Integrated Baseline System (IBS)

Version 2.0

Models Guide

March 1994

**Prepared for the
U.S. Army Nuclear and Chemical Agency
under a Related Services Agreement
with the U.S. Department of Energy
Contract DE-AC06-76RLO 1830**

**Pacific Northwest Laboratory
Operated for the U.S. Department of Energy
by Battelle Memorial Institute**

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor Battelle Memorial Institute, nor any of their employees, makes **any warranty, expressed or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights.** Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof, or Battelle Memorial Institute. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

PACIFIC NORTHWEST LABORATORY
operated by
BATTELLE MEMORIAL INSTITUTE
for the
UNITED STATES DEPARTMENT OF ENERGY
under Contract DE-AC06-76RLO 1830

Printed in the United States of America

Available to DOE and DOE contractors from the
Office of Scientific and Technical Information, P.O. Box 62, Oak Ridge, TN 37831;
prices available from (615) 576-8401. FTS 626-8401.

Available to the public from the National Technical Information Service,
U.S. Department of Commerce, 5285 Port Royal Rd., Springfield, VA 22161.

The contents of this report were printed on recycled paper

INTEGRATED BASELINE SYSTEM (IBS)
Version 2.0

MODELS GUIDE

March 1994

Prepared for the
U.S. Army Nuclear and Chemical Agency
under a Related Services Agreement
with the U.S. Department of Energy
Contract DE-AC06-76RLO 1830

Pacific Northwest Laboratory
Richland, Washington 99352

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

NOTICE

The Integrated Baseline System (IBS) is an emergency planning, training, and response tool intended to supplement existing resources for emergency planning, training, and management. IBS results must NOT be used as the sole basis for a decision regarding public safety. Although IBS is operational, not all IBS software/models have been verified. The assumptions of the IBS software must be reviewed to assure its appropriateness for an intended use.

Preface

The Integrated Baseline System (IBS) is an emergency management planning and analysis tool being developed under the direction of the U.S. Army Nuclear and Chemical Agency. The following documents were developed to support system users. The audience for each is identified.

This *IBS Models Guide* summarizes the IBS use of several computer models for predicting the results of emergency situations. These include models for predicting dispersion/doses of airborne contaminants, traffic evacuation, explosion effects, heat radiation from a fire, and siren sound transmission. The guide references additional technical documentation on the models when such documentation is available from other sources. **Audience:** chiefly emergency management planners and analysts, but also data managers and system managers.

The *IBS User Guide* explains how to start and use the *IBS program*, which is designed to help civilian emergency management personnel to plan for and support their responses to a chemical-releasing *event* at a military chemical stockpile.^(a) **Audience:** all users of the IBS, especially emergency management planners and analysts.

The *IBS Utilities Guide* explains how you can use IBS utility programs to manage and manipulate various kinds of IBS data. These programs include utilities for creating, editing, and displaying maps and other data that are referenced to geographic location. **Audience:** chiefly data managers but also system managers and some emergency management planners and analysts.

The *IBS Data Management Guide* provides the information needed to manage the data files and database used to support the administrative, user-environment, database management, and operational capabilities of the IBS. **Audience:** chiefly database administrators and system managers, but also emergency management planners and analysts who want to know details of the emergency management data.

The *IBS System Management Guide* defines IBS hardware and software requirements and gives instructions for installing, upgrading, or transferring the IBS software package. **Audience:** system managers.

(a) The IBS program was developed as part of the U.S. Army's Chemical Stockpile Emergency Preparedness Program (CSEPP).

About This Guide

Purpose

The Integrated Baseline System (IBS), is an emergency management planning and analysis tool that is being developed under the direction of the U.S. Army Nuclear and Chemical Agency (USANCA)^(a). This document is the models guide for the IBS and explains how to use the emergency related computer models.

Scope

This document provides information for the experienced system user, and is the primary reference for the computer modeling software supplied with the system. It is designed for emergency managers and planners, and others familiar with the concepts of computer modeling.

Although the IBS manual set covers basic and advanced operations, it is **not** a complete reference document set. Emergency situation modeling software in the IBS is supported by additional technical documents. Some of the other IBS software is commercial software for which more complete documentation is available. The IBS manuals reference such documentation where necessary.

General questions regarding IBS should be directed to the IBS Project Manager for the U.S. Army Nuclear and Chemical Agency (USANCA):

COL Lawrence Skelly
Chief, Chemical Division
U.S. Army Nuclear and Chemical Agency
7150 Heller Loop, Suite 101
Springfield, VA 22150-3198

Software use questions and problems should be addressed to the IBS Help Desk operated by Innovative Emergency Management (IEM):

(504) 767-1138
Innovative Emergency Management
Suite E
Baton Rouge, LA 70808-4362

(a) The IBS is being developed by the Pacific Northwest Laboratory (PNL). PNL is operated for the U.S. Department of Energy by Battelle Memorial Institute under Contract DE-AC06-76RLO 1830.

About This Guide

All references to IBS should be made through USANCA. If an emergency contact is necessary, and USANCA cannot be reached, a second source of information is the software developer, Pacific Northwest Laboratory:

Pacific Northwest Laboratory
Attn: Blanche M. Wood, K7-22
POB 999, MS K7-22
Richland, WA 99352
Phone (509) 375-2615

Audience

The primary audience for this document is composed of emergency planners and analysts, information managers, and geographic database administrators.

Prerequisites

Knowledge Requirements for Using the IBS. Users of the IBS software should receive training in the use of IBS capabilities for emergency management planning and operational tasks associated with CSEPP. For those users, this document is a reference guide to IBS models.

Organization

The chapters of this guide introduce IBS concepts and functions and explain how to navigate the menus and operate the various parts of the system.

1. Overview

A description of general system modeling and support functions.

2. Regional Evacuation Analysis

A brief guide to managing the job environment used to support the computer models.

3. D2 Chemical Hazard Prediction Model

A reference guide to the model inputs for the D2 chemical hazard prediction model.

4. Model Library — MESORAD Model

A reference guide to the MESORAD model and components such as atmospheric dispersion, deposition, decay of radiological materials, and dose modeling.

5. Generate Evacuation Network

A reference guide to the Generate Evacuation Network, a capability for generating evacuation cases and a traffic network for the evacuation model, IDYNEV.

6. Dynamic Evacuation Model

A reference guide to the Interactive Dynamic Evacuation model (IDYNEV), which helps in predicting the progress of area evacuations by vehicle.

7. Model Library — Outdoor Sound Propagation Model

A reference guide to the Outdoor Sound Propagation Model (OSPM), which is useful in evaluating placement of warning sirens.

8. Model Library — CHEMS Models

A reference guide to the CHEMS models: Explosion, Fire, and Dispersion, that help in assessing the hazards of a chemical release.

9. Integrated Modeling (EESF)

A reference guide to the integrated modeling capabilities of the Exercise Evaluation and Simulation Facility (EESF).

How to Use this Document

This guide is arranged into sections which describe the use of a computer model or suite of related models. The experienced user can read the overview in Section 1, and then turn to the section that contains the model of interest. These models will be of use to the Information Manager and emergency planners and analysts.

About This Guide

Contents

Preface	III
About This Guide	V
 Section 1 Overview	 1.1
1.1 User Environment	1.3
1.1.1 Description of the User Environment	1.3
1.1.2 What You Need to Know to Manage the User Environment	1.4
1.1.3 Terminology for the User Environment	1.5
1.2 IBS Application Shell	1.6
1.3 IBSSH Application Shell	1.6
1.3.1 Graphic Models Menu	1.7
1.3.2 Non-Graphic Models Menu	1.9
1.4 IBS User Environment	1.10
1.4.1 Environment Files	1.10
1.4.2 Changing the User Environment	1.10
1.4.3 Using Case/Site Functions	1.11
1.5 Using Mapping Functions within Model Subsystems	1.12
 Section 2 Regional Evacuation Analysis	 2.1
2.1 Description of the Regional Evacuation Model	2.2
2.2 What You Need to Use the Regional Evacuation Model	2.3
2.3 Starting a Regional Evacuation Model Work Session	2.5
2.4 Using the Regional Evacuation Main Menu	2.6
2.5 Generate a First-Cut Evacuation Case - Graphic Mode Menu Only	2.8
2.6 Update Current Site Network (Model Input Data) - Text Mode Menu Only ..	2.8
2.7 Display Site Map from Site File - Graphic Mode Menu Only	2.8
2.8 Execute Evacuation Model - Batch	2.10
2.9 Execute Evacuation Model - Interactive	2.10
2.10 Show Model Results - Graphic Mode Menu Only	2.11
2.11 Display Reports	2.13
2.12 Change Site	2.13
2.14 Case Maintenance	2.14
2.15 Population Maintenance	2.15
 Section 3 D2 Chemical Hazard Prediction Model	 3.1
3.1 Description of D2	3.2
3.2 Modifying D2 Model Input Data	3.3
 Section 4	 4.1
4.1 Description of MESORAD	4.2
4.1.1 Map-Related Display of Modeling Results	4.2
4.2 What You Need to Know to Use MESORAD	4.2
4.2.1 Meteorology Model	4.3
4.2.2 Dose Model	4.3
4.3 Using the MESORAD Top-Level Menu	4.4
4.4 MESORAD Graphic Options - Graphic Menu Only	4.5

Contents

4.5	Update Elevation - Graphic Menu Only	4.11
4.6	Update Case Data	4.11
4.6.1	Change Control Parameters	4.13
4.6.2	Change Source Details	4.15
4.6.3	Change Weather Data	4.16
4.6.4	Change Checkpoints	4.18
4.6.5	Change Station Definitions	4.19
4.6.6	Change Terrain	4.20
4.6.7	Change Cartesian Grid	4.21
4.6.8	Change Polar Grid	4.22
4.6.9	Display Weather Directory	4.23
4.6.10	Change Output Options	4.25
4.6.11	Generate Polar Grid	4.26
4.6.12	Generate Cartesian Grid	4.27
4.7	Display Base Map - Graphic Menu Only	4.27
4.8	Print/Display Reports	4.27
4.9	Perform File Maintenance	4.30
4.10	Execute MESORAD - Interactive	4.30
4.11	Execute MESORAD - Batch	4.31
4.12	Change Case	4.31
4.13	If You Have Problems	4.32
4.14	Status and Diagnostics Messages	4.32
4.15	MESORAD/FEMA	4.33
4.15.1	Ingestion Pathway Doses	4.33
4.15.2	MESORAD/FEMA Team Reports	4.39
Section 5		5.1
5.1	Description of Generating an Evacuation Network	5.1
5.2	What You Need to Know About Generating an Evacuation Network	5.1
5.3	Generating a First-Cut Evacuation Traffic Network and Cases	5.2
5.3.2	Specifying a Network Parameter File	5.6
5.3.3	Selecting Topographies to Be Used During Network Generation	5.7
5.3.4	Using the Network/Case Generation Menu	5.8
5.3.5	Defining an Evacuation Boundary	5.9
5.3.6	Defining Host Locations	5.11
5.3.7	Qualifying Roadway Selection	5.14
5.3.8	Generating the Network	5.16
5.4	Details of Automatic Evacuation Case Generation	5.20
5.4.1	Network Building	5.20
5.4.2	Trip Generation: Associating Populations with the Network	5.24
5.4.3	Evacuation Cases and Case Files Created	5.27
Section 6	Dynamic Evacuation Model	6.1
6.1	Description of IDYNEV	6.2
6.2	What You Need to Know to Use IDYNEV	6.4
6.3	Starting an IDYNEV Work Session	6.5
6.3.2	Selecting Graphic or Text Mode	6.6
6.3.3	Selecting Evacuation Model Input or the IDYNEV Function Menu	6.6
6.3.4	Starting IDYNEV from the IBS Program	6.6
6.4	Modifying Evacuation Model Input Data	6.7
6.4.1	Key Terminology for Evacuation Model Input	6.7
6.4.2	Starting a Data Input Session with PREDYNGR	6.8

6.4.3	Specifying a Time Period and Subnetwork	6.11
6.4.4	Selecting Which and How Input Data Will Be Modified	6.12
6.4.5	Title (Type 00)	6.16
6.4.6	Identification (Type 01)	6.16
6.4.7	Run Control (Type 02)	6.16
6.4.8	Time Period Specification (Type 03)	6.17
6.4.9	Time Step Data (Type 04) and Output Options (Type 05)	6.17
6.4.10	Origin Control (Type 06)	6.18
6.4.11	Link Characteristics (Type 11)	6.19
6.4.12	Link Turn Movements (Type 21)	6.20
6.4.13	Sign or Signal Control Data (Types 35 & 36)	6.20
6.4.14	Evacuation Data (Type 49)	6.21
6.4.15	Entry Link Volumes (Type 50)	6.22
6.4.16	Source/Sink Volumes (Type 51)	6.22
6.4.17	Load Factors (Type 52)	6.22
6.4.18	Traffic Assignment Parameters (Type 175)	6.23
6.4.19	Origin Trip Production Volumes (Type 176)	6.23
6.4.20	Internal Centroids (Type 177)	6.24
6.4.21	Destination Capacities (Type 178)	6.24
6.5	Graphics Mode of Modifying Input to the Evacuation Model	6.25
6.6	Using the IDYNEV Function Selection Menu	6.29
6.6.1	Execute Evacuation - Interactive	6.31
6.6.2	Execute Evacuation - Batch	6.32
6.6.3	Display Results On Map - Graphics Mode Only	6.33
6.6.4	Display Base Map - Graphics Mode Only	6.36
6.6.5	Print/Display Reports	6.36
6.6.6	Change Case/Site	6.37
6.6.7	Perform File Maintenance	6.38
6.6.8	Exit	6.38
6.7	If You Have Problems	6.38
6.8	Status and Diagnostic Messages	6.39
Section 7	Model Library - Outdoor Sound Propagation Model	7.1
7.1	Description of the Outdoor Sound Propagation Model	7.3
7.2	What You Need to Know to Use the OSPM	7.4
7.2.1	Overview of the Model	7.4
7.2.2	Obtaining and Using Elevation Data	7.4
7.3	Using the OSPM Function Selection Menu	7.8
7.4	Change Case	7.10
7.5	Display Base Map - Graphic Menu Only	7.11
7.6	Contour Results - Graphic Menu Only	7.12
7.7	Report Decibel (dB) - Graphic Menu Only	7.14
7.8	Edit Elevation File - Graphic Menu Only	7.15
7.9	Update Impedance - Graphic Menu Only	7.16
7.10	Place Siren - Graphic Menu Only	7.23
7.10.1	Change Default Siren	7.23
7.10.2	Add Siren	7.24
7.10.3	Delete Siren	7.25
7.11	Update Siren - Graphic Menu Only	7.27
7.12	List Site Specs	7.28
7.13	Execute OSPM - Interactive	7.29
7.14	Execute OSPM - Batch	7.31

Contents

7.15	Perform File Maintenance	7.31
7.16	Delete Contours - Graphic Menu Only	7.31
7.17	Create a New Siren Type	7.32
Section 8	Model Library - CHEMS Models	8.1
8.1	Description of the CHEMS Models	8.2
8.1.1	Explosion Model	8.2
8.1.2	Heat Radiation Model	8.2
8.1.3	Dispersion Model	8.2
8.2	Using the CHEMS Models	8.3
8.2.1	Overview of the Explosion Model	8.3
8.2.2	Overview of the Heat Radiation Model	8.6
8.2.3	Overview of the Dispersion Model	8.8
8.2.4	Supplementary References	8.9
8.3	Using the CHEMS Models Menu	8.10
8.4	Update Case Data	8.11
8.4.1	Chemical Spilled	8.12
8.4.2	Time of Spill / Current Time	8.13
8.4.3	Environmental Conditions	8.14
8.4.4	Mode of Operation	8.16
8.4.5	Model Selection/Model Dependent Inputs	8.17
8.4.6	Source Inputs	8.18
8.4.7	Output Specifications	8.21
8.4.8	Exit	8.21
8.5	Select Spill Location	8.23
8.6	Change Case/Site	8.23
8.7	Display Base Map - Graphic Menu Only	8.24
8.8	Execute Model Inter or Execute Model Batch	8.25
8.9	Print/Display Reports	8.25
8.10	Perform File Maintenance	8.26
Section 9	Model Library - Integrated Modeling (EESF)	9.1
9.1	Description of EESF	9.2
9.1.1	Evacuation Model (IDYNEV)	9.3
9.1.2	Meteorology/Dose Model (MESORAD/FEMA)	9.3
9.1.3	Display Map-Related Modeling Results	9.4
9.2	Using EESF	9.4
9.3	Obtaining On-Line HELP with Data Input	9.5
9.4	Starting an EESF Work Session	9.5
9.4.1	Selecting Graphic or Text Mode	9.6
9.5	Obtaining and Using the Top-Level EESF Menu	9.6
9.6	Display Base Map - Graphics Menu Only	9.8
9.7	Modify Meteorology/Dose Model Input Data	9.9
9.8	Model Evacuation - Batch	9.10
9.9	Model Evacuation - Interactive	9.10
9.10	Modify Evacuation Start Time for Dose Calculations	9.11
9.11	Execute MESORAD - Batch	9.12
9.12	Execute MESORAD - Interactive	9.13
9.13	Print Reports	9.14
9.14	Display Reports - On Terminal	9.16
9.15	Display Map-Related Graphics - Graphics Menu Only	9.18
9.16	Case File Directory and Optional Delete	9.24

9.17 Change Case Numbers	9.25
9.18 Concurrent Execution of MESORAD and IDYNEV	9.26
9.18.1 Initial Sequence of Operations	9.26
9.18.2 Using the Concurrent Execution Options Menu	9.27
9.18.3 Overall Sequence of Operations	9.31
9.19 If You Have Problems	9.33
9.20 Status and Diagnostic Messages	9.33
9.21 References	9.33

Section 1

Overview

The IBS contains a set of emergency-related computer models--computational programs for simulating or analyzing some aspect of emergency situations. Emergency managers and organizations can use these models to supplement other tools for emergency response planning and training and for evaluating emergency decision processes.

This section supplies background information for understanding and using the IBS modeling and support functions.

The dynamic nature of the IBS system development has created some ambiguity within the modeling environment. The user environment of each user is used regardless of which module within IBS is accessed.

Note: The IBS main program and some of the utilities use operational mode under certain circumstances to direct operations to the infomanager current operational data set. The models do not generally support the operational mode concept and refer only to the user's current planning data set. An exception is the D2 model, which does support operational mode.

The following diagram shows the relationship between the entire IBS system, the user environment, and the various methods of access to the models themselves.

Figure 1.1. IBS System Diagram

There are three methods that can be used to access models, these methods are:

1. Use the IBS application shell developed specifically to support the CSEPP program. The IDYNEV evacuation model and the D2 dispersion model are accessible through the IBS application shell.

2. Use the IBSSH application shell to access the MESORAD dispersion model, the Outdoor Sound Propagation Model (OSPM), the CHEMS model, and the IDYNEV evacuation model and its supporting network generation capability. IBSSH also enables you to access other application shells that enable you to use the same models in specific ways. These application shells are
 - Exercise Evaluation Simulation Facility (EESF)
 - Regional Evacuation Analysis (REA)
3. All models can be accessed directly from the command line prompt. This method is useful for experienced users who wish to interact only with a specific model.

Access options for each model are described in the section preceding the description of the model itself.

1.1 User Environment

The IBS system maintains a **user environment**, which defines the overall setup of equipment, software, and data that you are currently using. The user environment affects both mapping (graphics) interactions and modeling functions.

1.1.1 Description of the User Environment

The IBS user environment is the current setup of hardware, software, and data that you can access. The job environment includes (but is not limited to) the following:

- the type of graphics input device that you are using
- the type of graphics display that you are using
- the current site and the sites available to you
- the current case for any model used for the current site
- the current sets of site elevation and population data for use as model input
- the files that control which map database and features are used.

You can modify the user environment in one of the following ways:

- From the IBSSH application shell, use the Setup option and then select the Job Environment option.
- From the IBS application shell, select the Setup option and use the Job Environment option.

General information about the user environment and further information on specific files controlling the user environment is contained in the *IBS Data Management Guide*.

1.1.2 What You Need to Know to Manage the User Environment

This part is intended for all users of IBS. It defines many basic concepts and some advanced ones. This section defines some terms that you will need for understanding the user environment. For further information on the files accessed by the user environment, see the *IBS Data Management Guide*.

You will modify the user environment by selecting items from a menu and then changing values on data forms. The following paragraphs are a few general questions and answers about the user environment.

Why Change the User Environment? If you work with more than one site, or as a planner with your own separate version of the IBS common site, you will need to switch between sites. The user environment enables you to do that. Switching between sites means switching between different sets of data. IBS lists the sites that you can access in a **site master file** called SITE.DAT in your top-level directory. Even if you generally work with only one site, you may wish to switch between hardware devices or between sets of data used by a particular computer model.

What is the start-up user environment? At the start of each IBS work session, IBS sets up the user environment as it was when you quit your last work session. IBS stores user environment information in the file JOB_ENV.DAT in the user's top-level directory. If you are using IBS for the first time, the start-up user environment is specified by your system manager.

How does the user environment affect graphics displays? Each site has a database of cartographic (map) information. Some options for managing the job environment enable you to set up the IBS map display software to display features that are specific to your needs. For example, you might want to use special icons (graphic symbols) on your map displays. To take full advantage of these options, you must have an in-depth knowledge of how the map display software works. For further information, see the *IBS Utilities Guide* and the *IBS Data Management Guide*.

1.1.3 Terminology for the User Environment

IBS brings together a large assortment of computer tools and models related to emergency planning and response in one system. The IBS **models** are computer programs that simulate some aspect of emergency situations; for example, the Dynamic Evacuation Model (IDYNEV) simulates the evacuation of vehicles over a traffic network. You can apply the IBS models to specifically defined geographic locales called **sites**. For each model, a site can have a number of sets of model data that describe different situations. Each set of data (and situation) represents a model **case**. For example, one evacuation case might include all roadways on a site while another evacuation case includes all roadways except those that are flooded and impassable.

The options for modifying the user environment enable you to select which site is in use, which case is in use for each model, and which geographic database is currently accessible.

- **case**: one set of input data for a model. If the model has been run (executed) for a case, the case can also have output data. A three-digit **case number** (000 through 999) identifies each case.
 - **current case**: the current set of input (and output) data associated with a particular model in the user's environment.
 - **base case**: a set of data that represents a standard case for a particular model. The term **base case** can also refer to a case from which a second case is copied. For example, you might have an OSPM base case 005 that represents a site under typical conditions. You might copy that base case to create a case 006 and then change the input data for case 006 to reflect special conditions, such as a flooding situation.
 - **system base case**: For each model, IBS initially supplies a system base case. Each system base case is a complete but mock set of data that is not related to real conditions or site characteristics. The system base case serves as a template that can be edited to create a different case that reflects real conditions. System base cases have a **case number** of 000.
- **elevation set**: a set of elevation data associated with a site. The **current elevation set** is the one selected for use with the site and the OSPM model.
- **graphics control files**: files that control what map and graphic items (topographies, icons, areas, etc.) can be displayed by the map display software. See the *IBS Data Management Guide* for complete information.
- **IDYNEV cases**: IDYNEV has two kinds of cases: Traffic Assignment and Simulator. By convention, Traffic Assignment cases have odd numbers and associated Simulator cases have even numbers.

