be recalled at some later time.

The user may also create text (ASCII) files of the habitat grids. (The working P#.HAB files and the original files are in binary format and cannot be viewed directly.) The text files may be viewed in DOS or in a word processor, or printed from DOS.

The text files are located in the following directory, named as indicated.

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥ASCII¥P#.ASH

At the same time, a second file, with a .ASD extension is created containing the corresponding depth data (in meters) for each grid cell. The format of the text files is simply to print the habitat codes (Table 6.3 of Volume I) for the grid cells in a 100 x 100 matrix in order of west to east from left to right, and north to south from top line to bottom line. This file is 80 columns wide. The habitat text file is split into four groups of 100 rows each: 1-25 (western-most cells), 26-50, 51-75, 76-100 (eastern-most cells) in that order in the file. The corresponding depth data are split into seven groups: 1-15, 16-30, ..., 91-100.

In making changes to the habitat grids using the editing functions, the user may reassign shoreline cells to another shoreline habitat type or water cells to another water habitat type. The user may not change the location of the shoreline, however. Thus, land cells may not be modified, shoreline cells may not be changed to water and water cells may not be changed to shoreline. Shoreline habitats are defined as "fringing" habitats (F) in Table 4.1. Water cells are either subtidal or extensive wetland or extensive mudflat (noted by W in Table 4.1). The user may not use the editing function to change the seaward versus landward designation of the grids. (See Table 4.2) These restrictions on habitat editing are made because the biological abundances and economic valuations underlying the NRDAM/CME would be affected by the prohibited changes.

An important aspect of the original habitat grids is not immediately evident upon first viewing most of these grids, and this can cause some confusion. Most grids appear to be 50 x 50 cell grids when in fact they are 100 x 100. This condition results from the fact that the original grids were created as 50 x 50 cells grids. They were later subdivided into 100 x 100 cell grids to increase the potential resolution of the habitats and of the model. When you commence editing it will become apparent that each of the larger blocks in most grids is really composed of four grid cells which can be edited individually.

To use the habitat editor:

- Select DATA from the main menu, and then HABITAT EDITOR from the submenu.

- Choose **Select grid** from the side menu, and labeled outlines of the habitat grids will appear on the screen with a listing of available grids. If the screen view is zoomed in too close, you may not see any labels or grid outlines. The list of grids may be moved by dragging with the mouse to see behind it. Select a grid from the list, and habitats for that grid will appear on the screen. Adjust the zoom window view if necessary.

- A key of habitats used in the present location (HABITATS.KEY) may be viewed by selecting the F4 keyboard key or clicking on F4 with the mouse. The HABNAMES.KEY is a more extensive listing displaying all of the habitat types available to the model.

- Select **Edit grid** and select a habitat to which you wish to change a grid cell. Note the rules above: if you are editing a shoreline type of habitat you must select another shoreline type; if the existing habitat is "seaward", the replacement type must be seaward as well, etc. Click on the cells to be changed (you can hold down the mouse button and "paint" in areas) and the new habitat type will appear. If you click on an inappropriate cell type you will hear a small beep. Moving the cursor into the menu area, or pressing ESCAPE will terminate editing of the chosen habitat type and you may select another by reselecting **Edit Grid**.

- When done editing, select **Save**. A submenu will appear. Select **Active Grid** if you wish to save the edited grid for use immediately in a model run, then confirm that you wish to overwrite the existing active grid by selecting "Yes." Select **Backup grid** if you wish to save the edited grid for future use but do not presently wish it to serve as the active grid. Provide a two-character identifier, when prompted, to identify the file for later use. This file will be saved as:

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥USER¥P#.Unn

where 'nn' is the user identifier. Save twice if you wish to use the edited grid as the active grid but also retain a copy which will not be overwritten by some future replacement of the active grid.

Table 4.1 Classification of ecological habitats. Seaward (Sw) and landward (Lw) system codes are listed. (Fringing types indicated by (F) are only as wide as intertidal zone in that province. Others (W = water) are a full grid cell wide and must have a fringing type to landward.) (This is Table 6.3 in Volume I.)

Ecological Habitat Code (Sw,Lw)	Zone	Ecological Habitat	Tropic Habitat Code (Table 4.4)	Fish and Shellfish Habitat	(F)/(W)
1,31	Intertidal	Rocky shore	1	Open	F
2,32		Gravel beach	2	Open	F
3,33		Sand beach	3	Open	F
4,34		Fringing mud flat	4	Open	F
5,35		Fringing wetland, saltmarsh or mangrove (by province)	5	Structured	F
6,36		Macrophyte bed (algal - provinces 1-21; 40-74; seagrass-provinces 22-39, 75-77)	6	Structured	F
7,37		Mollusk reef	7	Structured	F
8,38		Coral reef	8	Structured	F
9,39	Subtidal	Rock bottom	9	Open	W
10,40		Gravel bottom	10	Open	W
11,41		Sand bottom	11	Open	W
12,42		Silt-Mud bottom	12	Open	W
13,43		Wetland (Subtidal), saltmarsh or mangrove (by province)	13	Structured	W
14,44		Macroalgal bed	14	Structured	W
15,45		Mollusk reef	15	Structured	W
16,46		Coral reef	16	Structured	W
17,47		Seagrass bed	17	Structured	W
18,48	Intertidal	Man-made, artificial	18	Open	F
19,49		Ice edge	--	None	F
20,50		Extensive mud flat	4	Open	W
21,51		Extensive wetland, saltmarsh or mangrove (by province)	5	Structured	W

Table 4.2Explanation of systems for landward and seaward portions of a province, by type of province. Fishery species abundances vary by landward open water, seaward open water, and structured (wetland, bed or reef) habitat. Seaward area is always the major portion of the province. (This is Table 6.2 in Volume I.)

Code	Province Type	Landward	Seaward
1	Rivers	Upper reaches	Lower reaches
1a		tidal fresh (<0.5‰)	estuarine = mixing zone
1b		estuarine = mixing zone (0.5- 25‰)	seawater (>25‰)
2	Estuarines, bays, harbors, sounds	Rivers, inlets	Open bay or sound
2a		estuarine = mixing zone	seawater (>25‰)
2b		estuarine = mixing zone + tidal fresh	seawater (>25‰)
2c		tidal fresh	estuarine = mixing zone + seawater (>.5‰.)
2d		estuarine = mixing zone inside major constriction	mixing zone + seawater outside major constriction
3	Coastal (where includes shoreline and entire province is <200 m)	Inlets, bays, coastal ponds, river mouths (estuarine = mixing zone + tidal fresh if significant area + seawater inside major constriction	Open coastal water <200 m deep = shelf (marine = seawater)
4	Offshore (where part of province is >200m)	shelf (<200 m) (marine = saltwater)	oceanic (>200 m) (marine = seawater)

- If you wish to replace the active grid with the original habitat types or with some other edited grid, select the grid you wish to restore, then select **Restore grid**. Choose either **NRDAMCME grid** to replace the active grid with the original habitat types or choose **Backup grid** to replace the active grid with a user-created grid. In the latter case, choose a file from the listing of available backup grids which is presented. Verify that you wish to do so by selecting "yes". If you are then satisfied, upon viewing the replacement grid, that it is as you want it (you also may edit it at this point if you choose) then select SAVE to overwrite the old active grid with the replacement grid.

- To create text (ASCII) output files, select **Text Output**. The resulting text files will be found in the subdirectory

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥ASCII¥

as files P#.ASH and P#.ASD. A message will appear on screen describing the name and location of the habitat text file just saved. Press ESC or the right mouse button and a message with the name of the depth text file will appear. Escape again and the text file creation is complete. Note that the name assigned to these text files is fixed for any given grid. If you wish to save text files of more than one version of any single grid, you must rename the first set (in DOS) before saving the next.

4.4.4 Run Model

4.4.4.1 Physical Fates: Specifying the Spill Scenario

From the Main Menu, select RUN MODEL, then PHYSICAL FATES to view the Physical Fates screen. The primary function for this module is to prepare a spill scenario and start it running. It also contains a submodule for preparing cleanup scenario files. In this section, we will first discuss cleanup, how to prepare a file and how the model uses that information. This will be followed by discussions of setting the spill location and setting the spill scenario. Finally, there will be a brief discussion of how to select an oil type for oil and petroleum product spills.

CLEANUP

The options under **Set cleanup** allow the user to create, modify and view text files of cleanup scenarios for incorporation into chemical spill scenarios. A file may contain several areas, within each of which there may be several cleanup periods with specified volumes to be removed. Cleanup is specified as occurring from the surface or from the shoreline within each area. The cleanup option works as follows when a cleanup file is specified to be used in a fates run: Within each area and time period, the amount of chemical specified in the cleanup file will be removed from the surface or shore only up to the amount available in that area and time window as predicted by the model. Since removal will occur immediately upon becoming available, a series of small removals over short time periods (such as daily removal) will better simulate the effects of cleanup than will larger areas and time steps. Note also that if the fates model does not predict that the chemical will be present within the space and time specified in the cleanup file, no chemical will be removed in the simulation.

A series of steps is involved in generating a cleanup file, and an incorrect sequence may create an error not obvious to the user. It is therefore recommended that the text file of a finished, saved file should be viewed to verify the data entry before using it in a spill scenario. To prepare a cleanup file, select **set Cleanup**, then **Add Cleanup Loc.** Crosshairs will appear which are used to outline an area for cleanup. In the form that appears, specify whether cleanup is to occur from the shoreline or the water surface (the area must be defined separately for each of these options). Add the optional location name and description, if you choose, then click OK (^ ENTER). Specify the start and end times (days and hours since spill) in the next form that appears along with the amount and units. Only units of volume are available in the cleanup option. If cleanup data are available in units of mass, they will have to be converted. Click OK and enter the data for the second period. The period for which you are entering data is noted at the bottom of the screen. After you have entered data for the last period and clicked OK, click ESC (press ESCAPE key) and the crosshairs will reappear to specify a new area. You may enter as many times for an area as you like but each period within an area must be later than the previous period, without overlap. When you have completed all the areas to be specified, press ESCAPE or the right mouse button when the crosshairs appear. Select **Save** and give the new cleanup file an appropriate name (no extension).