- **IBS model:** a computer program that simulates some aspect of an emergency situation and produces information that is useful for emergency planning or response
- **population set:** a set of population data associated with a site. The **current population set** is the one selected for use with the site, the IDYNEV network generation capability, and the PREDYN population update utility.
- **site:** one of the specifically defined geographic locales for which the system contains map data and other information related to the models. The **current site** is the site selected for display or modeling.
- **site master file:** a file that contains a master list of all the sites to which you have access.

1.2 IBS Application Shell

The IBS program enables you to access two models:

IDYNEV

D2

These models can be started and used within the IBS program. For further information, see the *IBS User Guide*.

1.3 IBSSH Application Shell

The IBSSH (IBS Shell) utility enables you to access many of the IBS models, utilities, and other programs by using a menu-driven interface. It has one top-level menu that accesses nine submenus. Except for items in the Setup submenu, all options in all the other submenus run a specific utility or IBS program: the program names appear on the menu along with short descriptions. If you know the name of a program, it is much faster to just type its name at the system prompt because IBSSH does have some overhead associated with it.

The models you can access through the IBSSH application shell are:

IDYNEV
MESORAD
OSPM
CHEMS
D2

Also accessible through IBSSH are the following application shells:

Exercise Evaluation Simulation Facility (EESF) Regional Evacuation Analysis (REA)

To start IBSSH, perform the following procedures.

1. Login to the VAX as usual. For complete login procedures, see the *IBS User Guide*. The VAX system prompt will display.
2. At the VAX system prompt, type *ibssh* and press the Return key. The IBSSH top-level menu will display as shown in Figure 1.2.

```

      IBS Shell: Main Menu
-----
0 : EXIT
1 : Setup                                -->
2 : IBS Programs                        -->
3 : System Administration Utilities     -->
4 : Conversion Utilities                -->
5 : Miscellaneous Utilities             -->
6 : Models                             (Graphic) -->
7 : Models                             (Non-Graphic) -->
8 : GIS Utilities                       (Graphic) -->
9 : GIS Utilities                       (Non-Graphic) -->
-----
Menu Choice (?=Help) ==>
```

Figure 1.2. IBS Shell Top-Level Menu

To start a Models program, select either the Models (Graphic) or the Models (Non-Graphic) option and then choose the specific utility or program from the submenu. The Models submenus are illustrated here.

The menus labeled "(Graphic)" include programs for which a graphic terminal is required.

1.3.1 Graphic Models Menu

Most items on this menu are described in this Guide or the *IBS User Guide*.

IBS Shell: Graphic Models

- 0 : EXIT
 - 1 : D2 Dispersion Model
 - 2 : IDYNEV Evacuation Model
 - 3 : CHEMS Chemical Model
 - 4 : OSPM Outdoor Sound Propagation Model
 - 5 : MESORAD Radiological Dispersion Model
 - 6 : PREDYN Update Evacuation Data
 - 7 : EESF Integrated Model
 - 8 : REA Regional Evacuation Analysis
 - 9 : BUILDNET Generate Evacuation Network
 - 10 : I..POP Reallocate Evacuation Population
-

Menu Choice (?=Help) ==>

1.3.2 Non-Graphic Models Menu

Most items on this menu are described in this Guide or the *IBS User Guide*.

```

      IBS Shell: Non-Graphic Models
-----
0 : EXIT
1 : D2      Dispersion Model
2 : IDYNEV  Evacuation Model
3 : CHEMS   Chemical Model
4 : OSPM    Outdoor Sound Propagation Model
5 : MESORAD Radiological Dispersion Model
6 : PREDYN  Update Evacuation Data
7 : EESF    Integrated Model
8 : REA     Regional Evacuation Analysis
9 : I..MERGE Merge Evac Assignment & Sim Case
10 : TTXTTDAT Extract TIGER Roads for Evac
-----
Menu Choice (?=Help) ==>

```

1.4 IBS User Environment

IBS maintains a **user environment**, which defines the overall setup of equipment, software, and data that you are currently using in IBS. The user environment affects both mapping (graphics) interactions and modeling functions.

1.4.1 Environment Files

IBS maintains two special files in the top-level directory of each user, **SITE.DAT** and **JOB_ENV.DAT**.

- **SITE.DAT** contains one record for each site in your **SITES** directory. Included in this record are the site name, four-character site code, disk location of data, longitude and latitude of site center, and radius of the site. Note that it is possible for you to have site data on more than one device if this is designated in the **SITE.DAT** file and if you have a top-level directory on the various devices so referenced.
- **JOB_ENV.DAT** contains information on what data you are currently (or were most recently) interested in and how you wish to interpret that data. Examples of such data:
 - Current site code
 - Current case numbers for each model
 - Current map control file names
 - Current parameter settings for viewing model contour output.

1.4.2 Changing the User Environment

You can change the user environment by selecting the *Setup* option from the IBS Shell: Main Menu, or from the IBS Main Menu. This option provides you with a consistent means of telling IBS what data you want to use and how you want to use it. You can also easily find data of interest and delete data that are no longer needed.

1.4.3 Using Case/Site Functions

Most of the model subsystems have a case/site function for switching between model input data sets. These functions typically display a screen form such as the one shown in Figure 1.3. The site name that appears on the form will be the current site and the initial case number will be the current (last-used) case number. You can change these fields to switch to a different site, switch to a different case, or create a new case.

To switch between sites: Enter the site name in the *Site Name* field and press GOLD-Z. The current case number will change to the last-used case for the new site.

Note: If you wish to change to a new site and select a specific case number for that site, it is necessary to perform these functions separately. Select the desired site first. The system will set the *current* model case number and the *Based on* case number to the highest case number found for the site you switch to. You can then select a specific case number for the current case.

To switch to a different case: For the *current* case number, enter the three-digit case number of an existing case. When you finish the form (by pressing GOLD-Z), the selected case will be the current case.

To create a new case (a new set of case data):

1. For the *current* case number, enter a three-digit case number (001-999) that has not been used for the specific model and site. (Case number 000 must not be used: case number 000 is a *base case* maintained by IBS for each model.)
2. For the *Based on* case number, enter the case number on which you would like to base the new case data. (The *Based on* number must be the number of an already existing case. If you ask for a new current case to be *based on* a nonexistent source case, the system will notify you.)

When you finish the form (by pressing GOLD-Z), IBS copies a model data set from the *Based on* case to create a data set for the new case number. Creating a new case requires a few moments after you have entered the changes.

3. Modify the model input data to make this new current case unique.

In this way you can generate 999 unique model cases (subject to disk space limitations), including both model input and output for each site.

When you are finished changing fields, press GOLD-Z to accept the changes and exit the form (or GOLD-Y to exit, restoring the original contents of the fields).

IDYNEV - Evacuation Model	Case Selection Form V2.00
Site Name: <u>TOOELE ARMY DEPOT</u>	
IDYNEV	Model Case: <u>4</u>
IDYNEV	Model Base: <u>4</u>

Figure 1.3. Example Change Case/Site Form

1.5 Using Mapping Functions within Model Subsystems

The IBS models are embedded in modeling subsystems that enable you to create input data sets (cases), execute the models, and obtain model results. Because IBS is a mapping graphics system, you often have the option to use mapping functions in creating model cases, displaying model results, or both.

Graphic or Text Mode Menus. When you select a model subsystem from the IBS Shell Main Menu, you can choose from Models (Graphics) or Models (Non-Graphic)

Displaying Site Maps. The graphic menus generally contain a DISPLAY BASE MAP option. By selecting DISPLAY BASE MAP on the main menu of any graphics model, you can obtain a submenu of standard Map Display functions for displaying site topographic data. (Map Display functions are explained in the *IBS Utilities Guide*.)

Displaying Model Data on Site Maps. Input and/or output data from the models can sometimes be interpreted graphically on a map of the site, making it easy to visualize a large amount of computed data. For example, the Mesorad model computes the dispersion and dose for airborne radionuclides, and the progress of the wind-dispersed plume of contaminants can be displayed graphically.

Graphic Editing with Site Maps. From the DISPLAY BASE MAP submenu you can select EDIT FUNCTIONS for adding graphics to the site map displays. As in Map Display, any graphic editing affects only a separate DMS *edit file*. However, you cannot specify a particular file as the edit file: instead IBS uses a predefined edit file that is associated with the current set of model input data (case data). That is, each model case can have a separate edit file.

Obtaining Hard Copies of Graphically Displayed Data. If you are using a standard IBS workstation (one of various Tektronix terminals), you can print copies of the site map and model graphics using the Tektronix color printer:

To obtain a COLOR hard copy of the terminal screen: Press <S Copy>.

To obtain a BLACK-on-WHITE hard copy of the terminal screen: you must first change the terminal display colors to white elements on a black background.

1. Press the <Menu> key (in the row of function keys above the standard keyboard keys).

The terminal will show a menu that defines some of the function keys for resetting the screen colors.

2. Press the <F1> key for resetting the color.

The cross hairs cursor will appear at the lower left corner of the display.

3. Use your terminal's graphics input device (joydisk or bit pad with mouse) to place the cross hairs on lines, background, or other graphic elements for which you want to change the color.
4. Use the <F1>, <F2>, and <F3> keys to change the **hue**, **lightness**, and **saturation** of the colors to be changed.

Depending on the hue, lightness can often be used to vary the colors from light (white) to dark (black). You can use this procedure to obtain whatever colors you wish.

5. When you are finished, press the <Menu> key again to remove the color setting menu.
6. Press the <S Copy> key.

The color copier will print the display in the colors you have selected. These color changes will remain in effect until you exit from one of the graphic display options. Upon exit, the standard colors (color table) will be restored.

To obtain a screen copy that includes text overlaid on graphics: This is not always possible, but you can try the following:

1. Within IBS, but before you generate the desired display, press the <SETUP> key to obtain the terminal's setup prompt (an asterisk).
2. Enter DAINDEX 1 0 0 <RETURN>.

Overview

3. Press the <SETUP> key to return to IBS.
4. Generate the desired display and then press the <S Copy> key.

Section 2

Regional Evacuation Analysis

IBSSH Menu: Select (Models (Graphic))

Natural Hazards Support Module: Select 6 (Models (Graphic)) and then 8 (Regional Evacuation Analysis)

This section describes **Regional Evacuation**, a capability for applying IBS's evacuation model to emergency situations in areas selected from the regional maps database. Regional Evacuation combines the IBS evacuation model (IDYNEV) with the IBS capability for generating a first-cut evacuation roadway network for the model. This section covers the following topics:

- 2.1 Description of the Regional Evacuation Model
- 2.2 What You Need to Use the Regional Evacuation Model
- 2.3 Starting a Regional Evacuation Model Work Session
- 2.4 Updating Demographic Data
- 2.5 Using the Regional Evacuation Main Menu
- 2.6 Generating a First-Cut Evacuation Case
- 2.7 Updating the Current Site Network (Model Input Data)
- 2.8 Displaying Site Map From Site File
- 2.9 Executing Evacuation Model - Batch
- 2.10 Executing Evacuation Model - Interactive
- 2.11 Displaying Evacuation Results
- 2.12 Printing/Displaying Reports
- 2.13 Changing Site Name, Set Code, Case Number
- 2.14 Case Site Directory and Optional Delete

2.1 Description of the Regional Evacuation Model

The IBS capability for analyzing Regional Evacuation integrates IBS functions for creating, modeling, and displaying evacuation cases.

The evacuation and traffic network model used by Regional Evacuation is the Interactive Dynamic Network Evacuation (IDYNEV) model, which predicts the progress of area evacuations by indicating the changing conditions of population movement over a traffic network. Several of the functions of the Regional Evacuation capability duplicate functions in EESF and other IBS subsystems: data input, model execution, results display. Other features of Regional Evacuation expand the evacuation modeling resources of IBS and permit the modeling of larger areas than those usually used in IBS.

A key feature of Regional Evacuation is the *first-cut* generation of a traffic network and evacuation case. When applied to a site extracted from a maps database, this feature uses roads and population information from the database to take a first cut at *creating* (assembling the data to define) the traffic network for the evacuation model. Although the resulting network is unlikely to be an accurate representation of the real roadways, you can use it as a starting point and refine the network to be as detailed as you wish.

The Regional Evacuation model can be used in planning and training for large-scale disaster situations that require evacuation from several hazard areas simultaneously (for example, the evacuation of several neighboring communities in the path of flood waters). The use of the Regional Evacuation capability will enable emergency management officials to predict how long it will take to evacuate residents from the various hazard areas, and when and where bottlenecks are likely to occur. The dynamic capability of Regional Evacuation analysis will enable planners to assess the impact of various remedial actions and to choose the best of these actions for inclusion in the region's planning documents.

The capabilities of Regional Evacuation as represented in this manual are a *snapshot* of a developing system. Regional Evacuation results should not be used as the sole basis for planning or decisions affecting public safety.

2.2 What You Need to Use the Regional Evacuation Model

The Regional Evacuation capability is essentially a combination of features found elsewhere in IBS. Although this section describes the general features of Regional Evacuation, other manual sections contain more complete operational details for some modeling and map display components:

<u>Regional Evacuation Section</u>	<u>Related Manual Section/Section</u>
2.4 Generate First-Cut Evacuation Case	Section 5
2.6 Update Current Site Network	Section 6, Section 6.4
2.8 Display Site Map From Site File	<i>IBS Utilities Guide</i> , MPDISPLAY

At the core of the Regional Evacuation capability is the evacuation model, IDYNEV (Interactive DYnamic Network Evacuation). The model produces several measures of evacuation effectiveness, including travel times, vehicle counts, queues and delays, and person throughput over the links (street segments) and nodes (intersections) of the network. This section describes how to apply the model to geographic regions that you specify. Reference to the evacuation model source documents will aid meaningful use of the model and interpretation of results. Section 6 of this manual includes an overview of IDYNEV and vocabulary that may be helpful.

References — Evacuation Model Documentation

E. Lieberman, B. Andrews, M. Davila, and M. Yedlin. 1983. Macroscopic Simulation for Urban Traffic Management: The TRAFLO Model. FHWA-RD-80, Federal Highway Administration, Washington, D.C.

E. Lieberman. 1984. Formulations for the DYNEV and I-DYNEV Traffic Simulation Models Used in EESF. TR-154. Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1984. Exercise Evaluation and Simulation Facility, Evacuation Events Model. Part I-PREDYN User's Guide. FEMA-REP-6, Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1984. Exercise Evaluation and Simulation Facility, Evacuation Events Model. Part II-Users Manual. FEMA-REP-7, Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1986. Description of the Integrated Trip Assignment and Distribution Model (TRAD) for the I-DYNEV System. TR-187, prepared by KLD Associates, Inc. for the Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1988. Updated User's Guide for the IDYNEV System Including the TRAD Model. TR-188, prepared by KLD Associates, Inc. for the Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1988. PREDYN User's Guide. TR-131A, prepared by KLD Associates, Inc., for the Federal Emergency Management Agency, Washington, D.C.

2.3 Starting a Regional Evacuation Model Work Session

A session is a continuous period of time during which you use the Regional Evacuation Model. You will start, control, and stop a session at your workstation. The session starts when you select Regional Evacuation Analysis from the Model Menu of IBSSH.

Overview. The program flow of functions in Regional Evacuation is generally divided into three parts:

- modification of evacuation input data (through PREDYNGR--the evacuation model data input program)
- modification or updating of the current population data set
- all other menu functions (automatic network generation, site map display, IDYNEV model execution, display of model results, printing or display of output reports, site and case control, and file maintenance).

2.4 Using the Regional Evacuation Main Menu

You will see one of the menus shown in Figures 2.3 and 2.4, depending on whether you selected graphics models or text menus when you began the session. Both menus list the basic Regional Evacuation functions and display information on the current site and population data set.

The *graphic mode* menu in Figure 2.3 includes graphic functions for generating a first-cut evacuation case, displaying the site map, and displaying evacuation results.

The *text mode* menu in Figure 2.4 includes an option for updating the current site network (evacuation model input data).

The Regional Evacuation functions are described in Sections 2.6 through 2.14.

```
EXIT REA

FIRST CUT APPROXIMATION      *

DISPLAY BASE MAP

EVACUATION MODEL - BATCH    *
EVACUATION MODEL - INTER.   *

SHOW MODEL RESULTS

CHANGE SITE                  *
CHANGE CASE
CASE MAINTENANCE

POPULATION MAINTENANCE
```

Figure 2.3. Main Menu for Regional Evacuation - Graphic Mode

```

                                Regional Evacuation Analysis Main Menu

0:  EXIT
1:  Update Network
2:  Evacuation Model - Batch
3:  Evacuation Model - Interactive
4:  Display Reports
5:  Change Site
6:  Change Case
7:  Case Maintenance
8:  Population Maintenance
-----
Menu Choice (? = Help) ==>

```

Figure 2.4. Main Menu for Regional Evacuation - Text Mode

2.5 Generate a First-Cut Evacuation Case - Graphic Mode Menu Only

Regional Evac Graphic Menu: Select First-Cut Approximation

Selecting *First-Cut Approximation* on the Regional Evacuation menu enables you to automatically create first-cut evacuation cases for traffic assignment and simulation for the current site. You will specify the case numbers for these cases on a subsequent form.

Table 2.1 summarizes the process of generating a first-cut evacuation case. Notice that this process requires roadway and population data. The evacuation case created by the process shown in Table 2.1 does not necessarily represent accurate or meaningful input to the evacuation model. Rather, this feature is a quick way to create a case and traffic network that can be modified later with more detailed input. This process is described in detail in Section 5, **Generate Evacuation Network**.

2.6 Update Current Site Network (Model Input Data) - Text Mode Menu Only

Regional Evacuation Text Menu: Select 1 (Update Network)

Select this option on the Regional Evacuation menu to update the traffic network for the current site. The system will turn you over to PREDYNGR, an interactive data input program that uses data forms and graphics displays to prepare input for the evacuation model.

This function duplicates the *Modify Evacuation Model Input Data* function of the Dynamic Evacuation Model. Refer to Section 6, Sections 6.4 and 6.5 for background, terminology, and procedures for modifying evacuation model input.

2.7 Display Site Map from Site File - Graphic Mode Menu Only

Regional Evacuation Graphic Menu: Select Display Base Map

Select this option on the Regional Evacuation main menu to display the current site map on the graphics display. This option uses the Map Display software described in *IBS Utilities Guide*, MPDISPLAY. Refer to that section for detailed descriptions of each option on the Map Display menu.

Table 2.1. Overview of Generating a First-Cut Network and Cases

What You Do	What IBS Does
1. Select A - Generate Evacuation Network on the Regional Evacuation Network	Presents the First Cut Case/Site Specification Form
2. Specify on the form: - the site name - % population that will evacuate - traffic assignment case # - traffic simulation case # - population set name - flood (SLOSH) output overlay.	Establishes % of potential evacuees who will evacuate. Asks you whether you wish to load case network parameter settings. (Parameters include the road types used in the network, free-flow speeds, etc. The network parameter settings are stored in a case-numbered file named <i>CnnnEVACPARAM.DAT</i> .) Presents the Map Display menu for displaying map features and information evacuate (<i>IBS Utilities Guide</i> , <i>MPDISPLAY</i>).
3. Select the topographies to be displayed, including a population topography. You should also select a roads topography if you are not going to use a Tiger Trails (TT) link extraction for roadway data. Use caution in selecting a road topography. Some topographies look complete when viewed over a large area, but lack the connecting roads necessary to build a network.	Retrieves and displays the topography data from the site files. Displays the graphic menu for network and case generation.
4. DEFINE EVACUATION BOUNDARY by graphically drawing a line around the evacuation area.	Creates a DMS file (<i>CnnnEVAC.DMS</i>) to hold the graphic boundary information for network generation.
5. EXTRACT LINKS FROM TIGER (optional)	Extracts links from Tiger Trails (TT) topography without requiring them to be brought into memory all at once. Also eliminates some low level roads and writes a short report on what it does: <i>TT_STATS.DAT</i> .
6. DEFINE HOST LOCATIONS (evacuation centers outside the boundary) that are capable of sheltering evacuees.	Adds host location information to the DMS <i>CnnnEVAC.DMS</i> graphics file.
7. QUALIFY ROADWAY SELECTION information (optional)	Updates the network parameter file to contain your qualifications.
8. GENERATE NETWORK based on site road data, roadway selection qualifications, evacuation boundary, and host locations.	Associates site population points with internal link locations (called centroids) where vehicles enter the network. Associates each host location with exit nodes on the evacuation boundary (estimates turns that vehicles could take to exit the evacuation area).
9. Pick GENERATE EVAC CASES on the menu, specify the number of people per vehicle, time interval, load time, and load curve.	Converts population values to a number of vehicles. Establishes the number of vehicle trips per hour originating at each centroid. Generates the input data for two initial cases: one traffic assignment case and one traffic simulation case.
10. Pick ENABLE ROADWAY NETWORK	Graphically displays the roadway network.

Note: The Map Display software enables you to store any graphic editing that you might choose to do through the *EDIT FUNCTIONS* option of Map Display. Your editing is stored in a separate *edit file*. Many IBS modeling applications edit files that are specific to each model case number. This simplifies inter-model and inter-case exchange of graphic objects derived from model input/output.

2.8 Execute Evacuation Model - Batch

Regional Evacuation Menu: Select 2 (*text mode menu*) or **Evaluation Model-Batch** (*graphic mode menu*)

This option displays a check list for the current IDYNEV case (case number, site name, run type, etc.) and the following prompt:

Execute IDYNEV for this case? (Y/N) [N] = >

Read the check list to be sure that the case is set up to run as you wish. Then respond to this prompt.

- If you answer N (no), you will return to the Regional Evacuation menu without executing the model.
- If you answer Y (yes), the model will execute in *batch* mode, using data defined for the current site and IDYNEV case number.

Batch execution of the model is convenient for larger cases. In batch mode, the system executes the model separately from your current interactions with IBS. While the model is running in batch mode, the system returns control to the Regional Evacuation top-level menu. You can then use other functions, or even exit. However, do not select any IDYNEV interactions while the model is running; these could interfere with the model run.

Refer to Section 6 for specific information on the evacuation model.

2.9 Execute Evacuation Model - Interactive

Regional Evacuation Menu: Select 3 (*text mode menu*) or **Evacuation Model - Inter** (*graphic mode menu*)

This option displays a check list for the current IDYNEV case (case number, site name, run type, etc.) and the following prompt:

Execute IDYNEV for this case? (Y/N) [N] = >

Read the check list to be sure that the case is set up to run as you wish. Then respond to this prompt.

- If you answer N (no), you will return to the Regional Evacuation menu without executing the model.
- If you answer Y (yes), the model will execute in *interactive* mode, using data defined for the current site and IDYNEV case number.

In interactive mode, the system executes the model as part of your current process: you will have to wait until the model is finished before you can continue any further operations. When the model is through running, the system will inform you that IDYNEV execution is complete and then return control to the Regional Evacuation menu. For large cases, execution in batch mode (Section 2.8) may be more convenient.

Stopping the Model. No clean process halts the evacuation model while it is running. Pressing <CTRL Y> is a standard system interrupt command and will halt model execution, but presents no guarantee where you will regain control. The model can be restarted only from the beginning of its execution.

Summary Report. If the model is executed for a traffic simulation, and if one or more time steps of the simulation are completed, the system displays a summary report of the evacuation following model completion. This report contains various statistics extracted from the last simulation time period reported by the evacuation model.

2.10 Show Model Results - Graphic Mode Menu Only

Regional Evacuation Graphic Menu: Select **Show Model Results** (Display Evacuation Results)

Use this option to graphically display results of the current evacuation case. When you first select this option, you will see messages about processing Evacuation Model Impact files. Then, the menu in Figure 2.5 will display. You may also be given the opportunity to select map background layers if you have not already done so.