The **Turn Off Cleanup** option may be used to reset the cleanup utility in order to create a new file. If a file is currently displayed, any areas added and saved will be appended to the data in that file. Using the turn off cleanup button removes the current file from memory so that newly added areas do not include these old areas in the saved file.

You must use **Save Cleanup** to save a cleanup scenario, giving it an appropriate name (eight characters or less, no extension) before leaving **Set Cleanup** If you do not, and you have entered new information, you will be prompted to save. Cleanup files are saved with an .LRF extension in the RESPONSE subdirectory for the present location.

The **Get Old Cleanup** option opens a previously-created cleanup file and displays the outlined areas in the file. The user may then add new areas and specify additional cleanup for a scenario. Save the changed file when done. There is no option for editing existing cleanup files beyond this ability to append additional locations.

View Text File presents a text listing of a selected cleanup file. This is the best way of verifying the contents of a given cleanup file, and it is recommended that all files be checked using this utility before using them in a scenario. Viewing a text file does not alter the current file in **Set Cleanup**. If you wish to alter a file after viewing the text file, you must reselect it using **Get Old Cleanup**.

In addition to the considerations of how the model interprets cleanup files, discussed at the beginning of this section, one additional point needs to be made. The volumes specified for cleanup are all assumed to be pure chemical. The area in which this may present the most difficulty is with the oils, many of which form emulsions with water. In some instances, the emulsion may contain a significant proportion of water. this water emulsion, or any other contamination of the cleaned up chemical must be removed from the volumes put into the cleanup files.

SETTING THE SPILL LOCATION

Two options exist for setting the spill location in NRDAM/CME. The first is to select **spill Locat**. from the Physical Fates side menu and click on the chosen spill site with the cross hair cursor using the mouse. The second option is to enter the latitude and longitude of the spill site into the SPILL INFORMATION form, discussed below. For most spill sites close to land, it will be preferable to select the spill site using the mouse.

When selecting the spill site, keep in mind that the model sees the habitat grids, not the shoreline that is usually drawn on the screen. For this reason, it is advisable to draw the habitats in the area of the spill (F8:Data → Habitat Grids → Habitat Data) before selecting the spill site. Then select **Spill Locat.** and select the spill site relative to the habitats as mapped. For spill sites well offshore, this will not be an important factor. In this case, it will be easier to get an accurate placement of the spill site by entering the longitude and latitude directly into the SPILL INFORMATION form.

If the selected spill site is outside the domain of the model grids, a warning message will appear, but within the model domain, no check is performed at this point to determine whether the spill site is on land. If it is, a message will report the condition after you have completed specifying the scenario and have initiated the model run. If the spill site was set on land close to water the model will move it to the nearest water cell and a message will appear in the fates output file noting that this has occurred. If too far removed from a water cell, the program will end abnormally, returning you to the NRDAM/CME interface in a few seconds (in OUTPUT DATA). The error messages may scan by too quickly to read. The scenario name will not be on the list of those you can view. Use the F8 function key to view the model grid domain, then use the **Spill Locat.** option again, repositioning the cross-hair within a water cell of the model domain.

The NRDAM/CME system will provide north and south latitude values, as appropriate. All longitude values in NRDAM/CME are in West longitude.

SETTING THE SCENARIO

Click on the **Set Scenario** button to fill in required information using the SPILL INFORMATION form, on which you describe the spill scenario. Values from the previous simulation (or an old case, if **Old cases** was selected) are shown in each field; they are the defaults. However, if a spill location was selected under the **Spill Locat.** option (above), this location is shown in the location fields. Any of the values shown on the form may be adjusted at this time.

A help line appears at the bottom of the form to clarify the type of information being requested in each field. (Pointing and clicking with the right mouse button also provides some help.) Move between fields by using the keyboard arrow keys, TAB or ENTER, or by clicking on the desired field with the mouse; an underline cursor indicates your position in the form. Once you have filled a field you will automatically be placed in the next field. If the data you enter in a field is outside the allowable values, an error beep will sound and you must input a valid response before moving to the next field. Each of the fields are described in more detail below.

- SCENARIO - identifies the scenario being constructed. It is a unique name provided by the user to identify the current spill scenario being run. It is used by the NRDAM/CME submodels as the basis for filenames of scenario-specific data created by the model. It can be a maximum of 8 characters long. Since the scenario name is used to create DOS filenames, the characters should be limited to letters, numbers, minus signs (-) or underscores (_). File extensions are not allowed to be entered by the user. Use of other characters such as asterisks or question marks in this field will result in errors.