IEMIS will draw the site map on the graphics screen and present the graphics menu. This menu is the same one used for the *Display Results on Map* function described for the evacuation model in Paragraph 6.6.3, Section 6. Refer to that portion for details on displaying evacuation graphics.

<i>identification</i>	<div style="border: 1px solid black; padding: 5px; display: inline-block;"><p>Site Model Case # EXIT</p><p>HELP</p><p>CONTINUE</p><p>DESCRIBE NODE DESCRIBE LINK DESCRIBE EXIT NODE DESCRIBE INT. CENTROID DISPLAY FLOOD WATER</p><p>RETURN POPULATION DETERMINE AREA ZOOM IN ZOOM OUT RESTORE PLOT</p><p>RUN RESTART STEP FORWARD STEP BACKWARD SELECT TIME STEP ----Hours, ----Minutes TEXT DISPLAY ENABLED ICON DISPLAY ENABLED REFRESH SCREEN</p></div>	<p><-- Site and case number</p>
		<p><-- Time Period label appears when model results are run or stepped.</p>

Figure 2.5. Graphics Menu for Display Evacuation Results

2.11 Display Reports

Regional Evacuation Menu: Select **4** (*text mode menu*) or **Display Reports** (*graphic mode menu*)

This option displays a menu (Figure 2.6) with options for printing or displaying the IDYNEV evacuation report. This large report on the modeled evacuation is the only report for the evacuation model. The output is designed for a high-speed line printer, so you may need to set your terminal screen for a greater width.

```
=====
P.  Print file on printer
D.  Display file on terminal
X.  Exit
=====
Choice ==>:
```

Figure 2.6. Menu for IDYNEV Print/Display Reports Option

2.12 Change Site

Regional Evacuation Menu: Select **5** (*text mode menu*) or **Change Site** (*graphic mode menu*)

Select this option on the Regional Evacuation menu to select a site from a menu of sites available to you. The standard IBS site-selection menu will be presented.

2.13 Change Case

Regional Evacuation Menu: Select **6** (*text mode only*) or **Change Case** (*graphic mode only*)

Select this option on the Regional Evacuation Menu to change entries displayed in the Case Selection Form, shown in Figure 2.7. You can change case numbers and population case names to access different data.

To change case identification entries:

1. Press the up, down, left, and right arrow keys (<F1> - <F2>) to position the input cursor in the correct field.

Cursor movements do not affect any entries already specified.

2. Type in the new entries. Pressing <RETURN> will advance the cursor through the input fields.

REA - Regional Evacuation Analysis

Case Section Form V2.00

Site Name: _____

Evacuation Model Case: _____

Evacuation Model Base: _____

Population Case Name: _____

Population Base Name: _____

Figure 2.7. Site/Case Control Information

New entries will replace old entries in the input fields, but will not be in effect until after you press <GOLD> <Z> in Step 3.

- Population Case Name: Enter a 1- to 20-character name that identifies the site-dependent population set to be used in making a first-cut approximation of the evacuation case.
 - Population Base Name: The name of the population set used in creating a new current population set.
 - Evacuation Model Case: 000 to 999.
 - Evacuation Model Base: The code of the evacuation case used as the basis for creating a new current case.
3. Press <GOLD> <Z> (the GOLD <F5> key followed by "Z") to put the changes into effect. (To escape without making changes, press <GOLD> <Y>).

2.14 Case Maintenance

Regional Evacuation Menu: Select 7 (*text mode menu*) or **Case Maintenance** (*graphic mode menu*)

Selection of this function from the Regional Evacuation menu will display the menu shown in Figure 2.8. The menu options enable you to list the directory of evacuation cases for the current site. You also may delete selected input and output files from the system (if you own the files). Exercise care here, especially in deleting input files, because re-creation of files can be time consuming.

```

                                PERFORM FILE MAINTENANCE

Selection
Designator   Selection

  A          Display Directory of Cases
  B          Delete Case Output for a Specified Case
  C          Delete Input and Output for a Specified Case
  X          Exit to Previous Menu
=====
Enter selection designator:

```

Figure 2.8. Menu for Case Maintenance

2.15 Population Maintenance

Regional Evacuation Menu: Select 8 (*text mode only*) or **Population Maintenance** (*graphic mode only*)

Selection of this function from the Regional Evacuation Menu will display the menu shown in Figure 2.9. Use these functions to review, copy, or delete population case files. Population case files contain the information that associates the traffic network with specific population points and populations.

```

Population Management Menu
-----
0 : EXIT
1 : Select population set
2 : Add population set
3 : Delete population set
4 : List population sets
5 : Create population set (from _____)
6 : Select base population set (_____)

Menu Choice (? = Help) ==>

```

Figure 2.9. Menu for Population Maintenance

Section 3

D2 Chemical Hazard Prediction Model

IBS Menu: Select **EMERGENCY ACTIVITIES—HAZARD ANALYSIS—EXECUTE D2**
D2 Dispersion Menu: Select **EXECUTE D2**

Note: The D2 model can also be started independently by typing *D2_GR* or *D2_NOGR* at the VAX system prompt.

This section supports the IBS implementation of the D2 chemical hazard prediction model by providing details about the model inputs. This information was obtained from model documentation and examination of the model code. This section covers the following topics.

3.1 Description of D2

3.2 Modifying D2 Model Input Data

3.1 Description of D2

The **D2 Dispersion Model** option on the **IBS Shell: Graphic Models** menu displays the following graphic screen menu of "D2 RUN Capabilities."

The **EXECUTE D2** menu includes options for creating a new case (based on an existing case), modifying the input, executing the model, and maintaining the model cases (input and output data).

EXIT	
D2 DISPERSION MENU	
<hr/>	
CONTINUE	
MAP ANALYSIS	->
CHANGE TRACK CASE	*
EXECUTE D2	*
DOSE DETAILS	
DESCRIBE TRACK	
PRINT D2 REPORT	
REPORT CURRENT D2 DESCRIPTION	
ANIMATE D2 TRACK	
SHOW ELEVATION	
DETERMINE AREA	
DISPLAY LOCATION	
ESTIMATE POPULATION	
ZOOM IN	
ZOOM OUT	
REFRESH SCREEN	
RELEASE OVERLAYS	

D2 DISPERSION supports the use of D2, the Army's modeling program for predicting the downwind hazard resulting from the release of a chemical agent. The output of the D2 model is expressed graphically on the map screen as track contours of dose or concentration.

D2 DISPERSION presents this menu of options (described in alphabetical order).

When you begin **D2 DISPERSION**, the map screen automatically displays the default map layers associated with this function. These map layers should include **POPULATION** and **D2 TRACK** for the current D2 case. If the map layers do not

display, you can add them through the **IBS MAP ANALYSIS** option or add them to your system defaults by using the **SETUP** option in IBS.

D2 Track. The D2 output track on the map screen consists of contour lines. Each contour represents a level of dose or concentration (depending on the original inputs to the D2 model). A series of asterisks (*) appearing along the track center line indicate slice points along the track where the **DOSE DETAILS** option can be used to display dose information about those locations. Some releases do NOT result in the generation and appearance of these points.

3.2 Modifying D2 Model Input Data

To work with and modify the D2 model, select **EXECUTE D2** from the menu. This option displays the Execute D2 Menu Options menu. The menu includes options for creating a new case (based on an existing case), modifying the input, running the model, and maintaining the model cases (input and output data).

Caution

The D2 model inputs are easy to change; however, the knowledge of *what* those changes will be is left up to you.^(a) For other than testing purposes, you probably should NOT change D2 model input unless you have a clear understanding of the model. The many input parameters are interrelated in non-straightforward ways. Also, interpret the D2 model results cautiously. The D2 model itself is unchanged from previous implementations: data that caused aborted runs in the D2PC model will do so in the IBS version as well.

Execute D2 Menu Options

```
-----
0 : EXIT
1 : Create New D2 Case
2 : Modify D2 Case Input
3 : Run Current D2 Case
4 : Perform File Maintenance
-----
```

Menu Choice (?=Help) ==>

(a) See Innovative Emergency Management, Inc. and M.M. Myirski. 1993. *Reference Manual: D2PC and Hazard Analysis*. DAHC35-92-D-0018. Chemical Division, U.S. Army Nuclear and Chemical Agency (USANCA), 7500 Backlick Road, Building 5073, Springfield, Virginia 22150-3198.

D2 Standard Input

The options of the Execute D2 Menu Options menu are described in the *IBS User Guide, Section 11, Emergency Activities* under the heading **HAZARD ANALYSIS**. The discussion of input options is expanded here.

D2 STANDARD INPUT FOR CASE #200			
SOURCE DATA		LOCATION	
Agent :GB	2-Min Correct: Y	D	Location: DPG
Munition:105	# Munitions: 1.0		Long:-112.35702
Release :SEM			Igloo: 1101
			Lat : 40.32125
MET DATA			
Quantity:	Time:		
1 1.3608E+08 mg	20. min		
Source Sigma X:	1.9	D	
Source Sigma Y:	1.9	D	
Source Sigma Z:	0.6	D	
Additional Input:	Release Num:	1	
			Met Change Period: 1 Latest Met:

Enter a text D2 description here: D2 case #200, Agent GB,
Release type Semi-continuous, Temperature 41.0 F, Wind speed 6.6 ft/s

To view information about the possible inputs on this screen, press ? (or the HELP key) at each field.

The various fields on the D2 Standard Input screen are defined as follows:

- **D2 STANDARD INPUT FOR CASE #:** the IBS D2 case number.
- **Location:**

DPG	Dugway Proving Ground and Tooele Army Depot
UAD	Umatilla Army Depot
AAD	Anniston Army Depot
EWA	Edgewood Area, Aberdeen Proving Ground
JHI	Johnson Island
LBG	Lexington-Blue Grass Army Depot
NAP	Newport Ammunition Plant
PBA	Pine Bluff Arsenal
PAD	Pueblo Army Depot
RMA	Rocky Mountain Arsenal
EUR	USAEUR
- **Longitude and Latitude:** a decimal number of degrees
- **Munition:**

105	105-mm Cartridge, M60, M360
155	155-mm Projectile, M110, M121A1
8IN	8-in. Projectile, M426
500	500-lb Bomb, MK94
750	750-lb Bomb, MC-1
M55	155-mm Rocket, M55
525	525-lb Bomb, MK116
139	Bomblet, M139
M23	Land Mine, M23
4.2	4.2-in. Cartridge, M2A1
TON	Ton Container
TMU	TMU-28/B Spray Tank
NON	Nonmunition

D2 Chemical Hazard Prediction Model

- **Agent:**

DM	Adamsite	ZS	Ethanol	CG	Phosgene
KB	Amine	EG	Ethyl GB	PR	Pyridine
CK	Cyanogen Chlor	HY	Hydrazine	GB	Sarin
DC	Dichloro	AC	Hydrogen Cyanide	HT	60% HD and 40% T
DF	Difluoro	BZ	Incap Agent	GD	Soman
HD	Distilled Must	IP	Isopropanol	GA	Tabun
GF	EA 1212	LL	Lewisite	TC	Thionyl Chloride
VX	EA 1701	H1	HN-1 Nitrogen Must	UD	UDMH
QL	Ester	H3	HN-3 Nitrogen Must	NA	User Definable

- **Release Type:**

INS	Instantaneous (explosive)
EVP	Evaporative from a puddle formed by a spill
SEM	Semi-continuous. Constant for a finite time
VAR	Variable. The source is defined as the number of release intervals, each constant for a finite time (maximum of 6).
STK	Release of heated effluent from a stack
STJ	Release from a stack when inertial or jet effect dominates.
FLS	Flash fire from ground level
FIR	Fire burning for a finite time
IGL	IGL fire for M55 with GB or VX
EVS	Still-air evaporative

Note: When choosing EVP, you must also specify TIME OF EVAP (see supplementary inputs).

- **Spill or airborne:** For evaporative release, this is the amount that is initially spilled. The EVAP program takes this Q and then computes Q prime, which is the amount that becomes airborne. For other forms of release, this is the total amount that is airborne. The units are "mg," "g," or "lb" in the units field.

- **Stability Type:**

A	Very unstable
B	Unstable
C	Slightly unstable
D	Neutral
E	Slightly Stable
F	Stable
U	Undefined
S	Select stability from meteorological observations
W	Wooded areas

Note: When you choose U, S, or W, you will be prompted for additional input after you exit the standard input form.

- **Description:** an 80-character description of the D2 case.
- **Two-Minute Correction:** Y (yes) or N (no)
 N Do not use 2-minute correction with GB and VX vapor
 Y Use 2-minute correction with GB and VX vapor (Default)
- **Surface:**
 GRA Gravel
 NPR Concrete
 NDF Not defined. Specify wetted area.
- **Woods Type:**
 DW Deciduous, winter
 MW Mixed, winter
 CF Coniferous forest
 MS Mixed, summer
 RF Rain forest

D2 Supplementary Input

The D2 Supplementary Input screen enables you to change some basic D2 input parameters that are listed as shown in the following (expanded) screen example.

D2 SUPPLEMENTARY INPUT FOR CASE #200

Assess: Dosage (mg-min/m ³)		From: D2 Default Levels		Num Levels: 3	
Num Dosage Levels		Dosage Level Descriptions			
1	0.5 mg-min/ ³	D.5	NO EFFECTS		
2	6.0 mg-min/ ³	D6	NO DEATHS		
3	10. mg-min/ ³	D10	1% LETHALITY		
Vapor Depletion	: OFF	N	Roughness Length	: 0.	N
Slope of Frost Wind	: 0.	N	Settling Velocity	: 0.	N
Breathing Rate	: 25.	N	Sampling Height	: 0.	N
Skin Factor	: 0.	N	Height of Source	: 0.	N
Reflection Coefficient	: 1.	N			

In D2 you can assess either dosage or concentration. The top half of this form enables you to specify what to assess, how many levels to assess, and what levels should be assessed.

When assessing dosage, you can use the three default dosage levels, 1% lethality, no deaths, and no effects, or you can specify your own levels.

The bottom half of this form enables you to access some miscellaneous variables which are not changed very often. To the right of each variable listed at the bottom of this form is a source flag. If you do not wish to specify any value for a

D2 Chemical Hazard Prediction Model

given variable, type an *N* (for Not defined) in the source flag field of that variable. If you do want to input a value for one of these variables, type a *U* (for User defined) in the source flag field for that variable. Unless you have a good understanding of these inputs into the model, it is recommended that all the source flags at the bottom of this form be set to *N*.

Section 4

Model Library — MESORAD Model

IBSSH Menu: Select (Models (GRAPHIC)) or (Models (NON-GRAPHIC))
Model Menu: Select (MESORAD)

This section describes the MESORAD (Meso-scale Radiation) model, a computer model that combines atmospheric dispersion, deposition, and decay of radiological materials with dose modeling logic. This section covers the following topics.

- 4.1 Description of MESORAD
- 4.2 What You Need to Know to Use MESORAD
- 4.3 Using the MESORAD Top-Level Menu
- 4.4 MESORAD Graphic Options
- 4.5 Update Elevation
- 4.6 Update Case Data
- 4.7 Display Base Map
- 4.8 Print/Display Reports
- 4.9 Perform File Maintenance
- 4.10 Execute MESORAD - Interactive
- 4.11 Execute MESORAD - Batch
- 4.12 Change Case/Site
- 4.13 If You Have Problems
- 4.14 Status and Diagnostics Messages
- 4.15 MESORAD/FEMA

For references, see **Section 9.21**.

4.1 Description of MESORAD

MESORAD is an extension of MESOI, an atmospheric dispersion model that includes deposition and decay of radiological materials. The MESOI model was enhanced with dose modeling logic to create MESORAD. The implementation of MESORAD in IBS combines a subset of MESORAD's capabilities with added capabilities for creating special dose reports and for calculating doses that are due to ingestion pathways.

Generally, this manual describes the use of these models without explaining their theoretical bases. Meteorologists and other technical experts concerned with creation of specific cases can refer to the source documents for detailed explanations of the models.

4.1.1 Map-Related Display of Modeling Results

MESORAD calls on the IBS map database and Map Display software for graphical display of modeling results overlaid on background maps. For example, you can display meteorological (met)/dose model results over the area for which the model was run. You can specify what static features to display: political boundaries, waterways, etc. Then you can customize the model results to be displayed by selecting dose type and other information such as the windfield and the path of the effluent plume(s). This interactive graphics display is a powerful tool for understanding the results of the technical models.

4.2 What You Need to Know to Use MESORAD

The materials in this section are intended for any user of the MESORAD model. The section includes the steps you must take in using MESORAD to simulate radiation release scenarios that might be used in emergency response exercises and training. MESORAD is designed to step you through the creation of these scenarios, but you must be familiar with the data entry conventions and user interactions described in the *IBS User Guide*, Section 1.

You can use this section as a step-by-step guide through the MESORAD functions. After you have selected MESORAD from the Model Library menu, MESORAD can supply background information with its HELP features (described in Section 4.13).

Although any user can use the MESORAD computer model, the model results will be only as meaningful as the input. Some static details, such as the physical characteristics of a site, would rarely be changed by a typical user. Other inputs would require the technical expertise of a meteorologist. Generally, a system manager and technical consultants will set up standard cases the typical user can

modify in less fundamental ways. If you have problems with MESORAD, refer to Section 4.13 (**If You Have Problems**) and Section 4.14 (**Status and Diagnostics Messages**).

4.2.1 Meteorology Model

The meteorology portion of MESORAD estimates the transport, diffusion, deposition, and decay of radioactive materials released to the atmosphere. The model is capable of treating simultaneous releases from as many as four different elevations at the point of release (site center). Puffs generated at the release points are affected by a three-dimensional, horizontal windfield that takes into account the topology of the area. The concentrations within the released puffs are initially assumed to be distributed normally in both the horizontal and vertical directions. Horizontal concentration distribution is modified by the windfield. Vertical concentration distribution is modified by assuming reflection at the ground and at the top of the atmospheric mixing layer.

Inputs to the model include parameters such as time intervals, atmospheric stability, wind speeds and directions, height of the mixing layer, surface and upper layer temperatures, dose check point locations, terrain and topographic features, and locations of weather stations. Section 4.6 describes data input forms for modifying input to the met/dose model.

The meteorology model calculates an integrated, two-dimensional windfield in the simulation area and creates files that define the concentration and deposition fields over time. This information is subsequently used by the dose part of the model.

4.2.2 Dose Model

The dose model calculates radiological doses simultaneously with the atmospheric dispersion and depletion calculations for as many as 50 radionuclides. Dose calculations include current and cumulative values for external doses to the whole body and for internal doses to critical organs (lungs, thyroid) and the whole body.

The model uses evacuation model results so that the options of evacuation and sheltering can be compared. Sheltering factors for portions of the population can be entered, and the model will account for exposures during the evacuation.

Once the desired models have been run, IBS provides an interactive graphic display environment (Display Base Map) for viewing the results.

4.3 Using the MESORAD Top-Level Menu

Model Library Menu: Select 5 (MESORAD)

Depending on your selection of Graphic or Text menus, you will see one of the MESORAD top-level menus shown in Figures 4.1 or 4.2.

EXIT	
HELP	
MESORAD GRAPHIC OPTIONS	<i>Explained in</i>
UPDATE ELEVATION	<u>Section</u>
UPDATE CASE DATA	4.4
DISPLAY BASE MAP	4.5
	4.6
	4.7
PRINT REPORTS	4.8
DISPLAY REPORTS	4.8
PERFORM FILE MAINT.	4.9
EXECUTE MESORAD INTER.	4.10
EXECUTE MESORAD BATCH	4.11
CHANGE CASE	4.12

Figure 4.1. MESORAD Top-Level Menu - Graphic Mode

The special options of the graphic menu are:

- MESORAD GRAPHIC OPTIONS
- UPDATE ELEVATION
- DISPLAY BASE MAP.

The other options are shared in common with the MESORAD top-level text menu.

In a typical work session, you might update case data to change MESORAD input, execute the model, review reports from the modeled case, and graphically display results on a site map. (Only the graphics menu allows you to display results on a site map.) MESORAD can be executed either interactively (while you wait) or as a "batch" program (start it and move on to other tasks).

```
NON-GRAPHIC MESORAD DRIVER MAIN MENU

0 : Exit
1 : Print/Display Reports
2 : Update Case Data
3 : Perform File Maintenance
4 : Execute MESORAD - Interactive
5 : Execute MESORAD - Batch
6 : Change Case
-----
Menu Choice (? = Help) ==>
```

Figure 4.2. MESORAD Top-Level Menu - Text Mode

To start a MESORAD function on the text menu, type the corresponding number and press <RETURN>.

Sections 4.4 through 4.12 explain the top-level menu options in further detail.

4.4 MESORAD Graphic Options - Graphic Menu Only

Picking MESORAD GRAPHIC OPTIONS from the MESORAD top-level menu will display a menu of **Graphical Mapping Options** (Figure 4.3) for displaying MESORAD output. The menu offers a choice of the static and dynamic mapping features that can be displayed. (For a summary of how to use this and related menus to display MESORAD graphics, refer to "Displaying MESORAD Map-Related Graphics," on the second page following the end of this menu description.)

- * Specify Limits for Graph Data Selection: Entering a "Y" in this entry field will cause a form of "Global Graphics Parameters" to appear after you finish the current menu. This form is shown in Figure 4.4.
- * Dynamic Features: Select one or both of the dynamic features by entering "Y" in the entry field in front of the feature name. Windfield and Initial Particle path are dynamic because they change with each model time step.
- * Grids: Select any of the grids by entering a "Y" in the entry field in front of the grid name. The several types of grids are static graphic "overlays" and do not change, once they are drawn.

MESORAD GRAPHICAL MAPPING OPTIONS

_ Specify Limits for Graph Data Selection ?

Dynamic Features:		Curr	Cumul:
- Windfield	-	-	Relative Plume Concentration
- Initial Particle Path	-	-	Relative Deposition
	-	-	Dose - Total Whole Body
	-	-	Dose - Thyroid
Grids:	-	-	Dose - Inhalation/Whole Body
- 10 Mile EPZ (Polar)	-	-	Dose - Deposition
- 16x16 Meteorolog. (Square)	-	-	Dose - Overhead Plume
	-	-	Dose - Lung
- 31x31 Dose	-	-	Iodine
- 3x36 Dose	-	-	Gamma
	-	-	Gamma + Beta

Press GOLD <F5> followed by 'Y' to return to MESORAD menu.

Figure 4.3. MESORAD Graphical Mapping Options Menu

- * Curr(ent) or Cumul(ative statistics): Select a maximum of two types of output statistics for display and reporting by entering a "Y" in one or both of the two entry fields in front of each item. "Curr(ent)" refers to the current time step; "Cumul(ative)" refers to totals since the start. Select only those statistics that have case output (see Section 4.6.10).

— **Changing Global Graphics Parameters** (Form in Figure 4.4)

- * **Maximum Wind Speed:** Enter a wind speed value (mph). When you view the windfield, the wind speed vectors (shown on a 16 x 16 meteorological grid) will be scaled (in length) to this maximum wind speed. Actual wind speeds larger than this will be displayed as this maximum value.
- * **Start/End Dates and Times:** Enter dates and times. The earliest results available for display will be after the first 15-minute time step (that is, at start time plus 15 minutes).
- * **Time Increment:** Enter a value corresponding to the smallest time step (15 minutes) or some integer product of the smallest time step.
- **Title:** This display is a list of the calculated values that can be displayed. When you move past the last item shown on the screen, the list will "scroll" up (five at a time) to reveal more calculated values.
- * **Minimum:** For each type of calculated value that you wish to display as contours (isopleths of the calculated value), enter a minimum value to represent the outer contour of the set of contours for that calculated value. (You may EITHER specify a nonzero minimum here OR choose to enter contour levels manually: see "User Detailed Levels" following.)
- * **Number of Contours:** Enter the number of contours to be shown for the associated parameter (between the release point and the minimum value).
- * **User Detailed Levels:** Enter a "Y" across from each item for which you want to manually specify a nonzero number of contour levels. This will cause separate Contour Specification forms (Figure 4.5) to be displayed for those items. (If you choose to specify contour levels, be sure to specify as many levels as you entered for the "Number of Contours.")