NOTE: It is important that you change the scenario name between model runs; otherwise, output files from a second model run will have the same filenames as those of the previous run, and will be written over the earlier output.

- LAT. - the latitude of the spill site. This information is held in three fields; the first contains the degrees, the second contains the decimal minutes, and the third contains N or S to designate north or south. These fields contain the latitude chosen under the location option described above, and may be changed here.

- LONG. - the longitude of the spill site. This information is held in three fields; the first contains the degrees, the second contains the decimal minutes, and the third contains E or W to designate east or west longitude. The fields contain the longitude chosen under the location option described above, and may be changed here.

NOTE: There are some locations in the NRDAM/CME domain that are west of 180 degrees, and thus, strictly speaking, are east. The model system is not configured to read east longitude. When you select a location, then open the spill scenario form you will see that the longitude is always given as west. You may enter a value in east longitude in the spill scenario form if desired. However the model will transform this into a west longitude, and report it as such in the various model outputs.

•YEAR - year in which the spill occurs.

•MONTH - month in which the spill occurs (1-12).

•DAY - day on which the spill begins (1-31, depending on month).

•HOUR - hour (24-hour time) in which the spill begins (0-23). For example, 1:00 p.m. is entered as hour 13.

•DESCRIPT - allows the user to enter a more descriptive text identification of the scenario. Up to 50 characters may be entered and there are no restrictions on the types of characters that may be used.

•UNITS - specifies the unit of measurement used in the amount field. A variety of units are used to measure chemicals and NRDAM/CME will accept any of the following units: Tonnes (metric tons), Barrels, Gallons, Liters, Pounds or Kilograms. When this field is entered, an arrow appears at the right side of the field; clicking on the arrow, or pressing ENTER, brings up a list of the choices. Select the desired unit by clicking on it with the mouse or by moving to it with the arrow keys and pressing ENTER. Since all internal computations performed by NRDAM/CME are in metric units, the model performs the appropriate conversions for the user.

•STAGE 1: - When an oil spill occurs in two distinct stages (e.g., a large initial release and a following slower release) release volumes may be entered as "stage 1" and "stage 2". Spills of chemicals may only specify a single stage.

AMT - specifies the total amount of the chemical released during the first stage of a spill. This amount is released evenly over the number of hours entered in the DUR field. The units for the amount are chosen in the UNITS field.

DUR - specifies the length of time (in hours) over which the first stage of release of a spill occurs. Input 0 for an instantaneous release.

•STAGE 2: - Enter 0's for single stage spills.

AMT - amount of second stage of release (oils and petroleum products only).

DUR - duration of second stage of release. This period starts at the end of the stage 1 duration period.

- CHEMICAL - specifies the chemical released in the spill simulation. The chemical shown when the form comes up is the one used in the previous simulation. Click on the arrow which appears at the right of the field, or press ENTER when the arrow appears, to bring up a list of the chemicals stored in the NRDAM/CME database. Select the desired chemical from the list. You can use the Search option at the top of the list to find a particular chemical in the database (see Section 3.3 for further instruction). The numbers to the right of the list are the CAS Registry numbers for the these chemicals.

- WIND - select the desired wind file from the list which appears when you click on the arrow at the right side of the field. Wind files are created under the DATA → WINDS option in the main menu, as explained in Section 4.4.3.1. A suitable wind file must include data for the period of the spill or an error message will result.

- CURRENTS/TIDES - select the desired tide and background currents (hydrodynamics) file from the list which appears when you click on the arrow at the right side of the field. Hydrodynamics files are created under the DATA → CURRENTS option in the main menu, as explained in Section 4.4.3.2. Select _NO_DATA.DIR or escape from the list of current files to run a scenario without currents. When a file is selected, a new window will appear listing the current components contained in that file. Select the component or components required. A check next to a component means it has been selected. Then click on OK to accept the choice.

- BIOLOGY - specifies whether or not to run the biological model immediately following the fates model. If you select no, you may still run the biology for that fates case by selecting RUN MODEL → BIO/ECON MODEL.

- CLEANUP - Select a cleanup scenario file. These files are created in the **set Cleanup** submenu of this module. Escape from the list of files to run a scenario without cleanup.

- ICE - Select an ice grid. Monthly ice grid files exist for areas on the west and north coasts of Alaska. The grids are numbered 1 to 5 from northeast to southwest. You may see the areas defined by each grid using the F8 - DATA function (escape from the Spill Information Form first) and selecting ICE → ICE GRIDS. Where ice coverage (grids) exist, the user should select an appropriate ice grid for the model run unless there is specific information that ice was not present.

- HIGH TIDE - specifies the time high tide occurred on the day of the spill event. This is entered in 24-hour time (0-23). for example, 1:00 p.m. is entered as hour 13.

•TIDE RANGE - enter the tide range in meters.

When done specifying the spill scenario, click on the OK button at the top of the form or press CTRL and ENTER simultaneously to proceed to the next step. If you do not wish to continue, click on the ESC key to return to the spill location screen. If you click ESC now, all the data entered will be lost. If you click OK, then ESC, the entries you have made will still be in the form when you selct **set Scenario** again.

The ENVIRONMENTAL VARIABLES form will come up next on the screen. This form is completed in the same manner as the SPILL INFORMATION form. Default values for each variable appear in the field. These are the values entered for the last case run. Upon entering a field, a description of the variable and the units of measurement will appear at the bottom of the form. An information window will provide the values contained in the environmental database for the chosen province and month.

The next form which comes up asks for the implicit price deflator for the Gross National Product for the quarter which includes the spill date. Recent values for this index will be found in the Survey of Current Business. See Section 14 Volume I of the documentation for the discussion of this index and a table (Table 14.2) of historical values.

You will then be asked if there were any closures. If you say Yes, you will be presented with a list of closure files from which to choose. These files are created in DATA-CLOSURES. Then you will be asked if you which to run the model now. Click yes or press Y and the model will proceed. If you select No, the entries you have made will be saved under the scenario name you gave to it. It can be recalled at any time by selecting **Old cases**.

Selecting **Old cases** brings up a list of previously-run cases (or previously-entered, but not run, cases, as noted above) from which to select. The SPILL INFORMATION form is then opened with the information from the selected old case. You may then edit any of the information in the form to rerun a modified scenario. Remember to rename the case if you wish to save both versions. The ENVIRONMENTAL VARIABLES and ECONOMIC FACTORS forms will likewise contain values from the old case and can be edited. If you previously ran with closures, you need to remember that you did so and which file was used, as **Old cases** does not default to the old values for closures.

4.4.4.2 Bio/Econ Model

This option brings up a form that allows you to run the biological effects and compensable value models to evaluate impacts in response to a previously-run fates scenario. You might choose to do this in the event that these models were not run following the initial fates run. The form requests you to enter the fates scenario name (select from the list). This form will be followed by the ECONOMIC FACTORS form and the opportunity to include a closure file before proceeding with the model run.

4.4.5 Output Data: Viewing Results of the Spill Scenario

When all submodels have completed their computations, the NRDAM/CME main menu will appear. To view model results, select the option OUTPUT DATA - MODEL RESULTS. This module allows the user to view animations of the chemical's movement once spilled and text listings of the fate of the spill and its biological impacts. Select from the following sidebar options:

Scenarioslists the spill scenarios available for display. Select a file. If a file has been selected, its name appears in an inset on this button.

Fatesallows you to view output of the physical fates model in a variety of formats. There are five options in the submenu:

Spillet Trajectory - to view the surface trajectory of the spilled chemical. You can view the display in Continuous Mode or Pause Mode, or switch from one mode to the other during the animation. Continuous Mode provides a complete animation of the surface spill trajectory, while Pause Mode pauses the display after each model time-step. To run in Pause Mode, press the P keyboard key; to switch back to Continuous Mode press the C keyboard key.

At any point during the animation, you can view a reverse trajectory - or rewind the animation to a point you would like to observe again - by pressing the left-arrow key on the keyboard. Press the right-arrow keyboard key to resume forward animation of the spill trajectory. The display can be halted at any time by pressing the ESC key. At the top right of the screen are three buttons which allow you to carry out some of these keyboard options with the mouse. The center button, with two vertical dark bars, turns pause mode on and off. The left and right buttons, with arrows on them, simulate the function of the left and right keyboard arrows allowing reverse (left) and forward (right) trajectory display.

If you have selected the DISPLAY TRAJECTORY SWEPT AREA option under **Options** → **Display Options** (see below), the track covered by the spill will be shown as the spill progresses. If this option is not activated, only the spill's time-relevant position will be shown. Display colors and patterns for the chemical and the trajectory swept paths are set in **Options** → **Set Display Colors** (see below). Three other display options affect the Spillet Trajectory output. These are DISPLAY SUBSURFACE SPILLETS, COLOR SPILLETS BY MASS, and SIZE SPILLETS BY MASS. They are discussed below under **Options**.

A mass balance histogram for the spilled substance is shown at the lower right of the screen as the spill is displayed. The mass balance histogram chronicles the relative mass of the chemical on the water surface, in the water column, in the bottom sediments, and lost to atmosphere, decayed, cleaned up, on the shoreline, or outside the model grid. Clicking in this area with the mouse displays a text listing of the current mass balance as does pressing the M Key on the key board.

During the animation, several keys are available:

F1 Brings up a screen listing these key functions
P Pause mode
C Continuous mode
F3 Measure distances
F10 Print
M Mass balance summary
→ Forward animation
← Rewind animation
CTRL-HHelp on control keys (or CTRL-F1)
CTRL-GHabitat grids
CTRL-DDepth grids
CTRL-TTides and currents
CTRL-M Map coastline outline
CTRL-K Color keys

Dissolved Chemical - to view the subsurface movement of the dissolved component of the spilled chemical. The chemical is represented in NRDAM/CME by a cloud of particles, displayed on the screen as dots, which are used by the model to compute water column concentrations. These represent dissolved concentrations and do not include amounts of a chemical that have been entrained in bulk form into the water column.