GLOBAL GRAPHICS PARAMETERS									
MAX. WIND SPEED		(MPH)	<u>30</u>						
START DATE/TIME		END DATE / TIME		TIME INCREMENT					
				DAYS	HOURS	MINUTES			
<u>01-JAN-88 13:15</u>		<u>01-JAN-88 20:00</u>		<u>00</u>	<u>00</u>	<u>15</u>			
NUMBER OF USER									
DETAILED									
TITLE				MINIMUM	CONTOURS		LEVELS?		
CURRENT DEPL PLUME CONC.REL QTY-HR/M**3				0.0000E+00	<u>03</u>		<u>Y</u>		
CUM. DEPL. PLUME CONC. REL QTY-HR/M**3				0.1000E-03	<u>05</u>		—		
CURRENT DEP. CONC. REL QTY/M**2				0.1000E-03	<u>05</u>		—		
CUM. DEP. CONC. REL QTY/M**2				0.1000E+00	<u>05</u>		—		
CUR. TOT. WBDR - T+K+M MILLIREM/HR				0.1000E-08	<u>05</u>		—		

Figure 4.4. Example of Global Graphics Parameters Form

CONTOUR SPECIFICATIONS	
TITLE: CURRENT DEPL PLUME CONC.REL QTY-HR/M**3	
CONTOUR NUMBER	CONTOUR VALUE (UNITS)
1	0.1000E-03
2	0.1500E-03
3	0.2000E-03
4	0.0000E+00
5	0.0000E+00
6	0.0000E+00
7	0.0000E+00
8	0.0000E+00
9	0.0000E+00
10	0.0000E+00

Figure 4.5. Example of Contour Specifications Form

For each calculated value for which you choose to enter "User Detailed Levels," you will see a form similar to the one shown in Figure 4.5 (in this case for "Current Depl Plume Concentration"). This type of data entry form allows you to specify values for each contour that will be displayed to show the concentration of the dispersing plume.

Displaying MESORAD Map-Related Graphics. You can display MESORAD graphic options after producing output by running the met/dose model. You CANNOT display met/dose model results for a case that you have deleted: output files are deleted as well. To display map-related graphics:

1. Select the appropriate options from the MESORAD Graphic Options menu (Figure 4.3) and then press <GOLD> <Z>.

Note: For PC keyboards use <F5> in place of the <GOLD> key.

The IBS Map Display software is then executed so that you can create a background over which the model results will be displayed. See MPDISPLAY in the *IBS Utilities Guide* for more detailed information.

2. Use the SELECT TOPOGRAPHIES function on the Map Display menu to choose the topographic features for your background map.
3. Exit the topography selection and Map Display menus.

When you exit the topography selection menu, the base map is displayed. When you exit the Map Display menu, the MESORAD Temporal Graphics menu shown in Figure 4.6 is displayed.

```

EXIT
HELP
CONTINUE
RUN
STEP
REPORT DOSE
  
```

Figure 4.6. MESORAD Temporal Graphics Menu

4. Use the graphics input device attached to your workstation (the joydisk on your keyboard, or a bit pad with a mouse) in selecting options on the MESORAD Temporal Graphics menu. (Graphics input devices are described in the *IBS User Guide*).
5. After you have displayed the desired met/dose model results, pick EXIT on the MESORAD Temporal Graphics menu to return to the MESORAD Graphic Options menu.
6. Repeat steps 1 through 5 with different options and parameters, or press <GOLD> <Y> to return to the main MESORAD menu.

Each option of the MESORAD Temporal Graphics menu is described here:

- **EXIT:** This option exits the graphics display and returns you to the MESORAD Graphics Options menu.
- **HELP:** Provides text display similar to this page.
- **CONTINUE:** Signals the completion of functions. Picking CONTINUE is an affirmative response to any outstanding request.
- **RUN:** This option displays model output, showing changes as the output steps from the first time step to the last.
- **STEP:** This option displays model results one step at a time (that is, for the next sequential time step.)
- **REPORT DOSE:** This option allows you to pick a point on the map display and return dose report output associated with that location.

Dose report output for a point is more than one screen of text. Be prepared to use <CTRL S> and <CTRL Q> to stop and restart the text as it scrolls by on your screen.

4.5 Update Elevation - Graphic Menu Only

Selecting the UPDATE ELEVATION option produces a menu of elevation functions for map display. Refer to the IBS Utilities Guide under MPDISPLAY for details on these functions.

4.6 Update Case Data

MESORAD Text Menu: Select 2 (Update Case Data)

Choosing UPDATE CASE DATA from the MESORAD top-level menu activates a second menu for choosing data forms for updating case input. Each form can be used to change parameters of the met/dose model. The text and graphic menus for choosing the data input forms are shown in Figures 4.7 and 4.8.

Select the form you want to inspect/change by picking the appropriate item on the form selection menu. The forms are explained in Sections 4.6.1 through 4.6.10.

Choosing EXIT will return you to the MESORAD top-level menu.

Frequently Used Options. Note that in designing radiation release scenarios, you will use these five options most frequently:

- Change Control Parameters
- Change Source Details
- Change Weather Data
- Display Weather Dir(ectory)
- Change Output Options.

Less Frequently Used Options. You will probably want to change checkpoints less frequently, and to generate the Cartesian and Terrain grids only if the length of the side of the grid is changed:

- Change Checkpoints
- Change Terrain
- Generate Cartesian Grid (graphic menu only).

Site Setup Options. The remaining selections are normally used by experienced users or by the System Manager during the setup of a specific site:

- Change Station Definitions
- Change Cartesian Grid
- Change Polar Grid
- Generate Polar Grid (graphic menu only).

Meteorological/Dose Model Parameter Change

The change option menu is as follows:

		<i>Explained in Section</i>
A	Change Control Parameters	4.6.1
B	Change Source Details	4.6.2
C	Change Weather Data	4.6.3
D	Change Checkpoints	4.6.4
E	Change Station Definitions	4.6.5
F	Change Terrain	4.6.6
G	Change Cartesian Grid	4.6.7
H	Change Polar Grid	4.6.8
I	Display Weather Directory	4.6.9
J	Change Output Options	4.6.10
X	Exit	

Enter option selection:

Figure 4.7. Menu for Selecting Forms for Changing Parameters in the Met/Dose Model - Text Mode

EXIT	
HELP	
CONTINUE	<i>Explained in Section</i>
CHANGE CONTROL PARAMS	4.6.1
CHANGE SOURCE DETAILS	4.6.2
CHANGE WEATHER DATA	4.6.3
CHANGE CHECKPOINTS	4.6.4
CHANGE STATION DEF.	4.6.5
CHANGE TERRAIN	4.6.6
CHANGE OUTPUT OPTIONS	4.6.10
DISPLAY WEATHER DIR.	4.6.9
GENERATE POLAR GRID	4.6.11
GENERATE CARTESIAN GRID	4.6.12
CHANGE POLAR GRID	4.6.8
CHANGE CARTESIAN GRID	4.6.7

Figure 4.8. Menu for Selecting Forms for Changing Parameters in the Met/Dose Model - Graphic Mode

Usually these inputs would be changed only when the site changes (for example, when a new weather station is added), or perhaps when improved terrain or topography data become available. The options to Generate Polar Grid and Generate Cartesian Grid are graphic mode options that need to be done only once at site setup; they require Terrain Analysis Cell (TAC) elevation data as input.

4.6.1 Change Control Parameters

When you select this item from the form selection menu, you will obtain a form that will allow you to change the basic control parameters of the met/dose model. This form is shown in Figure 4.9. The data fields are explained below.

- * **Run Title:** Enter up to 50 characters in each of two fields for a title to describe the run. Each case should have a unique title. Include the author's name and the date of creation or update. This title is displayed on printed reports and on the case file directory that you can select from the MESORAD top-level menu.
- * **Scenario Start Date:** Enter day-month-year, using a three-letter abbreviation for the month.
- * **Scenario Start Time:** Enter 24-hour clock time with minute settings at 0, 15, 30, or 45 (because the simulation time step is 15 minutes).

If you change the date and time, you should consider whether or not you should also change the dates/times on the forms shown in Figure 4.10 and Figure 4.11.

- * **Time Step Size:** Enter a duration (hours and minutes) for the MESORAD time step size. This parameter controls the frequency of output generation and the associated calculations.
- * **Scenario Duration:** Enter the simulation duration in hours and minutes (complete hours and additional minutes).
- * **Number of Sources:** Enter the number of sources (1 through 4).
- * **Release Height:** Enter the height (in meters above the ground) for each source to be used. Release height is used to assign the initial vertical element of puff center location. As puffs move, the puff center is assumed to remain a constant distance above ground. The elevation of the release can significantly change the scenario. A complete discussion of diffusion within puffs and its relationship to release height, height of mixing layer, and changes in terrain elevation may be found in NUREG/CR-3344 beginning on page 7, and in NUREG/CR-4000 Vol. 2, pages 5-12 (references in Section 9.21).

Model Library — MESORAD Model

METEOROLOGICAL/DOSE MODEL CONTROL PARAMETERS				
Run Title	:	<u>MAINE YANKEE</u> <u>10 mile Base Case</u>		
Scenario Start Date	:	<u>01 OCT 86</u>		
Scenario Start Time	:	<u>12:00</u>		
Time Step Size	:	<u>1:00</u>		
Scenario Duration	:	<u>006</u> hour <u>00</u> minutes		
Number of Sources	:	<u>1</u> (1 to 4)		
		Source 1	Source 2	Source 3 Source 4
Release Height (meters)	:	<u>1.</u>	<u>0.</u>	<u>0.</u> <u>0.</u>
Sigma Selection	:	<u>N</u> (N, A, B, or D)		
Length of side of grid	:	<u>75.00</u> (kilometers)		
Puff-track-area factor	:	<u>1.0</u> (0.0 to 1.0)		
Polar Grid?	:	<u>Y</u> (Y or N)		
Press the GOLD <F5> key followed by a 'Z' to complete this form.				

Figure 4.9. Form for Changing Met/Dose Model Control Parameters

- * **Sigma Selection:** Enter a letter to select one of four algorithms for a dispersion coefficient (sigma). Refer to the met/dose model documentation and the online HELP information to find out more about these algorithms. Your selection will not significantly change the speed of the met/dose model execution.
- * **Length of Side of Grid** (floating point value): Enter the length (in kilometers) of a side of the square defined by the corner nodes of MESORAD's 31 x 31 Cartesian grid. If the area defined by the grid is large enough, MESORAD may not use detailed terrain or elevation data in computing windfield data or dose adjustments. If you enter a value over about 150 km, you will see a message that not all data may be used in an area that size, and enter the value again if you really want it that big.
- * **Puff-track-area factor:** Enter a factor (0 to 1) that defines the size of the puff tracking area that surrounds the dose computation region. This factor is multiplied by the length of the side of the Cartesian grid to determine the puff tracking area.
- * **Polar Grid?:** Enter Y to use a polar grid near the source or N to avoid using a polar grid.

4.6.2 Change Source Details

When you select this item from the form selection menu, the screen will display forms that will allow you to enter data for radionuclide release sources. The form shown in Figure 4.10 is the first form for the first release of the first source. For each source you will see one form for each of six possible releases. This indicates that 6 releases for each of 4 sources would yield the maximum number of forms: 24.

When you complete one source form for one release, press <GOLD> <Z> to get one form for the next release until you have completed all forms. If you want to get back to the selection menu without going through all four sources, press <GOLD> <Y>. NOTE: If you are using fewer than four sources, check to make sure the remaining sources are zero, at least the first time you model this case.

The data fields are explained below. Active input fields are marked with an asterisk (*).

- * **Source Name:** Enter any sort of caption as a reminder to you while you are updating this form. The source name is not currently used in the met/dose model, so use it any way that suits you.
- **Source Number:** The source number is a display only, to tell you which of the four sources you are viewing.
- **Release Number:** The release number is a display only, to tell you which of the six releases you are viewing for the current source.
- * **Release Date and Time:** Enter the date and time at which this source will start being released from the grid center.

You must ensure this time is within the times you specified in the Met/Dose Control Parameters form, Figure 4.9. If you enter out-of-range dates and times, the met/dose model will not include this source in the scenario.

- * **Release Duration:** Enter number of hours, then minutes.

The met/dose model is a puff model and releases a single puff each simulated 15-minute time step. Therefore, if you enter three minutes for the release duration, a single 3-minute puff will be released, but the concentration of the nuclides that determine dose will be correct. The model interprets a release duration of zero as a continuous release for the duration of the simulation.

- * **Nuclide Quantities:** Enter the quantity, in curies, of each of the 50 nuclides listed. (You may enter floating point numbers directly. Your entry will be converted to the mantissa/exponent format for display.)

SOURCE SPECIFICATION					
Source Name	:	ACCIDENTAL VENT FOR 3 MIN.			
Source Number	:	—			
Release Number	:	—			
Release: Date		01-OCT-83	Time 12:00	Duration: 01 Hr	00 Min
Nuclide	Quantity	Nuclide	Quantity	Nuclide	Quantity
Kr-88	<u>.1000E+00</u>	Ru-106	<u>.1000E+00</u>	Te-132	<u>.1000E+00</u>
	I-131		<u>.1000E+00</u>	I-132	<u>.1000E+00</u>
I-133	<u>.1000E+00</u>		I-134	I-135	<u>.1000E+00</u>
	I-134		<u>.1000E+00</u>		<u>.1000E+00</u>
Xe-133	<u>.1000E+00</u>		Xe-135	Cs-134	<u>.1000E+00</u>
	Xe-135		<u>.1000E+00</u>		<u>.1000E+00</u>
Cs-137	<u>.1000E+00</u>		Ce-114	U-234	<u>.0000E+00</u>
	Ce-114		<u>.1000E+00</u>		<u>.0000E+00</u>
Zr-95	<u>.0000E+00</u>				

- Press GOLD <F5> followed by a 'Z' to obtain next source.

- Press GOLD <F5> followed by a 'Y' to return to Change Selection Menu.

- Exit this form using <RETURN> to change a source-release file.

Figure 4.10. Form for Changing Met/Dose Model Source Specification

A blank or zero for both mantissa and exponent fields indicates no release for that nuclide. If all nuclides are zero, no release is present for that source. Note these are total curies and will be divided by duration to yield rate of release. The five rows of nuclides in the form will scroll on the screen, five rows at a time, to reveal more nuclides as you come to the end of the fifth row.

4.6.3 Change Weather Data

When you select this item from the form selection menu, the screen will display the form shown in Figure 4.11 for changing weather data input. You can enter weather data for each 15-minute simulation time step. You can enter data at other intervals, such as one hour, and the met/dose model will use previous data or extrapolate for the time steps in the interval.

The met/dose model was designed to accommodate data from up to 30 different weather stations. In fact, the realism of the scenario is questionable when only one weather station is used because weather patterns are hardly ever capable of being described for a 10-, 20-, or 50-mile radius with data from only one geographical point. However, data from a single weather station is better than no data at all, and the met/dose model will work just fine with one station.

Because of the met/dose model design, the form is divided into two sections: one for global conditions applying to the entire area for the 15-minute interval, and another section for entering the data from 1 to 30 weather stations describing wind direction and speed for this 15 minutes. See the met/dose model documentation for further explanations.

- * **Date and Time:** Enter the date and time only if you want to generate new weather data. The convention for adding weather data is that if you change the time, you will generate a new data record with that time and whatever changes (if any) you make to the remaining data on the form. A record, in the context of weather data, is a set of meteorological data describing conditions at a specified time. You can also change the time and then press <GOLD> <S>; the system will then search the current case's weather data file for the specified time and display the data. In both cases, you have not changed the data for the original date/time that was displayed. Note that if the date does not coincide with the release date and time on the source specification form, the release may not take place during the simulation time. For instructions on entering weather dates and times, press the HELP key <F6> to get both field and form HELP information.
- * **Global Parameters:** The valid ranges for the five global parameters are shown on the display in parentheses after the entry fields.
- **Station Name, ID, and Location:** These station identifiers are displays only. The station name is specified on the Station Definition form (Section 4.6.5). If the station name field is blank, the station has no name. To include a weather station's data in a modeling run, the weather station must be defined using the Station Definition form. The X and Y are the grid location coordinates of the weather station in kilometers from grid center.
- **PREV(ious) and NEXT Direction and Speed:** These wind directions and speeds for the previous and next 15-minute data records are displays only. If these displays are blank, the previous and/or next records do not exist.
- * **CURR(ent) Direction and Speed:** Enter current wind direction and wind speed in miles per hour.

To complete the form and return to the form selection menu, press <GOLD> <Y>. If you press <GOLD> <Z> while in the modify weather data form, you will obtain and/or generate the data for the next 15-minute period.

Two special commands are used only on the Change Weather Data form.

Display or create new weather data when you enter a new time: Press <GOLD> <S>.

Entering a new time, followed by <GOLD> <S> refers you to that data set. No change in date is registered.


```

                                WEATHER DATA

                                For  01-OCT-83  12:00

Stability Class      :    B  ( A - G )
Precipitation Code   :    0  ( 0 - 6 )
Mixing Depth         :   200 ( 50 - 5000 m )
Upper Level Direction :   45 ( 0 -360 Degrees )
Upper Level Speed     :   02 ( 0 - 45 meters/sec )

Station ID  (    X    ,    Y    )  --- Direction ---  Speed ---
STA1 0001   0.0000   0.0000   PREV  CURR  NEXT  PREV  CURR  NEXT
              0.0000   0.0000              45              2
              0.0000   0.0000              --              --
              0.0000   0.0000              --              --

Press GOLD <F5> 'Z' for next 15 minute time step, GOLD <F5> 'Y' to
return to main menu. Press GOLD <F5> followed by a 'D' to delete this
record, GOLD <F5> followed by an 'S' to get data for changed time.

```

Figure 4.11. Form for Changing Met/Dose Model Weather Data

Delete the currently displayed weather data record: Press <GOLD> <D>. For instance, if you accidentally enter <RETURN> and generate an unintentional data record, you can then delete it by pressing <GOLD> <D>. When a record is deleted, the screen returns to the first weather data record.

4.6.4 Change Checkpoints

Use the form in Figure 4.12 to change the names and locations of up to 50 checkpoints for dose and concentration level. For hilly country, the checkpoint locations must be coordinated with the cartesian grid (Section 4.6.7) which changes the height of the checkpoint relative to a base plane.

- * Checkpoint: Enter a checkpoint name for reference.
- * X and Y: Enter east/west (X) and north/south (Y) location coordinates in kilometers from grid center. Note that these values may be negative. The signs of X and Y follow Cartesian convention: positive in upper right (northeast) quadrant and negative in the lower left (southwest) quadrant. If you enter X and Y values that are off the grid defined by the grid radius selection on the Control Parameters form (Section 4.6.1), the checkpoint will not be used by the model.

CHECKPOINTS			
<u>Checkpoint</u>	<u>X</u>	<u>Y</u>	
<u>CHKPT01</u>	<u>0.3000E+01</u> (km)	<u>0.3000E+01</u> (km)	
Level Type	Level 1	Level 2	
Concentration	<u>0.1000E+01</u>	<u>0.1000E+01</u>	
Thyroid	<u>0.0000E+00</u>	<u>0.0000E+00</u>	
Whole Body	<u>0.0000E+00</u>	<u>0.0000E+00</u>	

- Press GOLD <F5> followed by a 'Z' to save this checkpoint record and obtain the next one.
- Press GOLD <F5> followed by a "D" to delete this checkpoint record and obtain the next one.
- Press GOLD <F5> followed by a 'Y' to return to the Change Selection Menu.

Figure 4.12. Form for Changing Met/Dose Model Checkpoints

- * Level 1 and Level 2: The data generates results in the log output only. If you enter a "Y" for Whole Body, Thyroid, and/or Concentration, then the debug output will report the times at which the dose concentration quantities exceed Level 1 or Level 2. Even if you do not use these entries, you must move through the entry fields.

Press <GOLD> <Z> to get the next checkpoint. Press <GOLD> <D> to delete this checkpoint. Press <GOLD> <Y> to return to the form selection menu.

4.6.5 Change Station Definitions

Use the form shown in Figure 4.13 to define the weather stations when you first set up a site. You do not need this form to generate scenarios after the weather stations have been defined during site setup. You may define up to 30 stations. The data fields are explained below.

- * Station No.: Enter a number from 1 to 30. Each enumerated station must have associated data.
- * Location: Enter X and Y location values as kilometers from grid center. Note that these values may be negative. The signs of X and Y follow Cartesian convention: positive in upper right (northeast) quadrant and negative in the lower left (southwest) quadrant. If you enter X and Y values that are off the grid defined by the grid radius selected on the

STATION DEFINITIONS				
Station No.	Location		Elev	Name
	X	Y		
0001	0.0000	0.0000	11	STA1

Press GOLD <F5> 'Z' to obtain the next station.
Press GOLD <F5> 'Y' to return to Change Selection Menu.

Figure 4.13. Form for Defining Met/Dose Model Stations

Control Parameters form (Section 4.6.1), the station will not be used by the model.

- * Elev(ation): Enter elevation in meters above sea level.
- * Name: Enter a station name that will appear on reports.

To obtain the next station press <GOLD> <Z>. To return to the Change Selection menu, press <GOLD> <Y>.

4.6.6 Change Terrain

Use the form shown in Figure 4.14 to enter data for the site terrain. The terrain input to the system represents an estimated effect of the terrain on wind direction. The met/dose model uses these data to divert the windfield around and over mountains and other terrain. Entries consist of data for a 16 x 16 grid covering the area that was specified in the Control Parameters form. We recommend that you consult with a meteorologist familiar with the met/dose model's use of these values before you change them. If you change grid sizes (length of grid side) on the Control Parameters form, you must enter different terrain data for each grid size. For more immediate information, press the <HELP> key to get both field and form HELP.

- * Intersect Location: Enter integer coordinate values from 1 to 16 to signify west/east (X) and south/north (Y) intersects on the 16 x 16 grid, where grid cell 1,1 is the southwest corner.
- * Degrees from Horizontal: Enter the physical orientation of the terrain features as degrees counterclockwise from horizontal (east/west = horizontal = 0 degrees = 180 degrees; northeast/southwest = 45 degrees; northwest/southeast = 135 degrees).

TERRAIN				
Intersect Location (1-16)		Degrees From Horizontal (0-180)	Slope Coefficient (0-1)	
X	Y		Down	Up
--	--	---	-----	-----
<u>1</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>2</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>3</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>4</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>5</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>6</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>7</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>8</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>9</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>10</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>
<u>11</u>	<u>1</u>	<u>0.</u>	<u>0.000</u>	<u>0.000</u>

Press GOLD <F5> followed by a 'Y' to return to the Change Selection Menu.

Figure 4.14. Form for Changing Met/Dose Model Terrain

- * **Slope Coefficient:** Enter decimal values from 0 (vertical) to 1 (flat). Down = wind from east; up = wind from north. The slope coefficient is the component of wind blowing perpendicular to a flat, horizontal surface.