If you have selected the DISPLAY TRAJECTORY SWEPT AREA option under **Options → Display Options** (see below), the track covered by the spill will be shown as the spill progresses. If this option is not activated, only the spill's time-relevant position will be shown. Display colors and patterns for the chemical and the swept paths are set in **Options → Set Display Colors** (see below). The keys listed in Trajectory are available in Particle mode as well.

Concentrations - to view the water column and sediment concentrations of the spilled chemical. The model will display the maximum or the mean concentrations in the water column or the surface sediment concentration for each grid cell. Click on water or sediment from the menu which will appear. For water concentrations, you may then select either the maximum or mean water column values.

Color-coded rectangles are used to show different concentration levels. Use the F4 function to obtain a color key (CONC-PPB.KEY); locate the box on the screen by moving it with the mouse, then clicking when the desired position is attained.

In the CONCENTRATIONS display, the lightest color (white) represents concentrations just above the toxic threshold, while the darkest color (black) represents concentrations more than 5000 times the toxic threshold.

When the "Simulation Complete" message appears, click the mouse or press any key to return to the sidebar options.

Scenario Description - to view parameters defining the spill scenario selected in the scenarios option. The scenario name and description, start time, location, spill duration, the wind and currents files used, spill mass, the chemical name and environmental parameters are displayed in this text file.

Fates Listing - displays a complete text listing of the mass balance history of the spill.

Bio/Econ Tabular data created by running the NRDAM/CME Biological and Compensable Value submodels may be viewed and printed from within the NRDAM/CME system. To access the text files, select this option from the side menu, which will display a sidebar with the following options:

No Restoration

Damage Summary - to view a brief summary of the output from the biological effects and compensable value models for injuries and resulting damages when no restoration has occurred. The summary is stored in scenario.CVT files. Select the desired scenario from the list of scenarios and view the text.

Short Impact Table - to view summary output from the biological effects and compensable value models for injuries and resulting damages when no restoration has occurred. This text is stored in scenario.PS1 files. Select the desired scenario from the list of scenarios and view the text.

Detailed Impacts - to view detailed output from the biological effects and compensable value models for injuries and resulting damages when no restoration has occurred. This text is stored in scenario.PD1 files. Select the desired scenario from the list of scenarios and view the text.

With Restoration

Damage Summary - to view a brief summary of the output from the biological effects and compensable value models for injuries and resulting damages when restoration is undertaken. This file consists of a short summary of the impacts tables along with the costs of restoration. The summary is stored in scenario.DAM files. Select the desired scenario from the list of scenarios and view the text.

Short Impact Table - to view summary output from the biological effects and compensable value models for impacts when restoration is undertaken. This text is stored in scenario.PS2 files. Select the desired scenario from the list of scenarios and view the text.

Detailed Impacts - to view detailed output form the biological effects and compensable value models for impacts when restoration is undertaken. This text is stored in scenario.PD2 files. Select the desired scenario from the list of scenarios and view the text.

Assesmnt Rpt This option produces an Assessment Report for the selected scenario. The Assessment Report is a complete reporting of the inputs used by the model and the biological and compensable value outputs produced for a given scenario. Listings include: (1) a description of the inputs for the model system setup; (2) wind data, (3) current data, (4) cleanup data, (5) closure data, (6) the physical fates submodel listing, (7) the detailed report of the biological and compensable value submodels without restoration; (8) the detailed report of the biological and compensable value submodels with restoration costs, (9) the total damages report. When **Assesmnt Rpt** is selected, you will be prompted to provide a name for a file in which to store the results. A .DOC extension is added to the file name and the file is placed in the NRDAMCME¥DATA¥COMBINED directory. The resulting file will typically be very large.

Optionsto control features of the animation displays. There are three suboptions:

Display Options - to toggle following options on/off, click on the option; a check mark will appear to the right of an option when it is activated:

Display Trajectory Swept Area - When this option is activated, the track covered by the spill will be shown; if this option is not activated, only the spill's time-relevant position will be shown.

Display Subsurface Spillets - When this is selected, spillets of oil which are submerged but not dissolved will be displayed during SPILLET TRAJECTORY output. They will be the same color as surface spillets (whether or not they are color-coded by size) but will be shaded and thus darker than the corresponding surface spillets.

Color Spillets by Mass - When this is checked, the individual spillets making up a surface slick will be colored to reflect their individual masses. A key for these colors (SPILMASS.KEY) may be displayed using the F4 function key.

Size Spilled by Mass - when this is checked, individual spillets used to display a surface slick will be drawn at a size to more realistically reflect their calculated size. This may be used in any combination with COLOR SPILLETS BY MASS.

Display Shoreline Hits - will highlight areas of shoreline contaminated by the spilled chemical slick in a color chosen under SET DISPLAY COLORS (below).