To return to the Change Selection menu, press <GOLD> <Y>.

4.6.7 Change Cartesian Grid

Use the form shown in Figure 4.15 to change Cartesian Grid data. The form allows entry of elevations for a 31 x 31 grid covering the area that was specified on the Met/Dose Control Parameters form (Figure 4.9). Elevation correction is necessary in hilly country where the height of the checkpoints above otherwise level terrain requires adjustment to compute a better value of dose.

Enter new data only after consultation with a qualified person who is familiar with the met/dose model.

Data Entry: Enter elevations in meters above sea level. If you change the "Length of side of grid" on the Control Parameters form, you must enter different topographical data for each choice of grid size. The Generate Cartesian Grid Function should be used for this (Section 4.6.2).

Topography															
West														Center	
Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col
Row	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	200	200	200	200	200	200	200	200	200	200	200	200	200	200	200
Center														East	
Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col
16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31
200	210	210	200	200	200	200	220	220	230	230	210	210	200	200	200

- Press GOLD <F5> followed by a 'Z' to obtain next row.
 - Press GOLD <F5> followed by a 'Y' to return to the Change Selection Menu.

Figure 4.15. Form for Changing Cartesian Grid

Column numbers increase from 1 to 31, west to east. Row numbers increase from 1 to 31, south to north.

To obtain the next row, press <GOLD> <Z>. To return to the Change Selection menu, press <GOLD> <Y>.

4.6.8 Change Polar Grid

Use the form shown in Figure 4.16 to change site topography data. The form allows entry of elevations for a 3 x 36 grid covering the area of 3200 meters in radius. Topography correction is necessary in hilly country where the height of the checkpoints above otherwise level terrain requires adjustment to compute a better value of dose.

Data Entry: Enter elevations in meters above sea level.

POLAR GRID TOPOGRAPHY

Radius(1) = 800.000 meters

DEG	ELEV (meters)	DEG	ELEV (meters)	DEG	ELEV (meters)
10	<u>3.000</u>	130	<u>0.000</u>	250	<u>0.000</u>
20	<u>0.000</u>	140	<u>0.000</u>	260	<u>0.000</u>
30	<u>0.000</u>	150	<u>0.000</u>	270 (W)	<u>0.000</u>
40	<u>0.000</u>	160	<u>0.000</u>	280	<u>0.000</u>
50	<u>0.000</u>	170	<u>0.000</u>	290	<u>0.000</u>
60	<u>0.000</u>	180 (S)	<u>0.000</u>	300	<u>0.000</u>
70	<u>0.000</u>	190	<u>0.000</u>	310	<u>0.000</u>
80	<u>0.000</u>	200	<u>0.000</u>	320	<u>0.000</u>
90 (E)	<u>0.000</u>	210	<u>0.000</u>	330	<u>0.000</u>
100	<u>0.000</u>	220	<u>0.000</u>	340	<u>0.000</u>
110	<u>0.000</u>	230	<u>0.000</u>	350	<u>0.000</u>
120	<u>0.000</u>	240	<u>0.000</u>	360 (N)	<u>0.000</u>

Press GOLD <F5> then 'Z' to save your changes
Press GOLD <F5> then 'Y' to get Change Selection Menu

Figure 4.16. Form for Changing Polar Grid

To save changes, press <GOLD> <Z> and to return to the Change Selection menu, press <GOLD> <Y>.

4.6.9 Display Weather Directory

Figure 4.17 lists dates and times of weather observation data. You cannot interact with this form; it is only a listing for informational purposes.

Model Library — MESORAD Model

Directory of DRC1:[D3C088.SITES.JCB1.INPUT.MESORAD]C000OBS.DAT

Rec#	Date	Time	Rec#	Date	Time	Rec#	Date	Time
001	01-JAN-88	12:00	021	31-DEC-89	23:45	041	31-DEC-89	23:45
002	31-DEC-89	23:45	022	31-DEC-89	23:45	042	31-DEC-89	23:45
003	31-DEC-89	23:45	023	31-DEC-89	23:45	043	31-DEC-89	23:45
004	31-DEC-89	23:45	024	31-DEC-89	23:45	044	31-DEC-89	23:45
005	31-DEC-89	23:45	025	31-DEC-89	23:45	045	31-DEC-89	23:45
006	31-DEC-89	23:45	026	31-DEC-89	23:45	046	31-DEC-89	23:45
007	31-DEC-89	23:45	027	31-DEC-89	23:45	047	31-DEC-89	23:45
008	31-DEC-89	23:45	028	31-DEC-89	23:45	048	31-DEC-89	23:45
009	31-DEC-89	23:45	029	31-DEC-89	23:45	049	31-DEC-89	23:45
010	31-DEC-89	23:45	030	31-DEC-89	23:45	050	31-DEC-89	23:45
011	31-DEC-89	23:45	031	31-DEC-89	23:45	051	31-DEC-89	23:45
012	31-DEC-89	23:45	032	31-DEC-89	23:45	052	31-DEC-89	23:45
013	31-DEC-89	23:45	033	31-DEC-89	23:45	053	31-DEC-89	23:45
014	31-DEC-89	23:45	034	31-DEC-89	23:45	054	31-DEC-89	23:45
015	31-DEC-89	23:45	035	31-DEC-89	23:45	055	31-DEC-89	23:45
016	31-DEC-89	23:45	036	31-DEC-89	23:45	056	31-DEC-89	23:45
017	31-DEC-89	23:45	037	31-DEC-89	23:45	057	31-DEC-89	23:45
018	31-DEC-89	23:45	038	31-DEC-89	23:45	058	31-DEC-89	23:45
019	31-DEC-89	23:45	039	31-DEC-89	23:45	059	31-DEC-89	23:45
020	31-DEC-89	23:45	040	31-DEC-89	23:45	060	31-DEC-89	23:45

<RETURN> to view next screen, <-> to view preceding screen, or <Q> to quit.

Figure 4.17. Weather Directory Listing Example

4.6.10 Change Output Options

This option on the Parameter Change menu produces a sequence of two screen forms for limiting the types of output that are produced when the current MESORAD case is executed. Limiting the output reduces the time needed to execute the model and reduces the disk space needed for output reports.

1. MESORAD Report Generation Options. Use the form shown in Figure 4.18 to specify which reports will be generated from the current MESORAD case.
 - a. Enter a "Y" in any of the form fields to specify that the corresponding report(s) will be generated.

Met/Dose Case Debug Output: Entries in this left column will vary the detail written to the "Debug" file, the ones of most interest to programmers or very experienced users are:

- TESTMOV - puff positions and dimensions
- Puff Summary - summary status of puffs from each source
- TSTWF - windfield information
- TIMER - CPU time used by MESORAD execution

Special Tabular Reports: If a "Y" is NOT entered in a right-column field, the corresponding calculations will NOT occur. (For descriptions of these tabular reports, refer to Section 4.8, Print/Display Reports.)

- b. Press <GOLD> <Z> to accept the choices and see the next form.

MESORAD REPORT GENERATION OPTIONS	
Met/Dose Case Debug Output	Special Tabular Reports
- TESTMOV	- Dose Report
- Puff Summary	- Field Team Summary Release Report
- TSTWF	- PAG Dose Summary Report
- TIMER	- Ingestion Pathway Report
Enter "Y" next to each report that you require.	

Figure 4.18. Form for Specifying MESORAD Report Generation Options

2. Binary Output Matrix Specification. Use the form in Figure 4.19 to specify which binary output will be generated from the current MESORAD case. The binary output consists of the calculated values that can be displayed as contours or (in the case of doses) picked at point locations on the site map.

- a. Enter a "Y" in any field to specify that the corresponding values will be generated in binary matrix files.

Calculations for an output item will be made only if a "Y" occurs in one or both of the corresponding fields for Current values (for the current time step) or Cumulative values (since the start).

- b. Press <GOLD> <Z> to accept the choices and return to the Change Parameters menu.

BINARY OUTPUT MATRIX SPECIFICATION

Curr Cumul:

-	-	ALL
-	-	Relative Plume Concentration
-	-	Relative Deposition
-	-	Dose - Total Whole Body
-	-	Dose - Thyroid
-	-	Dose - Inhalation/Whole Body
-	-	Dose - Deposition
-	-	Dose - Overhead Plume
-	-	Dose - Lung
-	-	Iodine
-	-	Gamma
-	-	Gamma + Beta

Enter "Y" in each field for which you wish to generate the corresponding binary matrix files (for each time step).

OR

Enter "Y" in the ALL fields to produce all possible output.

Figure 4.19. Form for Binary Output Matrix Specification

4.6.11 Generate Polar Grid

The polar coordinate grid is a fixed-radius grid that is independent of the model area specified on the Control Parameters form. It needs to be generated only once for a given site unless new, more accurate elevation data become available. A Terrain Analysis Cell (TAC) elevation data set for the current site is the only input required by this function.

4.6.12 Generate Cartesian Grid

The Cartesian coordinate grid is a 31 x 31 grid that covers the area specified in the Control Parameters form. It is necessary to regenerate this elevation grid if the "Length of side of grid" is changed on the Control Parameters form or if new, more accurate elevation data become available. A Terrain Analysis Cell (TAC) elevation data set for the current site is the only input required by this function.

4.7 Display Base Map - Graphic Menu Only

MESORAD calls on the IBS map database and Map Display software for a graphical display of modeling results overlaid on background maps. For example, you can display met/dose model results over the area for which the model was run. You can specify what static features to display: political boundaries, waterways, etc. Then you can customize the model results to be displayed by selecting dose type and other information such as the windfield and the path of the effluent plume(s). This interactive graphics display is a powerful tool for understanding the results of the technical models.

For detailed descriptions of the Map Display software, refer to MPDISPLAY in the *IBS Utilities Guide*.

4.8 Print/Display Reports

MESORAD Text Menu: Select A (Print/Display Reports)

When you pick this function from the MESORAD top-level menu, the screen will display five options for printing or displaying MESORAD output, as shown in Figure 4.20. Figures 4.21 and 4.22 are examples of print and display output.

- * Met/Dose Case Debug Report: This first option is of interest only to the very experienced user or programmer and will tie up the printer for some time printing tables of figures. The report will describe the current input to the met/dose model and output for the model time steps.
- * Dose Report: This option prints the standard dose report, usually 3 to 10 pages.
- * PAG Dose Summary Report: This option prints a report that relates the modeled doses to Protective Action Guides (PAG) established by various government agencies. PAG reports are recommended dose commitment levels that require some sort of protective action. The report indicates the grid sector and time at which a PAG was exceeded during a met/dose run. The report also indicates the distance from the release point, total doses, and the active nuclides from the sources.

- * **Ingestion Report:** This option prints a report that shows the whole body (WB) and thyroid (Thy) dose commitments expected through ingestion of vegetables, meat, and milk. The doses are also related to PAG; the report indicates the distance from the release point at which emergency and preventative PAG were exceeded because of ingestion of radioactive materials.
- * **Field Team Summary Release Report:** This option prints tabular reports that relate field readings of iodine concentrations, gamma dose rates, and gamma + beta dose rates to isopleth numbers for different time periods. Note this report is generated from model output by the graphic display software.

REPORT OPTIONS

- (N) Met/Dose Case Debug Report
 - (N) Dose Report
 - (N) PAG Dose Summary Report
 - (N) Ingestion Report
 - (N) Field Team Summary Release Report
-

Figure 4.20. Menu for Print Reports (and Display Reports) Options

For more information on the PAG dose summary report, ingestion pathway report and the field team summary release report, refer to Section 4.15, MESORAD/FEMA.

To select report options: Type Y in the input field of each report that you want to print or display. Press <GOLD> <Z> to print the selected reports or press <GOLD> <Y> to escape to the MESORAD menu.

The system will then prompt you to choose among four options for printing or displaying the report(s). Select an option by pressing the appropriate key:

<u>Key Press</u>	<u>Function</u>
P	Print file on printer
D	Display file on terminal
X	Exit

MAINE YANKEE MULTIPLUME WITH 10:00 EVACUATION
20-OCT-86 12:15:20

DOSE TO POPULATION

Time of Day at End of 15 minutes Period	CURRENT 15 MINUTE PERIOD		CUMULATIVE	
	THYROID REM	WHOLE BODY REM	THYROID REM	WHOLE BODY REM
20-OCT-86 10:15	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 10:30	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 10:45	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11: 0	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:15	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:30	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:45	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 12: 0	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 12:15	1.30178	0.03212	1.30178	0.03212
20-OCT-86 12:30	39.3930	0.94264	40.6948	0.97477
20-OCT-86 12:45	70.7735	1.66823	111.468	2.64300

Figure 4.21. Example Page of Dose Report Print Output

MAINE YANKEE MULTIPLUME CASE WITH 10:00 EVACUATION
20-OCT-86 12:15:20

ALL TIMES THAT DO NOT APPEAR HAVE VALUES OF ZERO

DOSE AT CHECKPOINT BOOTHBAY

Time of Day at End of 15 minute Period	CURRENT 15 MINUTE TIME PERIOD		
	I-131 micro Ci/cc	PLUME mrem/hr	GROUND mrem/hr
20-OCT-86 13:15	1.169146E-07	.320722	.016055
20-OCT-86 14: 0	2.421295E-05	62.2961	3.25024
20-OCT-86 14:15	2.745193E-04	666.396	36.0906
20-OCT-86 14:30	7.320443E-04	1690.78	94.5235
20-OCT-86 14:45	9.793648E-04	2169.62	124.548
20-OCT-86 15: 0	9.279536E-04	1971.88	116.196
20-OCT-86 15:15	4.346244E-04	882.184	53.4811
20-OCT-86 15:30	1.109445E-04	215.269	13.4149
20-OCT-86 15:45	1.782245E-05	33.0999	2.11789

Figure 4.22. Example Page of Dose Report Display Output

4.9 Perform File Maintenance

MESORAD Text Menu: Select C (Perform File Maintenance)

Selecting this function from the MESORAD top-level menu displays the menu shown in Figure 4.23. The menu options allow you to list the directories of evacuation cases and of met/dose cases for the current site. You also can delete selected input and output files from the system (if you own the files). Exercise care here, especially in deleting input files, because re-creation of files can be time-consuming.

```

                                     PERFORM FILE MAINTENANCE

Selection
Designator      Selection

  A      Display Directory of Cases
  B      Delete Case Output for a Specified Case
  C      Delete Case Input and Output for a Specified Case
  X      Exit to Previous Menu

Enter Selection Designator:  _
```

Figure 4.23. Menu for Case File Directory and Optional Delete

4.10 Execute MESORAD - Interactive

MESORAD Text Menu: Select D (Execute MESORAD - Interactive)

Selection of this function from the MESORAD top-level menu causes the met/dose model to run using data defined for the current site and met/dose case number.

You will first see a checklist and a prompt as shown in Figure 4.24. The check list is displayed for your information only: it is NOT a form that you can change. Read the checklist to be sure the case is set up to run as you wish, and answer the prompt:

- If you answer N (no), the screen returns to the menu without running the model.
- If you answer Y (yes), the model will run. While the model is running, you cannot interact with any other IBS function. The system will return to the menu when the model has completed its run.

The current MESORAD case has the following attributes:

```

Site: _____
MESORAD CASE NO.: _____
IDYNEV CASE NO.: _____
MET. OBSERVATION START TIME  __/__/__ __:__
MET. OBSERVATION END   TIME  __/__/__ __:__
SIMULATION START TIME  __/__/__ __:__
SIMULATION END   TIME  __/__/__ __:__
MIN/MAX LON          _____
MIN/MAX LAT          _____
  
```

Execute MESORAD for this case? (Y/N) [N] ==>

Figure 4.24. Checklist for MESORAD Model Execution

4.11 Execute MESORAD - Batch

MESORAD Text Menu: Select E (Execute MESORAD - Batch)

Selection of this function from the MESORAD top-level menu causes the met/dose model to run using data defined for the current site and met/dose case number. While the model is running in batch mode the MESORAD top-level menu displays and you can exit the system or perform other functions. The batch mode is for large modeling runs and you must not try to interact with the MESORAD model by changing input data, case, or sites while it is running.

You will first see a checklist and a prompt asking whether the case should be executed (Figure 4.24). Check the list to be sure the case is set up properly, and answer the prompt:

- If you answer N (no), you will return to the menu without running the model.
- If you answer Y (yes), you will see that the modeling job has been added to a system batch queue; you will then return to the menu.

4.12 Change Case

MESORAD Text Menu: Select F (Change Case)

Selecting CHANGE CASE from the MESORAD top-level menu will display the MESORAD case selection form (example shown below in Figure 4.25). The "Site

Name" in the form refers to the current site. The "MESORAD Model Case" is the "current case" number. The "Mesorad Model Base" number shown for the model is the case from which the current case was originally created.

Use the up, down, left, and right arrow keys (<F1> - <F4>) to move the input cursor to the field you want to change. When you are finished changing fields, press <GOLD> <Z> to accept the changes and exit the form (or <GOLD> <Y> to exit, restoring the original contents of the fields).

MESORAD MET/DOST MODEL	Case Selection Form V2.00
Site Name: _____	
Mesorad Model Case: _____	
Mesorad Model Base: _____	

Figure 4.25. Example Change Case/Site Form

4.13 If You Have Problems

IBS applications are designed to provide as much help to the user as possible in the form of data range checks, and status and diagnostics messages to make the system's use independent of computer systems specialists. If you have questions, try the following:

1. To obtain on-line help with data forms and fields, press the HELP (<F6>) key on your terminal keypad, as described in Section 1 of the *IBS User Guide*.
2. Contact the System Manager for assistance with problems related to the operation of MESORAD.
3. Refer to your equipment service contract for the name of your vendor representative if problems seem to be related to hardware.

4.14 Status and Diagnostics Messages

The system provides a number of status messages that keep you informed about system activity. These messages generally do not require any reciprocal action on your part. They will be generated automatically under certain circumstances, such

as the start of a process that requires substantial computing time and does not involve user interaction.

The following diagnostic messages will appear under certain circumstances. Where practical, the associated actions are recommended. (This list is currently not complete and is being updated.)

- YOU MUST ENTER 3 DIGITS TO SPECIFY THE MET/DOSE CASE TO DELETE. FOR EXAMPLE "001" TO SPECIFY CASE 1. THIS IS FOR YOUR PROTECTION.
- YOU MUST USE DIGITS ONLY TO SPECIFY THE MET/DOSE CASE TO DELETE. FOR EXAMPLE "001" NOT "C1".
- PLEASE TRY AGAIN.
- The met/dose model has completed processing.
- The met/dose model was initiated at a previous time and is still running.
- The met/dose model has begun execution.
- THE MET/DOSE CASE NUMBER THAT YOU SPECIFIED FOR A BASE CASE DOES NOT EXIST.
- AN UNANTICIPATED ERROR HAS BEEN ENCOUNTERED WITH THE MET/DOSE BASE CASE - PLEASE REPORT TO YOUR SYSTEM MANAGER.

4.15 MESORAD/FEMA

This section documents two special adaptations of MESORAD for use by FEMA: 1) Ingestion pathway doses and 2) Field team reports. This information is for reference completeness and is not intended for the typical user who might use the reports without needing to know the details of the technical adaptations to the model.

DOCUMENTATION NOTE: If you have used IEMIS to print this User's Guide through your text printer, some terms and equations in this section will not show the correct use of superscripts and subscripts.

4.15.1 Ingestion Pathway Doses

For estimating the potential long-term effects of an accidental release of radionuclides, MESORAD/FEMA (the implementation of MESORAD in IBS) calculates doses that would be incurred by the ingestion pathway. The general

method of calculating these doses is to use the ground deposition values calculated by the atmospheric dispersion models in MESORAD (NUREG/CR-4000), convert these values to radionuclide concentrations in the food items, and use these concentrations to calculate dose commitments to a person eating the food products. Three ingestion dose pathways are considered by MESORAD/FEMA: leafy vegetables, milk, and meat (beef). For each of the three pathways, 50-year dose commitments are calculated for the whole body and for the thyroid.

The mathematical models used for these calculations are derived from the recommendations of the National Council on Radiation Protection and Measurements, as published in NCRP-76 (1984). Although the NCRP-76 models were developed for continuous releases over a period of about a year, slight modifications were made to the equations to apply them to short-term releases.

The mathematical techniques used for calculating ingestion pathway doses necessarily contain many assumptions to assure conservatism in as many situations as possible. In an actual accident, the most reasonable method of assessing the hazard to humans due to the ingestion pathway would be to physically monitor the food items to determine the concentrations of radioactive material. For prediction of the effects of a hypothetical accident, however, calculations must be done instead of these measurements. Thus MESORAD/FEMA has been designed to calculate the effects of these long-term dose pathways.

— Concentration of Radionuclides in Leafy Vegetables

For calculation of whole body and thyroid doses due to the ingestion of leafy vegetables, MESORAD/FEMA assumes the accidental release occurs during the growing season. Uptake of radionuclides by the plants is assumed to occur by direct deposition of radionuclides on the plant leaves and by root uptake from radionuclides deposited on the soil. Ingestion doses are calculated for each reference point on the grid, but these doses refer to plants that are grown at the reference point. It is assumed the person potentially receiving the ingestion dose would eat 200 kg/year of vegetables grown at the reference dose point.

For a given reference point on the grid, the concentration of radionuclide i in the edible portion of leafy vegetables is calculated as the sum of the concentrations due to deposition on the leaves and uptake from the soil by plant roots:

$$C_{iv} = C_{iv}^D + C_{iv}^S \quad (1)$$

where:

C_{iv} = Concentration of radionuclide i in the edible portion of leafy vegetables (Ci/kg)

C_{iv}^D = Concentration of radionuclide i in the edible portion of leafy vegetables due to direct deposition on leaves (Ci/kg)

C_{iv}^S = Concentration of radionuclide i in the edible portion of leafy vegetables due to uptake by the roots (Ci/kg).

For an accident situation, the contribution of uptake from the soil through the roots is much smaller than the direct deposition. Both components of C_{iv} were included for completeness.

C_{iv}^D is calculated by:

$$C_{iv}^D = \frac{G_i}{t_s} \frac{f_R T_{iv}}{Y_v} \frac{1 - \exp -\lambda_{Ei} t_s}{\lambda_{Ei}} \quad (2)$$

where:

G_i = Concentration of radionuclide i on the ground at the dose point (Ci/m²)

f_R = intercept fraction (the fraction of deposited material intercepted and retained on the foliage) = 0.25

T_{iv} = translocation factor (fraction of externally deposited radionuclides that are translocated to the edible portion of the plant) = 1.0

Y_v = standing plant biomass (edible portion) at harvest growing above a unit surface area = 2 (kg/m²)

t_s = duration of the accident scenario, from the start of the scenario (days)

λ_{Ei} = effective removal constant for radionuclide i from the plant.

$$\lambda_{Ei} = \lambda_i + \frac{\ln 2}{t_w} \quad (3)$$

where: λ_i = radionuclide decay constant (d⁻¹)

t_w = weathering half-life = 14 (d).