Display Wind Vector - This brings up a display of wind direction and speed during the time covered by the spill simulation. When a simulation starts, a box will appear on the screen with a moving arrow indicating the wind speed and direction during each time period. Wind speed is given at the bottom of the box in meters per second. You may relocate the box at any time during the display by dragging it with the mouse.

Set Display Delay - sets the speed of animation. Enter 1 for rapid animation (1/100 sec. pause between time-steps), and higher numbers to slow the animation down for closer examination.

Set Display Colors - sets the colors used to display the chemical trajectory in the SPILLET TRAJECTORY and DISSOLVED CHEMICAL spill animations. A form will be presented, with five settings to define SPILLET TRAJECTORY (surface) animation color, the path followed by the surface trajectory, a setting for shoreline chemical color, and two settings to define DISSOLVED CHEMICAL (subsurface) and subsurface trajectory path colors. The "path" settings define colors and patterns used only when the DISPLAY TRAJECTORY SWEPT AREA option is activated (see above, DISPLAY OPTIONS).

4.4.6 System

SYSTEM provides access to several functions in the NRDAM/CME system. Three sub options are available.

4.4.6.1 System Parameters

NRDAM/CME requires large data files stored in multiple directories on the system hard drive. To set the system path to these files, select SYSTEM from the main menu and then select SYSTEM PARAMETERS. From within the system parameters form, the user may reset the path pointing to location-specific files ("Location Data Path"). The ability to change this path allows the user to store the data on any hard drive where space is available.

Maps in NRDAM/CME are displayed using a geographic coordinate system (Longitude, Latitude) to facilitate worldwide coverage within a single reference system. Distances measured within the map window may be obtained using either kilometers or nautical miles. The other field in the system parameters form allows the user to switch between these two distance units.

4.4.6.2 Set Map Colors

The color used to depict water and the color and pattern used for the shoreline within the NRDAM/CME map window may be changed to any of the 16 available colors or color/pattern combinations. Locations in NRDAMCME are not configured in a way that allows land to be displayed as a distinct color from water. Thus, you can normally control only the "land color" which is, in effect, the shoreline, and the "water color", which will be everything else. By selecting SYSTEM from the main menu and then SET MAP COLORS from the sub menu, a form pops to the screen within which the user may change map colors and patterns. The new map colors will take effect when the map is redrawn.

When you select OK, all the old shoreline drawing files are erased, and new ones are created each time you select a new location. Each new location will appear to draw quite slowly while the new drawing file is being created. For this reason, be sure that if you select SET MAP COLORS and decide not to change anything, you ESCAPE rather than selecting OK.

4.4.6.3 Delete Files

Select SYSTEM → DELETE FILES to delete data files stored on the hard disk that are no longer needed. Types of files which may be deleted include scenario files, wind files, current files, cleanup files, closure files and assessment report files. Selecting any of the options brings a window to the screen listing the files of the type selected. Simply click on the file name to be deleted.

4.4.7 Help

Selecting the option HELP → SYSTEM HELP will bring up a text file giving an overview of the NRDAM/CME menu options and how to use the system. For more extensive help in an individual option, press the Fl key while in that option. View the help text as described in Section 3.5.

4.4.8 Quit

Selecting the option QUIT → EXIT TO DOS will close down the NRDAM/CME system and put you back at the DOS level in the ¥NRDAMCME directory.

5. FILES AND BATCH PROCESSING

In the process of creating and running a spill scenario, numerous files are created in various parts of the NRDAM/CME system directory structure. This section gives a brief consideration to what these files are and where they are stored, and which files must be saved to archive a scenario for running in the future, and then discusses how to run the model system in batch mode.

5.1 Files Used and Created By a Spill Scenario

Below is a partial list of the files used or created by a spill scenario. It is not a complete list in that there are several additional files which are created or modified in the process of running a scenario that are not on the list, but most of these are not essential to a model run as the model system will create them if they are missing. Notes are given on the contents or function of some of the files where this may prove useful.

Scenario description files:
  ¥NRDAMCME¥LOC_DATA¥*location*¥CASES¥*scenario*.DAT
Wind files:
  ¥NRDAMCME¥LOC_DATA¥*location*¥WINDS¥*filename*.WND
Current files:
  ¥NRDAMCME¥LOC_DATA¥*location*¥CURRENTS¥*filename*.DIR
  (require other filename.* files in this directory to edit *.DIR file)
Closure files:
  ¥NRDAMCME¥LOC_DATA¥*location*¥CLOSURES¥*filename*.CLS
Cleanup files:
  ¥NRDAMCME¥LOC_DATA¥*location*¥RESPONSE¥*filename*.LRF