C_{iv}^S is calculated by:

$$C_{iv}^S = \frac{G_i}{t_s} \frac{B_{iv}}{P} \frac{1 - \exp -\lambda_{Bi} t_s}{\lambda_{Bi}} \quad (4)$$

where:

B_{iv} = concentration ratio for plant uptake of radionuclide i from soil (pCi/kg wet mass of plant per pCi/kg dry soil)

P = soil "surface density" (the soil density divided by the depth of the plow layer = 200 kg/m²)

λ_{Bi} = effective removal constant of radionuclide i from soil (d^{-1}).

$$\lambda_{Bi} = \lambda_i + \frac{\ln 2}{t_{sr}} \quad (5)$$

where:

t_{sr} = soil-removal half-life = 70×365 (d)

--- Concentration of Radionuclides in Pastures

The calculation of radionuclide concentrations in milk and meat products both depend on the calculation of concentrations in the pasture plants that the animals eat. The concentration in pasture is calculated as the sum of direct deposition and soil uptake components.

$$C_{ip} = C_{ip}^D + C_{ip}^S \quad (6)$$

where:

C_{ip} = concentration of radionuclide i in pasture forage (Ci/kg)

C_{ip}^D = concentration of radionuclide i in pasture forage due to direct deposition (Ci/kg)

C_{ip}^S = concentration of radionuclide i in pasture forage due to uptake from the soil (Ci/kg).

C_{ip}^D is calculated by:

$$C_{ip}^D = \frac{G_i}{t_s} \frac{f_R T_{iv}}{y_p} \frac{1 - \exp -\lambda_{Ei} t_s}{\lambda_{Ei}} \quad (7)$$

where all variables have the same meaning as in Equation 2, except:

f_R and T_{iv} have the same values as in the calculation of deposition on leafy vegetables, and

y_p = standing pasture plant biomass above a unit area of pasture = 0.12 (kg/m²).

C_{ip}^S is calculated by:

$$C_{ip}^S = \frac{G_i}{t_s} \frac{B_{iv}}{P} \frac{1 - \exp -\lambda_{Bi} t_s}{\lambda_{Bi}} \quad (8)$$

where all variables have the same meaning as in the previous equation, except:

B_{iv} = concentration ratio for pasture plant uptake of radionuclide i from soil (pCi/kg dry mass of plant per pCi/kg dry soil).

— Concentration of Radionuclides in Milk

This calculation assumes that the accidental release occurs during the growing season when the dairy cows are feeding solely on pasture. If the accident occurred during the winter months when the animals were eating only stored feed and were not grazing on pasture, these estimates of milk concentration would be much too high. An estimated concentration of zero would probably not be accurate for winter months, however, since many animals would ingest radionuclides through licking dry brush, grass or even the ground. The concentrations are calculated for dairy cattle that are feeding in a pasture at the reference grid point.

The concentration in milk is calculated from the concentration in pasture plants by

$$C_{im} = F_{im} Q_{Fm} C_{ip} \quad (9)$$

where:

C_{im} = concentration of radionuclide i in milk (Ci/liter)

F_{im} = transfer coefficient of radionuclide i from daily intake of forage by the cow to milk (pCi/L per pCi/day)

Q_{Fm} = quantity of fresh pasture forage consumed per day = 16 (kg/d).

— Concentration of Radionuclides in Meat

This calculation uses the same assumptions as the calculation of the concentrations in milk: that release occurs during the growing season, while beef cattle are eating exclusively pasture plants at the reference dose point.

The concentration in meat is calculated from C_{ip} by:

$$C_{if} = F_{if} Q_{Ff} C_{ip} \quad (10)$$

where:

C_{if} = concentration of radionuclide i in meat (Ci/kg)

F_{if} = transfer coefficient of radionuclide i from daily intake of forage by beef cattle to meat (pCi/kg per pCi/day)

Q_{Ff} = quantity of fresh pasture forage consumed per day = 12 (kg/d).

— Dose Commitment Due to Ingestion of Radionuclides in Food Products

The dose to man due to ingestion by any food pathway is calculated using three terms: the radionuclide concentration in the food product, a usage factor (the quantity of food product consumed by the person at risk), and a dose factor (converting the quantity of ingested radionuclides to a dose commitment).

The only one of the component factors that is dependent on position on the grid is the concentration of radionuclides in the food product. The grid position of concern here is the location of the plants that were directly exposed to depositing radionuclides; the person who eats the contaminated food may live many miles from the contaminated pasture or vegetable patch. Thus the code user must be aware that the summation of doses from different dose pathways is difficult, unless it can be assumed that the person eats food products that are grown in his own backyard.

The usage factors are the quantities of food products consumed by the person on a yearly basis. The factors used by MESORAD/FEMA are:

<u>Food Product</u>	<u>Quantity Consumed per Year (u)</u>
Leafy Vegetables	200 kg/yr
Milk	300 liters/yr
Meat	100 kg/yr

In the MESORAD/FEMA calculations, the usage factors are combined with the concentrations in the food products to give a total quantity of ingested radionuclides. The time period of consumption is assumed to be one year, considering that contaminated food products could be eaten fresh (as vegetables are picked from a garden) during the harvest season, or preserved (canned or frozen) for up to a year after the accident. During this time period, radionuclides with half-lives shorter than a year would experience significant radioactive decay, so the effects of decay are factored into the calculation as follows:

$$UC_{ix} = \dot{u}_x C_{ix} \frac{1 - \exp -\lambda_i t_u}{\lambda_i} \quad (11)$$

where:

UC_{ix} = activity of radionuclide i ingested via pathway x (Ci)

$\dot{u}_x$ = daily intake of food products x (kg/d or L/d)

C_{ix} = concentration of radionuclide i in food product x (Ci/kg or Ci/liter)

λ_i = radioactive decay constant of radionuclide i (d^{-1})

t_u = consumption time period = 365.25 (d).

The total dose commitment to organ o due to consumption of contaminated food product x is found by summing the effects of all radionuclides in the release:

$$D_{ox} = \sum_i DF_{oi} UC_{ix} \quad (12)$$

where:

D_{ox} = 50-year dose commitment to organ o due to consumption of food product x (rem)

DF_{oi} = dose factor for the ingestion of radionuclide i, considering the dose to organ o (rem/Ci).

Dose commitments are calculated for the whole body and thyroid.

4.15.2 MESORAD/FEMA Team Reports

The MESORAD/FEMA code is designed to produce output that simulates the readings that field monitoring teams could expect to see on their instruments during an accidental release. There are three types of field team reports produced by MESORAD/FEMA:

- Air concentrations of radioiodines
- Gamma exposure rates
- Gamma + Beta exposure rates

— Radioiodine Air Concentrations

The concentration of all radioisotopes of iodine in air at a given grid point at a given time step in the simulation is calculated by:

$$R_I = \sum_i C_i n_p 10^{-3} \quad (13)$$

where:

R_I = instantaneous concentration in air of iodine isotopes (mCi/cm³)

C_i = concentration of the ith iodine nuclide in air (Ci-hr/m³)

n_p = number of puffs per hour released on to the grid (hr⁻¹)

10^{-3} = conversion factor (mCi/cm³ per Ci/m³).

— Gamma Exposure Rate

The gamma exposure rate corresponds to the reading that would appear on a field survey instrument that was shielded against weakly-penetrating radiation, such as betas. These instruments usually read out in units of mR/hr. The gamma exposure rate for a given grid point as a given time step is calculated by:

$$\dot{R}_\gamma = D_x n_p 10^3 \quad (14)$$

where:

- $\dot{R}_\gamma$ = gamma exposure rate; (mR/hr)
- D_x = external whole-body dose (rem)
- 10^3 = conversion factor (mR per rem).

An important approximation is made in using Equation 15, the assumption that the external whole-body dose rate is numerically equivalent to the gamma exposure rate. Exposure (measured in Roentgens) is nearly equal; numerically to dose (measured in rad), and the quality factor used to convert dose to dose equivalent (measured in rem) is 1.0 for gammas, so this approximation introduces an uncertainty of less than 10% into the calculation. The other assumption that must be made is that whole-body dose is equal to the free-in-air dose that would be measured by an instrument; this assumption ignores the attenuation of radiation by surface tissues of the body. Thus, a total uncertainty of perhaps 25% is introduced by this approximation.

— Gamma + Beta Exposure Rates

The gamma + beta exposure rate corresponds to the reading of an unshielded survey instrument that would respond to both gamma and weakly-penetrating beta radiation. (The term "exposure" is defined only for gamma radiation, so it is a misuse of the term to apply it to the measurement of beta radiation. However, survey meters have readouts in units of mR/hr, and they do respond to beta particles, so in common usage a survey team will measure both gamma and beta radiation with the meters.)

The gamma exposure rate, as calculated by Equation 15, is added to a beta response as follows:

$$\dot{R}_{\beta\gamma} = \dot{R}_\gamma + n_p \sum_i C_i DF_{\beta i} \quad (15)$$

where:

- $\dot{R}_{\beta\gamma}$ = gamma + beta exposure rate (mR/hr)

C_i = time-integrated concentration in air of radionuclide i (Ci-hr/m³)

$DF_{\beta i}$ = beta dose factor for radionuclide i (mR/hr per Ci/m³).

This equation uses a semi-infinite cloud approximation for the calculation of beta doses. This approximation is a good one for betas, because air concentrations can be assumed to be uniform over the range of betas in air.

Section 5

Generate Evacuation Network

IEMIS Menu: Select **MODELS**
Models Menu: Select **BUILDNET**

This section describes Generate Evacuation Network, a capability for automatically generating initial evacuation cases and a traffic network for the evacuation model, IDYNEV (Interactive DYnamic Network EVacuation). This section covers the following topics.

- 5.1 Description of Generating an Evacuation Network
- 5.2 What You Need to Know About Generating an Evacuation Network
- 5.3 Generating a First-Cut Evacuation Traffic Network and Cases
- 5.4 Details of Automatic Evacuation Case Generation

5.1 Description of Generating an Evacuation Network

Generate Evacuation Network is a capability for assembling a set of data files that define a traffic network for the IDYNEV (Interactive DYnamic Network EVacuation) model. When applied to a site map database, the network generation feature uses roads and population information from the site database to take a first cut at "creating" (assembling the data to define) a traffic network to be used by the evacuation model. This automatic feature also generates first-cut IDYNEV traffic assignment and simulation cases for the site.

Although the resulting network is unlikely to be an accurate representation of the real roadways, you can use it as a starting point and refine the network to be as accurate and detailed as you wish.

5.2 What You Need to Know About Generating an Evacuation Network

The IDYNEV model predicts the progress of area evacuations by indicating the changing conditions of population movement over a traffic network. This section includes most of what you need to know about generating a first-cut evacuation

network and initial input data sets (cases) for IDYNEV. Section 6 on the IDYNEV model provides insight about what the network and these cases really mean. Once you have generated a first-cut evacuation network, you will need to refer to Section 6 for IDYNEV references and information on revising the network to meet your needs. **Section 6 includes an overview of IDYNEV and vocabulary that may be helpful.**

Generating an evacuation network and case requires a graphics terminal and interactions with IBS map display graphics. You should be familiar with using the graphics input device (mouse or joydisk) and the map display menus, as described in the IBS User Guide. You should also be familiar with filling out data forms as described in the IBS User Guide.

The procedures that you must follow to generate a traffic network and IDYNEV cases are described in Section 5.3. The processes underlying the procedures are described in Section 5.4. Understanding of Section 5.4 is NOT necessary for generating a first-cut traffic network and evacuation cases. However, before you can later refine the traffic network and evacuation cases to reflect a real situation, you must read Section 5.4 to understand the underlying assumptions.

Limitations of Automatic Evacuation Case Generation: It is important to remember that traffic networks and evacuation cases generated automatically are first-cut approximations only. First-cut approximations provide as much information as possible, based on available database information. However, some information required by the evacuation model is not yet available in the map database. In these cases, the system uses default values or guesses. Such information (including turn movements to downstream nodes, link records, link directions, and signal timing information) should be updated by modifying the evacuation model input data (Section 6.4). Further, never take the system's accuracy for granted. Each traffic network that is generated by the first-cut approximation process should be reviewed to apply the most current information available.

5.3 Generating a First-Cut Evacuation Traffic Network and Cases

Selecting "Generate Evacuation Network" on the Model Library menu allows you to create a first-cut evacuation traffic network and IDYNEV cases for a specified site. Table 5.1 summarizes the steps of the network and case generation. Notice that this process requires that site files contain "ROADS" and "POPULATION" data (OR other data sets with similar attributes). See Section 5.3.7, "Qualifying Roadway Selection," for further information on relevant attributes for Roads data. See the *IBS Data Management Guide* for information on population attributes.

The evacuation cases created by the process shown in Table 5.1 do NOT necessarily represent accurate or meaningful input to the evacuation model.

Rather this feature is a quick way to create case data and a traffic network that can later be modified with more detailed input.

Table 5.1 presents the sequence necessary for first-time generation of a network and case data; however, the network and case generation options used in Steps 5 through 10 may also be used to modify existing network and case data. For example, you may change host location data for an existing evacuation boundary and then regenerate the cases.

Table 5.1. Overview of Generating a First-Cut Network and Cases

What You Do	What IBS Does
1. Select BUILDNET on the Models menu (Graphic menu only).	Presents the First Cut Case/Site Specification form.
2. Specify on the form: - traffic assignment case # - traffic simulation case # - population set name - flood (SLOSH) output overlay.	Establishes % of potential evacuees who will evacuate. Asks you whether you wish to use default network parameter settings. (These parameters include the road types used in the network, free-flow speeds, etc. The network parameter settings are stored in a case-numbered file named <i>CnnnEVACPARAM.DAT</i> .)
3. Specify % population that will evacuate.	Establishes % of potential evacuees that will evacuate.
4. Load the default parameter settings (or use an existing network parameter file).	Presents the Map Display menu for displaying map features and information (MPDISPLAY, in the <i>IBS Utilities Guide</i>).
5. Select the topographies to be displayed, including POPULATION and ROADS (or others of similar content).	Retrieves and displays the topography data from the site files. Displays the graphic menu for network and case generation.
6. DEFINE EVACUATION BOUNDARY by graphically drawing a line around the evacuation area.	Creates a DMS file (<i>CnnnEVAC.DMS</i>) to hold the graphic boundary information for network generation.
7. DEFINE HOST LOCATIONS (evacuation centers outside the boundary) that are capable of sheltering evacuees.	Adds host location information to the DMS <i>CnnnEVAC.DMS</i> graphics file.
8. QUALIFY ROADWAY SELECTION information (optional).	Updates the network parameter file to contain your qualifications.
9. GENERATE NETWORK based on site road data, roadway selection qualifications, evacuation boundary, and host locations.	Associates site population points with internal link locations (called <i>centroids</i>) where vehicles enter the network. Associates each host location with exit nodes on the evacuation boundary (estimates turns that vehicles could take to exit the evacuation area).
10. Pick GENERATE EVAC CASES on the menu, specify the number of people per vehicle, time interval, load time, and load curve.	Converts population values to a number of vehicles. Establishes the number of vehicle trips per hour originating at each centroid. Generates the input data for two initial cases: one traffic assignment case and one traffic simulation case.
11. Pick ENABLE ROADWAY NETWORK.	Graphically displays the roadway network.

Generate Evacuation Network

The steps of generating an evacuation traffic network and initial cases are described in the following subsections:

Step(s)	Section	Topic
1 and 2	5.3.1	Specifying Case Numbers, Population Set, and Flooding
3	5.3.1	Specifying % of Population that Will Evacuate
4	5.3.2	Specifying a Network Parameter File
5	5.3.3	Selecting Topographies to be Used During Network Generation
5-11	5.3.4	Using the Network/Case Generation Menu
6	5.3.5	Defining an Evacuation Boundary
7	5.3.6	Defining Host Locations
8	5.3.7	Qualifying Roadway Selection
9	5.3.8	Generating the Network
10	5.3.9	Generating Initial Evacuation Cases
11	5.3.10	Displaying (Enabling) the Roadway Network

5.3.1 Specifying Case Numbers, Population Set, and Flooding

The system presents the form in Figure 5.1 for specifying the conditions for generating the evacuation case(s) and traffic network. The fields of the form are described in the following paragraphs. An asterisk (*) appears next to each input item.

Site Name: Your current site name - reference only.

- * **First Cut TA Case:** Enter a case number for an IDYNEV traffic assignment (TA) case to be created. By convention, this should be an odd-numbered case.
- * **First Cut Sim Case:** Enter a case number for an IDYNEV simulation case to be created. By convention, this number should be one larger than the traffic assignment case number.

First Cut Case/Site Specification Form

Site Name: _____
First Cut TA Case: 1
First Cut Sim Case: 2
Based on Population Set Code: BASI
Flood Output Overlay (optional): _____

Figure 5.1 Menu for Generate Evacuation Network

- * **Based On Population Set Name:** Enter the four-character code of the site population set to be used for case generation.

The name of the default site population set is generally "Basic Population" and its four-character code is "BASI" -- which corresponds to the population set extracted from the site database. Note the population set is more than the DMS population data from which it was taken. It also includes pointers that relate the population to the evacuation network centroids.

- * **Flood Output Overlay** (optional): Enter the name of the flood (SLOSH output) data file.

Example: DISK1:[yourdirectory]FLOOD.DAT

This item gives you the option of using data from the SLOSH flood model to indicate areas where flood waters would prevent roadway links of the network. When you enter a flood data file name, Regional Evacuation will later use SLOSH output from that file to eliminate any roadway links that appear to be covered with water after flooding.

Caution

No method is currently available for adjusting SLOSH output to the actual landscape (see IBS Implementation of SLOSH Data, below).

SLOSH Background. The National Oceanic and Atmospheric Administration (NOAA) has a computer model (SLOSH) for predicting the flood water levels that would result from a specifically defined storm at a specific coastal location. The model input includes the site location and a storm case defined by the National Weather Service. The output of the model is in terms of maximum surge height above mean sea level (National Geographic Vertical Data) for longitude-latitude points on a grid. These data are called a Maximum Envelope of Waters (MEOW). Each SLOSH output data file is the result of a specific site and storm condition, so no time variables are associated with the output data.

IBS Implementation of SLOSH Data. If you have obtained a SLOSH data file for your site from the NOAA, you may use it in determining the evacuation roadway network. That is, when you are building an evacuation network or rebuilding the network after changing demographic data, you can specify that the SLOSH flood data be used to eliminate any roadway link that appears to be covered with one or more feet of water after flooding.

During network building, the system compares the MEOW data from the SLOSH output file to the DMS ROADS topography file. Flooded areas, indicated graphically by contours overlaid on the site map, are eliminated from the roadway network. Because network links (drawn in white) are expressed as vectors resulting from the

road data, the links may appear to cross flooded areas, although the underlying roads do not. It is possible that flooded areas would prohibit creation of a roadway network capable of evacuating the population from the desired area.

This use of SLOSH output data is an interim implementation to demonstrate the capability for controlling roadway network building based on SLOSH output. Because the system does not include site elevation data in these calculations, extreme care should be taken in interpreting the results, which are in terms of surge heights above mean sea level. Implementation of the capability with elevation data is being considered.

As a practical matter, one would probably wish to generate cases with and without SLOSH data and then consider which, if any, of the potentially flooded links should be eliminated or blocked from use. Such analysis requires in-depth knowledge of the area being studied.

When you are done modifying input items, press <GOLD> <Z> (or <RETURN> in the last field). The system then clears the screen and asks you what percent of the population will evacuate and what network parameters to use.

Percent of population evacuated. The system gives you a chance to limit the population that will try to evacuate:

```
*** MSG: Of the total population in the evacuation area, what
          percentage will evacuate?
*** MSG: Enter value in range of 0. to 100. (USE DECIMAL POINT.)
          (DEFAULT IS 100.)
```

-> Enter a percentage value. For example, to indicate a 90% evacuation, type 90. and press <RETURN>. (The percentage of the population evacuating is NOT used as direct input to the IDYNEV traffic model.)

5.3.2 Specifying a Network Parameter File

After exiting the case/site specification form and specifying the percent of population to evacuate, you will see the following prompt:

```
Do you wish to load CASE network parameter settings? (Y/N) [Y]-->
```

Your response will decide what assumptions are made about the roads in the network:

- NO -- Load default network parameter settings. "N" is normally the correct response when you are creating a completely new network. The system then copies a standard set of network parameters to create a file named *Cnnn*EVACPARAM.DAT file, where *nnn* is the simulation case number. These parameters are assumptions about the network roads:
 - what types of roads are included in the network
 - the free-flow speed assumed for each type of road

- the number of lanes for each type of road
- whether the type of road is assumed to be two-way or one-way.
- **YES – Do NOT load default network parameter settings.** "Y" is appropriate only when you are modifying an existing case that already has a *CnnnEVACPARAM.DAT*. This also assumes that you have modified *CnnnEVACPARAM.DAT* to describe your site network more accurately than the system default parameter settings (see **Qualifying Roadway Selection**, Section 5.3.7).

If no current *CnnnEVACPARAM.DAT* exists, the system copies the system default parameter settings to create the new file, even if you answer "Y."

Later you will have an opportunity to change these network parameters before generating the network and case data (Sections 5.3.8 and 5.3.9). Next you will select topographies to be used and displayed during network and case generation.

5.3.3 **Selecting Topographies to Be Used During Network Generation**

After you respond about the network parameter file, the system presents a menu of map display functions as described in MPDISPLAY in the *IBS Utilities Guide*. You must choose features (topographies) that will be included on the site map display while you generate the network and evacuation cases.

1. Use your terminal's graphics input device to pick the "SELECT TOPOGRAPHIES" option on the menu.

You will then see a submenu that lists the available topographies. You may also see an evacuation boundary if one was previously created.

2. Pick the map features that you want to display while defining the evacuation boundary.

If an extract has or will be preformed from Tiger Trails data to be used as the basis of odd data is not necessary to select a roadway topography, but you may still wish to do so for visual reference. Otherwise, be sure to pick ROADS and POPULATION or other topographies that contain that data. The network and case generation software must have these data accessible. Other topographies may be picked for the display, but they are not necessary for network and case generation.

3. After selecting the desired topographies, pick "EXIT."

You will return to the main menu of map display functions.

4. Pick "EXIT" on the map display menu.

The chosen topographies will appear on the screen along with a new graphics menu for generating the roadway network and evacuation cases.

5.3.4 Using the Network/Case Generation Menu

After you exit the map display menu, the system presents the graphic menu for generating the network and the initial evacuation cases (Figure 5.2). How you use this menu depends on whether you are generating a network and cases for the first time or modifying data to create different initial cases. In either situation, your goal is to create a basic set of evacuation model input data that you can refine later.

Generating a Network and Cases for the First Time. If you are generating an original network and case data set for a site, then (in general) you must use the principal options of the menu in a specific order. Sections 5.3.5 through 5.3.10 guide you through the details of using these menu options and the required procedure.

Modifying Data to Create Different Initial Cases. After you have generated a network and initial cases and examined the results, you may want to refine your choices and recreate the network and initial cases. You might want to QUALIFY ROADWAY SELECTION in more detail and then regenerate the first-cut network. Or after you are satisfied with the evacuation boundary, you may want to experiment with different host location assignments in producing other cases.

```
EXIT
QUIT
HELP
CANCEL
CONTINUE

DISPLAY BASE MAP
DEFINE EVAC BOUNDARY
TTXTTDAT
QUALIFY ROAD SELECTION
DEFINE HOST LOCATIONS
GENERATE NETWORK
GENERATE EVAC CASES
ENABLE ROADWAY NETWORK
DISABLE ROADWAY NETWORK

ZOOM IN
ZOOM OUT
MAP ZOOM
RESTORE BASE MAP
REFRESH SCREEN
```

Figure 5.2. Graphics Menu for Generating the Network and Evacuation Cases

The principal options of the graphic menu are summarized here:

- **DISPLAY BASE MAP:** Presents the Map Display menu for displaying map features and related information (MPDISPLAY in the *IBS Utilities Guide*).
- **DEFINE EVACUATION BOUNDARY:** Allows you to draw a boundary line that defines the evacuation area. (See Section 5.3.5.)
- **EXTRACT LINKS FROM TIGER:** If you want to use extracted Tiger Trails (TT) links and you did not select them when you last defined the evacuation boundary, select this option now. This selection will create a direct access link file to be used as input by GENERATE NETWORK.
- **QUALIFY ROAD SELECTION:** Allows you to choose what types of roads are included in the network and to specify some characteristics of those roads. (See Section 5.3.7.) This option must be used before GENERATE NETWORK.
- **DEFINE HOST LOCATIONS:** Allows you to identify locations (such as relief centers) where various numbers of evacuees could be accommodated. (See Section 5.3.6.)
- **GENERATE NETWORK:** If you have not already defined an evacuation boundary and host locations, this option automatically takes you through a sequence to do so as follows. After these are defined, BUILDNET generates network information in preparation for case generation. (See Section 5.3.8.)
- **GENERATE EVAC CASES:** Generates input data for the IDYNEV evacuation model, based on the population set, roadway network, evacuation boundary, and host locations that you have defined. (See Section 5.3.9.)
- **ENABLE ROADWAY NETWORK:** Generates a network data file and graphically displays the network (use this option only AFTER generating the evacuation cases). The graphics display represents the links (streets) and nodes (intersections or points where road characteristics change) of the evacuation roadway network. The network is based on the site ROADS information (or other similar topographic information), the evacuation boundary, and any special qualifications that you have made with QUALIFY ROAD SELECTION.
- **DISABLE ROADWAY NETWORK:** Disables the graphic display of the network. This option does NOT immediately erase the network, but the network will NOT be drawn in any subsequent screen redrawing (REFRESH SCREEN, ZOOM IN, etc.).

The remaining menu items are standard options from Map Display's SCREEN VIEW FUNCTIONS menu (MPDISPLAY in the *IBS Utilities Guide*). The rest of Section 5.3 explains how to use the network/case generation options in the necessary sequence.

5.3.5 Defining an Evacuation Boundary

Generate Evacuation Network

After the site map is displayed, the first step in building a first-cut evacuation case is to define an evacuation boundary: Select **DEFINE EVAC BOUNDARY** on the network/case generation menu.

You will then see the following graphics submenu for drawing a graphic boundary line:

EXIT QUIT HELP CONTINUE DEFINE EVAC BOUNDARY DELETE EVAC BOUNDARY COORDINATE PICK ZOOM IN ZOOM OUT MAP ZOOM RESTORE BASE PLOT REFRESH SCREEN

Figure 5.3. Graphics Menu for Defining an Evacuation Boundary
The two principal options of the menu are explained here:

- **DEFINE EVAC BOUNDARY**: Allows you to define an evacuation boundary by the following procedure.

1. Pick "DEFINE EVAC BOUNDARY" on the graphics menu.

The menu item will start blinking, indicating the system is waiting for you to use the graphics input device to pick points that define the evacuation boundary on the map display.

2. Pick a point that is on the desired evacuation boundary.
3. Pick another point on the evacuation boundary.

The system will draw a boundary line between this point and the previous point.

4. Repeat Step 3 until you have defined all but the last side of a polygon that will enclose the evacuation area.
5. Pick "CONTINUE."

The system will draw the final side of the evacuation boundary, redraw the boundary as a dashed line, and the **DEFINE EVAC BOUNDARY** menu item will stop flashing.

6. Pick "EXIT" to accept the displayed evacuation boundary.

The evacuation boundary information is stored in a graphics file named *CnnnEVAC.DMS*.

If you do not wish to exit, you can pick "DELETE EVAC BOUNDARY" to erase the displayed boundary and start again.

- **DELETE EVAC BOUNDARY:** Deletes the current evacuation boundary and erases it from the screen. (You may need to use REFRESH SCREEN to clean up the display.)

5.3.6 Defining Host Locations

After you have defined the evacuation boundary, the next step in building a first-cut evacuation case is to define **host locations**—locations that could support evacuees from the evacuation area: Select DEFINE HOST LOCATIONS on the network/case generation menu.

You will then see a graphics submenu (Figure 5.4) for defining host locations on the site map. Host locations might represent homes, schools, or any facility designated to support evacuees or any area which will attract evacuees.

The function of defining host location is NOT to create a detailed database of emergency response facilities. Rather it is to aid the creation of first-cut evacuation cases by providing general destinations for use in building the traffic network and modeling the evacuation.

To satisfy the requirements of the evacuation model, the defined host locations must have a combined capacity sufficient to support the population from the evacuation area.

The three principal options of the menu are explained here:

- **DEFINE HOST LOCATION:** Allows you to define host locations by the following procedure.
 1. Pick "DEFINE HOST LOCATION" on the graphics menu.

The menu item will flash, indicating that the system is waiting for you to use the graphics input device to pick points that define the host locations.

Figure 5.4. Graphics Menu for Defining Host Locations

2. Pick a point on the map display to define as a host location.

A flashing symbol that represents the host location will appear on the display. The dialogue screen will display the following message:

```
TOTAL HOST CAPACITY CURRENTLY EQUALS          0
TOTAL POPULATION WHICH MUST EVACUATE EQUALS (population of
area)
ENTER CAPACITY OF HOST      1
```

3. Type a number to indicate how many people can be supported at that host location. Press <RETURN>. (The maximum host capacity is 32,767, but more than one host may be defined in the same place.)

The dialogue screen will summarize your entry. You may need to press the <DIALOG> key to see this summary.

4. Repeat Steps 2 and 3 until you have defined all desired host locations and entered their hosting capacities.

The system will display a running total of the host capacity. You must continue to define host locations until the total host capacity equals or exceeds the total population that must evacuate. (See the note after Step 6.)

5. Pick "CONTINUE."

The system will summarize the host capacity and the total population that must evacuate. For example:

TOTAL HOST CAPACITY CURRENTLY EQUALS	324808
TOTAL POPULATION WHICH MUST EVACUATE EQUALS	324808

6. Pick "EXIT."

This accepts the host location and capacity data for use in building the network and case data. (The system modifies the graphics file *CnnnEVAC.DMS*, storing the host data with the evacuation boundary data.)

- **DELETE ONE HOST:** Allows you to delete a single host location by the following procedure.

1. Pick "DELETE ONE HOST" on the graphics menu.

The menu item will blink, indicating that the system is waiting for you to pick a point to delete.

2. Pick a point on or near the host that you wish to delete.

The nearest host location will blink. If that is not the correct host location, pick another point.

3. Pick "CONTINUE" to confirm that the last-picked host location is to be deleted.

The system will delete the host location (creating a new version of *CnnnEVAC.DMS*), and the menu item will stop blinking.

- **DELETE ALL HOSTS:** Immediately deletes all currently identified host location data.

Note: There are no QUIT or CANCEL options on this menu. If you want to "undo" a change, you must either recreate your previous actions or use the unmodified version of the data file (*CnnnEVAC.DMS*). This second method involves exiting the system to the prompt and deleting the modified version of that file.

Host Location Considerations: It is important to be reasonably accurate in sizing the host capacities and in locating hosts near the actual exit links on which the evacuees will leave. The capacities of the various exit links in the evacuation network are based on

the volume capacities of associated host locations^(a). Therefore, if an exit node (located at the outer end of a link that crosses the evacuation boundary) has no host areas nearer to it than to any other exit node, then it will have no capacity: traffic will be unable to exit through that node. This situation could cause a fatal error during evacuation model execution. If such a situation occurs, the system automatically avoids that error by resetting the exit node's host capacity to 1 when you GENERATE EVAC CASES.

5.3.7 Qualifying Roadway Selection

The menu item QUALIFY ROADWAY SELECTION on the network/case generation menu allows you to specify details about the roadways that are included in the network. This is an optional step in generating a network; however, if you choose to qualify roadway selection, you must do it BEFORE you generate the network. This option requires some detailed knowledge about ROADS data and about what kinds of roads that you want to include in the roadway network.

When you select QUALIFY ROADWAY SELECTION, you will see a data input form similar to the one shown in Figure 5.5. Completing this form will modify a special network parameter file named *Cnnn*EVACPARAM.DAT, where *nnn* is the case number. This file supplies the network generation software with necessary information about the roadways that are to be included in the network for case *nnn*.

For initial case generation, the network parameter file always contains system default values (see Section 5.3.2). If you modify these values here, you are creating a case-specific set of parameters. If you later regenerate the network and case data for the same case number, you can re-use the modified parameters (by answering "YES" when the system asks if you wish to load CASE network parameter settings—described in Section 5.3.2). You may also use the modified parameters for a different case by copying the network parameter file (outside of the system) with a different *nnn* case number.

(a) Applies to TRAD data preparation. If TRAF data preparation is selected during GENERATE EVAC CASES the exit utilization is based on the ideal travel times from internal centroids to prospective exits.

EVACUATION NETWORK PARAMETERS

MINOR ATTRIBUTE	INCLUDE	FREE FLOW SPEED	LANE COUNT	ONE-WAY
10	Y	40	2	2
20	Y	40	2	2
30	Y	40	2	2
40	Y	25	2	2
50	Y	40	2	2
51	N	40	-1	2
52	Y	60	4	2
53	N	40	-1	2
54	Y	60	2	2
55	N	40	-1	2
56	Y	60	2	2
57	N	40	-1	2
58	Y	45	2	2
59	N	40	-1	2
60	Y	55	2	2

Figure 5.5. Scrolling Form for QUALIFY ROADWAY SELECTION

The fields of the form are explained here. An asterisk (*) appears next to each input item. If you elected to load default network parameters settings (Section 5.3.2), the values that appear in the form will be the system default values (as illustrated in Figure 5.5).

- * **MINOR ATTRIBUTE:** This column of minor attribute numbers identifies different features of data within the major "ROADS" data group (Major attribute = 5). For example, minor attribute 50 corresponds to a dead end road, which could be one link in the network. The minor attributes and the data items to which they correspond are listed in IEMIS's attribute dictionary of DMS mapping data (refer to the *IBS Data Management Guide*). The data form is a multi-page form that scrolls up to show several sequential screens of minor attribute numbers (from 10 to 4095).
- * **INCLUDE:** Enter Y to include (or N to exclude) this type of ROADS data in the network.
- * **FREE FLOW SPEED:** Enter a value for the free-flow speed (miles per hour) of vehicles on links that have the corresponding minor attribute.
- * **LANE COUNT:** Enter the total number of traffic lanes (including both directions) for the link. This number will be ignored if an attribute of the link specifies the lane count explicitly. (Enter -1 to indicate situations where no lanes are available and the link should not be included in the network. This has the same effect as an N in the INCLUDE column.)

Generate Evacuation Network

- * **ONE-WAY:** Enter 1 or 2. A 1 specifies a one-way condition where only one link is necessary. A 2 specifies a two-way condition where two links in opposite directions are necessary.

Pressing the <DOWN ARROW> in the final row of the screen will bring up the next screen and sequence of minor attribute numbers.

When you finish changing values, press <GOLD> <Z> to accept the changes. The system then creates a new network parameter file (CnnnEVACPARAM.DAT) for use by the network generation software.

5.3.8 Generating the Network

Select GENERATE NETWORK on the network/case generation menu. The menu item changes color, and the system checks that the following conditions have been met:

- **POPULATION and ROADS topography data are available.** If you try to generate a network without first selecting the ROADS and POPULATION topographies, the system will ask you whether you want to continue. Generally you will need to answer "N" and then use the BASE MAP DISPLAY option to go back and select these topographies. If you have selected other topographies that have appropriate roads and population data sets in the site maps directory or if you are using a TT.DAT file (an extract of links from Tiger Trail data), you may answer Y and the system will allow you to continue. In any case, the system will prompt:

Do you wish to load ROADS from a TT.DAT file? (Y/N) ==>

- **An evacuation boundary is defined.** If you have not defined an evacuation boundary for the current case, the system will display the submenu for defining an evacuation boundary (Section 5.3.5).
- **Host locations of adequate capacity have been defined.** If you have not defined host locations for the current case (or if the capacity of the host locations is smaller than the total population of the evacuation area), the system will display the submenu for defining host locations (Section 5.3.6).

When these conditions have been met, the system begins by building a crude traffic network based on the site roadway data. The network consists of links (roadways) and nodes (intersections). Assumptions about these links and nodes are taken from the network parameter file, CnnnEVACPARAM.DAT. The POPULATION data consists of population values associated with population points. The system assigns these population points and their populations to various centroids (vehicle entry points) on links of the network. The system then displays a message that summarizes the population point/centroid assignments. For example:


```

Total centroids                = 223
Population assigned to         = 223 centroids
Total population assigned      = 21140
Total population points to evacuate = 398
Total population points evaluated = 5049
Turns estimated

```

```

*** MSG: Host/exit associations completed.
*** MSG: Network generation complete.

```

Press <RETURN> to continue

5.3.9 Generating Initial Evacuation Cases

After you have used GENERATE NETWORK, select GENERATE EVAC CASES to complete the case generation for the two case numbers entered on the Case/Site Specification form (Section 5.3.1).

You will then see a sequence of messages and prompts for information. Arrows (→) mark places in the sequence where you must enter information.

- Number of people per vehicle.

On average, how many people will occupy each auto?
Enter value in range of 1.0 to 10.0 (DEFAULT IS 1.3).

- Enter a specific value, or press <RETURN> to use 1.3 people per auto. The First Cut Approximation uses these entries to calculate the traffic loading (number of vehicle trips started per hour) as input to the model. (The number of persons per vehicle will be used directly as load factor data [data form type 52, Section 6.4.17].)

Next, the system will display a menu from which you are to select a time interval length (used in governing how finely IDYNEV will model the data provided). See Section 6.4.9 for further information.

```

Time Interval Selection (Seconds)
-----
0 : EXIT
1 : 60
2 : 90
3 : 120
4 : 150
5 : 180
6 : 210
7 : 240
8 : 270
9 : 300
10 : 360
11 : 420
12 : 480
13 : 540
14 : 600
-----
Menu Choice (?=Help) ==>

```

Generate Evacuation Network

You will then be prompted for total load time:

*** MSG: Enter total time to load network (minutes) ==>

A form will be presented for you to specify the lengths and percent of total load to be applied to each of the first 10 Time Periods (TP) as shown in the following example.

SPECIFY LOAD CURVE

Total Load Time Required: 60(Minutes)

Time Period Sum: 300(Minutes)

Time Interval Specified:120(Seconds)

Time Period	Tenths of Minutes	% of Total Load
TP 1	300	50
TP 2	300	50
TP 3	300	0
TP 4	300	0
TP 5	300	0
TP 6	300	0
TP 7	300	0
TP 8	300	0
TP 9	300	0
TP 10	300	0

As each entry is made in the Tenths of Minutes column, the Time Period Sum field, near the top of the form is updated. The form requires that the Time Period Sum be at least as large as Total Load Time Required, and that the sum of % of Total Load equals 100.

After you complete the form, the following prompt will appear:

*** MSG: Two options are available:

*** MSG: 1. Generate trip table for TRAF

*** MSG: 2. Generate demand and capacity for TRAD

*** MSG: Enter 1 for TRAF input 2 for TRAD (2) ==>

Select one of the two options.

You will then be prompted

Enter name for pupulation set to create/replace: ==>

Enter the same four-character code that you used on the form, or enter a different code if you wish. If the code has already been used, you will see a message similar to the following:

*** MSG: File already exists SITE\$POP:BASIPOP.DAT
Replacing.
Press <RETURN> to continue.

- Host capacity adjustments. If the network includes an exit node that is NOT associated with a host location, you will see the following message:

WARNING: EXIT *nnnn* HAD HOST CAPACITY OF 0.
HOST CAPACITY HAS BEEN RESET TO 1.

- Population Dropped. You may then see a message similar to this one:

POPULATION DROPPED = 0

During "trip generation," the population groups are assigned to the nearest available centroid (internal link location) in the network. It is possible that a population group inside the evacuation boundary could be assigned to a centroid location that is outside the evacuation boundary or to a link that has no access to an exit. If this occurs, those people are considered already outside the evacuation boundary at the start of the evacuation. Therefore, those people are NOT included in the evacuation and are reported as the "POPULATION DROPPED." You can get around this by using the Update Demographic Data function to manually reassign the offending population points to different centroids inside the evacuation boundary. Even if the part of the population is "dropped," you must still specify adequate host capacity for the total population when you define host locations. Finally you should see:

*** MSG: Cases written.
Press <RETURN> to continue.

The results of this process are two initial sets of IDYNEV input data corresponding to the case numbers given in the Case/Site Specification form: one traffic assignment case and one traffic simulation case. (Two case data files of the form *Cnnn*INP.DAT are created in the IDYNEV subdirectory.)

The only remaining task is to actually display the graphic representation of the evacuation network (see the following section).

5.3.10 Displaying (Enabling) the Roadway Network

As the final step after you have generated the initial evacuation cases, select ENABLE ROADWAY NETWORK on the network/case generation menu. (If you enable the network display before generating the cases, you may see spurious data--see the following note.)

The system reads the case input files and graphically displays the network, overlaying white network links and nodes on the site map. The network links are shown as simplified vectors resulting from the road data; the links may appear as straight lines that cross over areas where the actual underlying roadways do not occur. For example, the network graphics may show a link across a lake rather than around it.

A new network link-node file, *Cnnn*LNK.DAT, is created in your evacuation model input directory, where *nnn* is the traffic assignment case number.

Note: The roadway drawing program needs the proper case data before it can draw the correct network. If you enable the network display with no existing case data, the system will display a warning message. If you have generated case data but later redefine the evacuation boundary without regenerating the case data, enabling the network will display the old roadway network, which may be in the wrong location entirely.

5.4 Details of Automatic Evacuation Case Generation

With a few inputs from you, the system automatically does most of the work in generating a crude evacuation case for the new site:

- Network Building
- Trip Generation (Associating Populations with the Network)
- Evacuation Case and Case File Creation.

These three processes are described in the following sections. You need **NOT** read this material to generate a first-cut traffic network and evacuation case. You must read this material to understand the assumptions made so that you can later refine the traffic network and evacuation case to reflect a real situation.

5.4.1 Network Building

As a first step in network building, the system searches graphic data in memory for roads (major attribute 5) and an evacuation boundary or uses the link data extracted from Tiger Trails data. The system then builds a crude roadway network designed to evacuate the region defined by the evacuation boundary. The roadway network is based on the road and boundary data. If you specified the use of flood data in the Case/Site Specification form (Section 5.3.1), "flooded" areas will be eliminated, preventing the use of roads that might be under water.

— Topography Requirements and Assumptions

The site files, or other graphic database utilized by the site setup command procedure, must include a population data set and a ROADS or similar topography data file. For example: [username.SITES.site.INPUT.MAPS.RT]xxxRT.DMS where "username" is your login directory, "site" is a four-letter site code that designates the current site, and "xxx" is part of the file name. Each data item in the ROADS topography is associated with attributes-- numeric values that identify the data items and describe additional item features. Each attribute consists of three values: a Major code, a Minor code, and a Parameter value. All ROADS data items have attributes with a Major code of 5.

Minor codes and Parameters values define particular features of the data items. For example, the roadway for a link in the network might have the following attributes: Tiger Trains (TT) topography meets the requirements of network building but is so detailed that one normally will wish to use the TTXTTDAT utility, available as the menu entry EXTRACT LINKS FROM TIGER, to reduce the quantity of data to be considered for a network.

<u>Major Code</u>	<u>Minor Code</u>	<u>Param Value</u>	<u>Description</u>
5	52	0	Interstate highway
5	650	80	Interstate route number = 80
5	490	0	Road width is 76-85 feet
5	600	4	Number of lanes = 4

You can determine the attribute of data items by using the **Attribute Functions** of the utility map display and map editing programs. You can find other possible attributes of ROADS data, as well as further explanation of attributes, in a master list of attribute codes in the *IBS Utilities Guide*.

The network building process makes the following assumptions based on the attributes associated with items in the ROADS topography data (and with values in the network parameter file—CnnnEVACPARAM.DAT):

- **Number of Lanes:** The number of lanes on any included roadway item are estimated with the following rules (listed in order of precedence):
 - If the roadway item includes a Minor code of 600, the number of lanes is specified by the Parameter value.
 - If only a road width is specified (Minor codes 460 - 550), the number of lanes will be taken from the network parameter file for those attributes. For example, the system default values are:

<u>Minor Code</u>	<u>Road Width (feet)</u>	<u>Assumed Number of Lanes</u>
460	46-55	2 (1 each way)
470	56-65	2 (1 each way)
480	66-75	4 (2 each way)
490	76-85	4 (2 each way)
500	86-95	6 (3 each way)
510	96-105	6 (3 each way)
520	106-115	8 (4 each way)
530	116-125	8 (4 each way)
540	126-135	10 (5 each way)
550	136-145	10 (5 each way)

Generate Evacuation Network

- If the attributes do NOT specify the number of lanes or the road width, the number of lanes is taken from the network parameter file for the existing attributes.
- If there is a conflict among the attributes of a roadway item, the LAST attribute of those in conflict takes precedence (that is, the last in the roadway item's ordered list of attributes).

Unless you modify the network parameter file (QUALIFY ROADWAY SELECTION), each link between two nodes of the roadway network is assumed to be paired with another link in the opposite direction: the two links then specify a two-way road. If you know that a particular road is one-way only, it is your responsibility to either modify the network parameter file appropriately or delete the spurious link later when you refine the traffic network.

- **Free-Flow Speeds:** The system default network parameter file assumes that the free-flow (maximum unrestricted) vehicle speed allowed on a network link is 45 miles per hour (mph) unless the following apply:

<u>Minor Code(s)</u>	<u>Free-Flow Speed</u>	<u>Description</u>
	60	Interstate Highways52
	60	U.S. Highways54
	60	Limited access routes 350
	60	Toll roads360
70, 80 90	55	Primary (Class 1) roads60,
	55	Interstate connector175
	55	Toll road connectors 365
58	45	Secondary state/county roads
130	45	Class 2 roads100, 110, 120,
	45	Clover leaf or interchange250
	35	Class 3 road or street140
	35	Class 4 road or street150
	25	Cul-de-sac40
	25	Traffic circle 240

- **"Non-Link" Items:** The following data items are NOT used in building the roadway network:

Minor Code(s) Description

51, 53, 55, 57All roads proposed or under construction

160, 170Trails (Class 5)

181Foot bridges

190Ferry crossings

200Parking area perimeters

210Arbitrary extensions of line (join or closure)

330Roads submerged or in ford

—
links determined by SLOSH flood data to be under one or more feet of water (see Section 5.3.1).

— Movements at Intersections of Network Links

For each node (intersection) in the roadway network, all links that connect at a node are assumed to be accessible from link to link by some movement either straight through or turning through the intersection. This may not always be a true assumption, but none of the topographic data sets currently available to the IBS contain details of turn movements between roadway links. Therefore the following method is used to assign likely "through" or "turn" movements for links at each network node.

- a. Where only two links connect, a subject link and one downstream link, the downstream link is a through movement from the subject link.
- b. Where three or more links connect, if one downstream link has attributes which match the subject link, it is a through movement from the subject link and the remaining downstream link(s) are designated based on the angle of turn movement. Turn movement is based on the coordinates of link end points.
- c. When no through movement is determined based on link attributes, and four or more links connect, the through movement is the downstream movement whose angle deviates the least from the subject link.
- d. If five links connect, the downstream link which does not turn right most or left most and is not designated through due to it deviating least from the subject link is designated a diagonal turn movement.
- e. Any link returning from a through movement is designated as opposing left turning traffic from a subject link.