Files found in ¥NRDAMCME¥LOC_DATA¥*location*¥MODELOUT

ASCII files
  scenario.PRT text listing of the mass balance history of the spill
  scenario.CVT summary of injuries and damages, without restoration
  scenario.DAM summary of injuries and damages, with restoration
  scenario.PS1 short impact table, without restoration
  scenario.PS2 short impact table, with restoration
  scenario.PD1 detailed injuries and damages, without restoration
  scenario.PD2 detailed injuries and damages, with restoration
  scenario.SUM
Binary files
  scenario.DS1
  scenario.DS2
  scenario.HDR listing of model parameters used in fates simulation
  scenario.KIL the KIL and LNK files are needed to run biological impacts for a previously-run fates case
  scenario.LNK record of chemical fates data passed to biological model
  scenario.PRC

scenario.REC
scenario.SBC data for graphic display of water concentrations
scenario.SBP data for graphic display of the dissolved chemical
scenario.SDC data for graphic display of sediment concentrations
scenario.SHO
scenario.SLB
scenario.TRJ data for graphic display of spill surface trajectory

5.2 Saving a Scenario

The NRDAM/CME system is not specifically set up to allow a user to archive a particular scenario outside of the NRDAM/CME itself. The Assessment Report is a complete document containing all the information necessary to recreate a case, but it is not in a format which makes this a simple process, and some aspects, such as recreation of a current file from the Assessment Report, could be quite tedious.

The user may find it useful or necessary to save an entire case in such a way that it can be run at some later date or on another computer. To do so, it is necessary to save a variety of files in several of the subdirectories of the NRDAM/CME installation. Most of these files will be found in the subdirectories of the location (E_COAST, ALASKA, etc.) in which the case was run. In the following list, the scenario name is indicated as *scenario*, any other filename you have used to distinguish a file is indicated as *filename*, and the location is indicated as *location*. In each case, where no file was created or edited, there will be no need to save a file, so if cleanup was not performed, there will be no cleanup file, and if habitats were not edited, there is no need to save the habitat file except to note that original NRDAM/CME habitats were used.

Saving these files will allow to you recreate a case:
Scenario description:

¥NRDAMCME¥LOC_DATA¥*location*¥CASES¥*scenario*.DAT

Winds:

¥NRDAMCME¥LOC_DATA¥*location*¥WINDS¥*filename*.WND

Currents:

¥NRDAMCME¥LOC_DATA¥*location*¥CURRENTS¥*filename*.*

(**NOTE**: only the filename.DIR file is required, but by saving the other *filename*.* files, you will able to edit this file if you choose).

Closures:

¥NRDAMCME¥LOC_DATA¥*location*¥CLOSURES¥*filename*.CLS

Cleanup:

¥NRDAMCME¥LOC_DATA¥*location*¥RESPONSE¥*filename*.LRF

Habitats:

[save any edited habitat grid both as a backup grid (grid P#; identifier *nn*) as well as the active grid. Upon restoring, it will be necessary to select the appropriate grid in HABITAT EDITOR and then restore this backup grid as the active grid.]

¥NRDAMCME¥PROVDATA¥NAMERICA¥GRIDS¥USER¥P#.U*nn*

Of course, it will be necessary to copy each of these files back to their respective directories to be able to recreate the scenario from the backed-up files.

5.3 Running NRDAM/CME in Batch Mode

For a variety of reasons, it may be desirable or necessary to run NRDAM/CME without using the graphic interface. One reason would be the unlikely event that you have a computer display which is not compatible with NRDAM/CME. This will present difficulties in any event, since it will be very difficult to prepare suitable data files for wind and currents, and to view the relationship of the spill site to habitat grids in the model system. It is not recommended that the system be run under these conditions. A more likely situation is that you might wish to run the model system in batch mode allowing a series of scenarios to be run without user intervention. The process for running NRDAM/CME in batch mode is described below. This process does not incorporate closures files into the model run. Do not try to run scenarios with closures in them in batch mode.

It will be easiest to allow the NRDAM/CME system to create the appropriate scenario description files. This will ensure that the format is correct and entries are made in the appropriate location. It is possible for the user to prepare or amend these files using a text editor, but description of the details of this process are beyond the intent of this manual.

1)In the NRDAM/CME system, using the RUN MODEL module, set up the scenarios required, just as you would to run the model within the system, with one change: when asked "Run the Chemical Fates Model Now?" answer NO. Continue to create all the scenarios in this way, then quit the NRDAM/CME system.

2)Using a text editor, create a file called RUNS.LST in the top level (¥NRDAMCME) directory. In this file, enter the location for the first scenario on line one and the scenario name on line two. Location names correspond exactly to those listed as locations in NRDAM/CME and are the names of the subdirectories in NRDAMCME¥LOC_DATA. Enter the location for scenario two on line three and the scenario name on line four. Continue in this manner until all scenarios have been listed. Enter no other information into this file.

3)In the NRDAMCME directory, type BATCH. The scenarios listed will be run sequentially. Operation of the computer will appear essentially the same as it does when run from the graphics interface.

4)When all runs are complete, run NRDAM/CME to view results of the model runs

社団法人　日本海難防止協会

東京都港区虎ノ門一丁目１５番１６号
〒105-0001　海洋船舶ビル４階

ＴＥＬ ０３（３５０２）２２３１
ＦＡＸ ０３（３５８１）６１３６