— Dropping Dead-End Links

If no downstream movement is found for a subject link and the subject link is not an exit, it is removed from the evacuation network. This process is recursive in that the elimination of one or more links may result in the elimination of other links which were upstream links of those just deleted. The process is continued until no more links can be removed from the network.

5.4.2 Trip Generation: Associating Populations with the Network

In a second stage, called **Trip Generation**, the system establishes how the populations will enter the network through **centroids** (internal link locations on the network). Trip Generation first assigns specific populations to specific centroids on the network; these population point/centroid assignments are stored in another population file. Then the system calculates the rate at which the population (in vehicles) will enter the network through each centroid. Several assumptions are made during Trip Generation:

- **Topography Requirements and Assumptions.** The site files, or other graphic database utilized by the site setup command procedure, must include a population data set such as:

[username.SITES.site.INPUT.MAPS.PO]xxxPO.DMS where "username" is your login directory, "site" is a four-letter site code for the current site, and "xxx" is part of the file name or such population data must be included in another topographic overlay. Population values are interpreted from attribute values associated with each population point:

- Major attribute codes of 9 designate a population value.
- If the Minor attribute code is 10, then the population value = the Parameter value.
- If the Minor attribute code is 20, then the population value = the Parameter value times 10.
- If the Minor attribute code is 30, then the population value = the Parameter value times 100.
- If the Minor attribute code is 40, then the population value = the Parameter value times 1000.
- **Population Point/Centroid Assignments:** The assignment of population points to centroids in the network is based on the following criteria:
 - Population points are assigned to the closest available centroid.
 - Where two centroids are equidistant from a population point (as in the case of two links traveling in opposite directions between the same nodes), the population point will be assigned to the centroid having the lesser current population (from other population points assigned previously). This provides a somewhat balanced selection of centroids for more equal loading of the network without dividing the population of a specific population point.
- **Percent of Population Evacuated:** By default, all (100%) of the population point populations assigned to internal centroids will be loaded onto the network. This value is not supported by statistical evidence and is used only to be conservative. When you generate a first-cut evacuation case, the system will give you a chance to change this default. Note this percentage is not a direct input to IDYNEV, which assumes total evacuation. Trip Generation uses the input in calculating the number of vehicle trips per hour started at centroids.
- **Number of Persons per Vehicle:** All traffic will evacuate by car with a default average of 1.3 persons per car (unless you altered this value—see Section 5.3.9). This value will be included in the load factors data of the IDYNEV model (see Section 6.4.17).
- **Loading Period and Percentages:** The information you provide in the SPECIFY LOAD CURVE form, combined with the number of people per vehicle, evacuation percentage and population assigned to each of the centroids is used to

Generate Evacuation Network

calculate the number of vehicles per hour to be loaded at each centroid for each time period during which you specified loading would take place. If a centroid has a non zero number of people to load in a given time period but has fewer people than the specified number of people per vehicle it will load one vehicle per hour during that time period. For this reason it may be that the total number of trips generated multiplied by the number of people per vehicle will exceed the total number of people to be evacuated by several percentage points (likely if a detailed population set is used for a reveal area).

These rates are subject to a discharge limit of 9999 vehicles per hour imposed by the IDYNEV model on centroids of the network. If the discharge plan at any time exceeds this discharge limit (only likely if crude data sets are used in an urban area), the Trip Generation function will automatically queue vehicles, attempting to discharge them during later time periods. For example, consider the following simplified case in which 100,000 vehicles are discharged from a given centroid:

Evacuation of 100,000 Vehicles				
<u>Evacuation Hour</u>	<u>%</u>	<u>Discharge Required</u>	<u>Discharge Allowed</u>	<u>Vehicles Queued for Discharge</u>
1	45.5	45500	9999	35501
2	12.2	12200	9999	37702
3	12.5	12500	9999	40203
4	10.6	10600	9999	40804
5	5.7	5700	9999	36505
6	4.2	4200	9999	30706
7	3.9	3900	9999	24607
8	0.9	900	9999	15508
10	1.3	1300	9999	6809
11	0.3	300	7109	0
12	1.6	1600	1600	0

In addition to the discharge limit of 9999 vehicles per hour, the model imposes a second limit, based on link capacity, which generally restricts loading rates to more modest figures. The link capacity (maximum vehicles per hour) is computed as $[(3600/\text{headway}) * .90]$, where headway is expressed in seconds. A headway of 2.3 seconds is used during Trip Generation. This formula for link capacity is an internal tool of Trip Generation and has no direct relation to the IDYNEV model.

- **Traffic Loading in Extreme Circumstances:** In some circumstances it is possible that some traffic will still not be loaded onto the evacuation network at the end of 10 Time Periods. Trip Generation will continue loading over eight more time periods (TP 11 through TP 18) of 10 hours each, during which it attempts to generate the remaining trips and load the remaining population onto the network. If the centroid queues are still not empty at the end of the extra time periods, the system will display a diagnostic message instructing you to review a report located in your login directory. This report provides further information. If you receive this message, use the Update Demographic Data function to check the following:

- whether the assignments of population points to internal centroids are reasonable
- whether the population values of the population points are reasonable
- whether links should be added in the vicinity of some population points.

Also, check to see whether the number of people per vehicle is reasonable. If actions based on these checks are not adequate to resolve the traffic loading problem, you can modify the evacuation model input data (Section 6.4) to create "pseudo links," nonexistent branches to "real" links with extra internal centroids that may be used to load vehicles onto the network.

5.4.3 Evacuation Cases and Case Files Created

Several data files are created as a result of first-cut network and case generation. All files are created in the evacuation model input directory:

`[username.SITES.site.INPUT.DYNEV]`

where *username* is your login directory and *site* is the four-letter site code for the current site. All created files have names beginning with *Cnnn*, where *nnn* is the three-digit case number.

Two input cases for the evacuation model are created. These are the cases listed on the Case/Site Specification form Section 5.3.1.

- First-Cut TA (Traffic Assignment) Case (CnnnINP.DAT): input data for a traffic assignment run. The purpose of a traffic assignment is to establish the optimal routing patterns for traffic to get from origins to destinations on the network.
- First-Cut Sim (Simulation) Case (CnnnINP.DAT): input data for a simulation run. A complete traffic simulation predicts the progress and flow of populations over the traffic network. Note that this case will NOT automatically include the turn movement data required in data form type 21 (Section 6.4.12).

The following data files are also created during network and case generation. (The case number *nnn* for these three files is the traffic assignment case number.)

- Network Parameter File (CnnnEVACPARAM.DAT): a binary file that contains general default values for necessary roadway data that do not occur in the ROADS topography data.
- Evacuation Boundary and Host Location File (CnnnEVAC.DMS): a DMS graphics file that contains the evacuation boundary and the host location data.
- Network Link-Node File (CnnnLNK.DAT): a binary file that contains locational data used to draw the roadway network on the site map.

Section 6

Dynamic Evacuation Model

IBS Menu: Select EMERGENCY ACTIVITIES—EVACUATION

Evacuation Menu: Select A (Change Input Data) --> See Section 6.4.

IBSSH Menu: Select (Models Graphic) or (Models Non-Graphic)

Model Library Menu: Select C (Dynamic Evacuation Model - IDYNEV)

This section describes the Interactive DYnamic Network EVacuation model (IDYNEV), a computer model that helps predict the progress of area evacuations by indicating the changing conditions of population movement over a traffic network. This section covers the following topics:

6.1 Description of IDYNEV

6.2 What You Need to Know to Use IDYNEV

6.3 Starting an IDYNEV Work Session

- Selecting Graphic or Text Mode
- Selecting Evacuation Model Input or the IDYNEV Function Selection Menu

6.4 Modifying Evacuation Model Input Data

- Key Terminology for Evacuation Model Input
- Starting a Data Input Session with PREDYNGR
- Choosing a Time Period and Subnetwork
- Selecting Which and How Input Data Will Be Modified
- Using Data Input Forms

6.5 Graphics Mode of Modifying Input to the Evacuation Model

6.6 Using the IDYNEV Function Selection Menu

- Execute Evacuation (*interactive and batch*)
- Display Results on Map
- Display Base Map
- Print/Display Reports
- Change Case/Site
- Perform File Maintenance

6.7 If You Have Problems

6.8 Status and Diagnostic Messages

For references see **Section 10.21.**

6.1 Description of IDYNEV

The evacuation model is called IDYNEV (Interactive DYnamic Network EVacuation model). The goal of the evacuation model is to accomplish a **traffic simulation**. A traffic simulation predicts the progress and flow of populations over a traffic network. The output of a traffic simulation includes travel times, vehicle counts, queues and delays, and person throughput over the **links** (street segments) and **nodes** (intersections) of the network. Many results of the traffic simulation can be viewed graphically. Other results serve as input to other models, such as the Mesorad dose model.

Before IDYNEV can be used to simulate traffic flow during an evacuation, a complete set of input data must be assembled. This input includes detailed descriptions of the traffic network, composition, and volume. Several steps must be taken to assemble this data. These steps are summarized in Table 6.1.

Step 1. First, the roadway network and traffic loading (vehicle trips per hour from origins) must be defined for the evacuation area. The required roadway network information includes the location of entry and exit points, how traffic lanes are directed and restricted, and how signals and signs are set up to control traffic at each intersection. Most of this data must be input "manually," as described in Section 6.4. (IDYNEV's input data sets are constructed by using a special interactive input program called PREDYNGR.) A crude, "first-cut" approximation of the roadway network can be created automatically by using the "Generate Evacuation Network" function (Section 5). This function extracts roadway and population data from the national database as a basis for approximating the network. It is able to locate nodes of the network accurately with respect to viewed map overlays but lacks information about signal timing. The resulting data can then be refined manually to correct details, as needed. After this data has been input once, it becomes part of the current case.

Step 2. When the roadway and traffic loading data are defined, the **Trip Distribution** and/or **Traffic Assignment** portion of the IDYNEV system is executed to determine a) how the number of vehicle trips will be distributed among the exit points of the network and/or b) what turn movements must be assigned at each intersection to route traffic from their **origins** to their destinations.

Step 3. Next, the results of the traffic assignment are ~~merged~~ merged with (added to the existing data for) the current case. Now the available input data are sufficient to describe the initial conditions of the traffic network, unaffected by outside influences. That is, the data describe the starting conditions of the first **time period** (time period 1) of a possible simulation.

Step 4. Then, the evacuation model (IDYNEV) could be executed with the current time period 1 data. The results would describe a traffic simulation controlled only by the initial conditions of the network. If outside factors are considered to affect the traffic conditions during the evacuation, more input data must be added to

describe their effects: additional data would be needed for subsequent sequential time periods. The data for each succeeding time period would describe any conditions that differed from the preceding time period.

Table 6.1. Task Steps in a Typical Traffic Simulation

Task	Software Used
1. Define roadway network and traffic loading (vehicle trips per hour from origins) as input for traffic distribution and assignment.	Use Generate Evacuation Network (Section 5) to extract roadway and population info from the national map database; use PREDYNGR (Section 6.4) to refine the network and loading assumptions.
2. a) Determine how the number of trips will be distributed among the exit points of the network. b) Determine optimal routings of traffic (turn movements) from origins to destinations.	Execute the Trip Distribution/Traffic Assignment portion of IDYNEV for time period 1 (see Table 6.2 in Section 6.4.4).
3. Merge the traffic assignment results into the current case for time period 1.	Use the "MERGE Time Period" option from the Block Change Options menu (see Figure 6.6 in Section 6.4.4).
4. Define specific input for later periods (if any) of the traffic simulation.	Use PREDYNGR to modify data for each time period after time period 1.
5. Simulate traffic flow over the network, obtaining traffic performance statistics for links & subnetworks.	Execute the IDYNEV simulation (Section 6.6.1).
6. Report or display results of the evacuation simulation.	Graphically depict traffic data for each time step with "Display Results on Map" (Section 6.6.3). Print reports or display them on your terminal screen with "Print/Display Reports" (Section 6.6.5).

Step 5. IDYNEV is then executed with a complete set of input data, producing traffic performance statistics.

Step 6. After IDYNEV finishes execution, the system makes the tailored model results available for printing or display. The progress of a simulated evacuation can be graphically depicted and overlaid on the site map by using the "Display Results On Map" function (Section 6.6.3). The system uses the map database and map display software for graphic display of background maps. When you display evacuation model results, you can specify what static features are displayed as the map background: political boundaries, waterways, etc. This interactive graphics display is a powerful tool for understanding the model results.

After modeling an evacuation, you can evaluate the results and adjust the model input data. Different "cases" (input and output data sets) that describe specific situations at a specific site can be saved for further reference or modification.

Note that you may combine steps 2, 3, and 5 if executing IDYNEV from the IBS Evacuation Menu by responding "Yes" to the prompt, "Do you wish to execute a related simulation case? (Y/N) [Y] = =>"

IDYNEV can be used to

- generate scenarios for emergency preparedness exercises
- evaluate the timing of emergency responses made during exercises
- evaluate the feasibility and acceptability of evacuation plans.

6.2 What You Need to Know to Use IDYNEV

The materials in this section are intended for any user of the IDYNEV model. IDYNEV is designed to step you through the creation of evacuation scenarios. You can use this section as a step-by-step guide through the IDYNEV functions. When you have selected IDYNEV from the Models menu, the system can supply background information with its HELP features (the <F6> key).

The evacuation model results will be only as meaningful as the input. Some static details, such as the physical characteristics of a site, would rarely be changed by a typical user. Other inputs would require the technical expertise of a traffic engineer. For example, consultation with a traffic engineer is suggested for specifying road network and traffic parameters. Generally, a system manager and technical consultants will set up standard cases the typical user can modify in less fundamental ways.

If you have problems with IDYNEV, refer to Section 6.7 (**If You Have Problems**) and Section 6.8 (**Status and Diagnostic Messages**).

6.3 Starting an IDYNEV Work Session

A session is a continuous period of time during which you use IDYNEV. You will start, control, and stop a session at your workstation. The session starts when you select the Evacuation Model from the Models menu of IBSSH or Evacuation, from the Emergency Activities menu of IBS.

6.3.1 Starting an IDYNEV Work Session from IBSSH

The software for IDYNEV is generally divided into two separate parts:

- Modification of IDYNEV input data (select PREDYN from the Models menu)
- All other functions (model execution, selection of output reports, display of model results, site and case control, and file maintenance).

The general flow of IDYNEV operations is depicted in Figure 6.1.

Figure 6.1. General Flow of IDYNEV Operations in IBSSH

6.3.2 Selecting Graphic or Text Mode

If you wish to use graphic interactives, see maps while doing your work. Be sure to select *Models (Graphic)*.

6.3.3 Selecting Evacuation Model Input or the IDYNEV Function Menu

After you select graphic or text menus, you select from a list of models and related programs:

- If you select PREDYN, you will begin a data entry sequence for modifying IDYNEV input (Section 6.4).
- If you select Evacuation Model, the system will present a menu of IDYNEV functions, including model execution, selection of output reports, display of model results, site and case control, and file maintenance. Section 6.6 explains the options of the IDYNEV Function Selection menu.

6.3.4 Starting IDYNEV from the IBS Program

The IBS Evacuation menu provides access to the graphic version of PREDYNGR. To start IDYNEV, perform the following procedure:

1. Select the Emergency Activities option from the IBS Main Menu.
2. Select the Evacuation option from the Emergency Activities menu.
3. Select the Execute Evac Model from the Evacuation menu.
4. Select option A, Change Input Data from the Evacuation Model menu.

6.4 Modifying Evacuation Model Input Data

When you select PREDYN, your screen will move to the **PREDYNGR user interface**, which enables you to create or modify the input data for the evacuation model, IDYNEV. As background for users who are unfamiliar with the evacuation model, Sections 6.4.1 supplies a list of key terminology for the evacuation model. Section 6.4.2 (**Starting a Data Input Session with PREDYNGR**) and subsequent sections describe the model input data and the actual process of modifying the data.

6.4.1 Key Terminology for Evacuation Model Input

The following terms are used in the process of modifying input to the Interactive DYnamic Network EVacuation model, IDYNEV.

- **Centroids:** A centroid is an **internal node** or point location at which traffic can enter or leave the **traffic network** internally. Each centroid is associated with 1) a population or number of vehicles that can enter or exit at the centroid, and 2) one **internal link** (street segment) of the network. A link can be associated with only one internal centroid. The traffic simulation model treats each centroid as a **source** or **sink** point.
- **Channelization:** Channelization is the description of how traffic is directed or restricted for particular lanes of a roadway; for example, a curb lane might be right-turn-only.
- **Links:** Links generally represent street or freeway segments of the traffic network. Links are unidirectional (that is, a two-way street would be represented by two links).
 - **Internal links** are the standard links inside the network.
 - **Entry links** allow traffic to enter the network.
 - **Exit links** allow traffic to leave the network.
 - **Interface links** are links at the edge of a **subnetwork**.
 - Entry interface links** allow traffic to enter a subnetwork from another adjacent subnetwork.
 - Exit interface links** allow traffic to leave a subnetwork and pass onto another adjacent subnetwork.
- **Nodes:** Nodes generally represent intersections or point locations where the road changes (adds a lane, climbs a grade, entrance to shopping mall, etc.).
 - **Interface nodes** are nodes at the edge of a subnetwork.
- **Origin:** An origin, or "source" point, is a point from which vehicles enter the traffic network. An origin represents a population center or flow of traffic from outside the network area.

- **PREDYNGR**: PREDYNGR is an interactive program that helps you prepare the input for the evacuation model, assembling the data that define the physical traffic environment. PREDYNGR gives you the option of manually entering data into text screen "forms" or of modifying a graphic depiction of the traffic network on the site map. PREDYNGR -- **PRE**processor for input to the **DYN**amic evacuation model with a **GR**aphics mode of data input. PREDYN performs similar functions but without the use of graphics. Your choice of PREDYNGR or PREDYN is based on your selection of graphics or non-graphics modules.
- **Sources and Sinks**: A source/sink centroid is a point at which traffic can enter or exit from the network by way of a specific **internal link**. A positive net volume of traffic onto the network defines a source. A negative net volume of traffic from the network defines a sink. The terms "source" and "origin" are synonymous.
- **Subnetwork**: A subnetwork is a portion of a larger traffic network that has been subdivided into multiple portions. IDYNEV processes each subnetwork independently, allowing the treatment of large, regional evacuations. Adjacent subnetworks are connected by **interface nodes** and **interface links** located at the interface of the subnetworks.
- **Time Period and Time Interval**: A **time period** (TP) represents a length of time during which all time-dependent inputs to the evacuation model remain constant. During the course of an evacuation, user-specified changes to the traffic situation (such as vehicle volumes) are represented by the input differences between one time period and the next. Each time period is subdivided into **time intervals** (or **time steps**). Within each time interval (TI), the traffic simulation model assembles data that describes the movement of traffic over the **links** of the **traffic network**.
- **Traffic Assignment**: Traffic assignment is the modeling process (submodel of the IDYNEV system) that establishes the optimal routing patterns (turn movements) for traffic to get from **origins** to destinations on the network. This occurs after a **trip distribution**. A traffic assignment must be executed to define network initial conditions before a traffic simulation can be executed. (Although the data produced by a traffic assignment might be input manually, it is usually most practical to use the traffic assignment submodel.)
- **Traffic Network**: The traffic network is the system of streets (links), intersections (nodes), and their detailed characteristics.
- **Trip Distribution**: Trip distribution is the modeling process that defines how the number of trips starting at various origins will be distributed to the various exit points of the network. This submodel of the IDYNEV system can be executed whenever **traffic assignment** is executed.

6.4.2 Starting a Data Input Session with PREDYNGR

When you select PREDYN from the IBSSH Models menu or select option A (Change Input Data) from the Execute Evac Model menu, PREDYNGR will greet you with a layered series of input interactions as depicted in Figure 6.2. First, you will have the opportunity to modify **link capacities** and **free-flow speeds**; these are global modifications that affect the entire traffic network. Next, you will choose to operate on data that is associated with a specific **time period** and traffic **subnetwork**.

For that time period and subnetwork, you will choose **data input forms** for creating or modifying IDYNEV input data.

For detailed descriptions of the necessary input data, refer to the evacuation/traffic simulation reference documents. Visual representations of these data forms appear in FEMA-REP-6, the *PREDYN Users Guide*. (PREDYN was the interactive forms display processor that preceded PREDYNGR. Although other aspects of the interface have changed, the input data forms in PREDYNGR are substantially the same as those shown in the *PREDYN Users Guide*, with noted exceptions.)

Figure 6.2. Overview of an PREDYNEV Data Input Session

Modifying the Capacities and Free-Flow Speeds of Network Links. On entry to PREDYNGR, prompts ask you whether you wish to modify the capacities or free-flow speeds for all the traffic network links:

DO YOU WISH TO MODIFY THE CAPACITIES OF ALL LINKS?

DO YOU WISH TO MODIFY THE FREE-FLOW SPEEDS ON ALL LINKS?

In general, you should type N to answer "No" to both these questions. You will then be asked to choose a time period (TP) for which to modify input data. Section 6.4.3 explains how to choose a time period.

Typing Y gives you the opportunity to globally adjust the capacities or free-flow speeds of the network links by a percentage of 1 to 500 (100% = no change). These features should be used when performing sensitivity studies to predict the effects of weather conditions such as rain, snow, icy roads, fog, etc. If you choose to adjust the capacities or free-flow speeds of the network links, the following prompts will appear:

Enter percentage to be applied to all capacities (1-500)

Enter percentage to be applied to all free-flow speeds (1-500)

After all changes to capacities and/or free-flow speeds have been made, you can terminate execution of PREDYNGR or continue editing the modified data:

MODIFICATIONS COMPLETE. ENTER "DONE", OR "CONTINUE":

Ending (or Continuing) the Data Input Session. To end the data input session here, enter DONE; otherwise enter CONTINUE. Entering CONTINUE will display the Time Period Menu explained in Section 6.4.3. Entering DONE here (or at other opportunities in the program) will display a prompt asking you whether you wish network changes to be made permanent:

Save the network changes? (Y/N) [Y]==>

You will always see this prompt as you end a data input session. Typing N will cancel any data modifications you have made and return the screen to the IBSSH menu. Typing Y will display the current case description and a prompt asking whether you wish to change the Case Description (because you have changed the case input):

DO YOU WISH TO CHANGE THE CASE DESCRIPTION? (Y/N) [N]==>

Typing Y will present you with Form 00, where you can alter the description. After changing the Current Case Description, press <GOLD> <Z>. The preceding prompt is then repeated.

Typing N will display messages indicating that the new data records are being copied to disk. When the copying is complete, you will exit from PREDYNGR.

6.4.3 Specifying a Time Period and Subnetwork

To identify what input data you wish to change, the system will ask you to specify a **time period** number and a **subnetwork** number. The system will then offer a choice of methods for modifying the input data.

— Specifying a Time Period

At the beginning of each input session, the system will ask you to indicate a time period (TP) and whether you want to create/insert a new time period or revise the input data for an existing time period. The system will indicate time periods for which data already exists, as shown in Figure 6.4. Generally, network data resides in TP 1; higher TPs are used to describe traffic flow into and on the system.

```

                                TIME PERIODS

Data already exists for time period(s):
1, 2, 3, 4, 5

You now have the option to

C      Create
I      Insert
R      Revise

Enter selection (C, I, R, X). R

Enter time period. _

```

Figure 6.4. Example of Time Periods Form

— Specifying a Subnetwork

The software described in Section 5 (Generate Evacuation Network) can automatically split a large traffic network into smaller adjacent subnetworks. After the interactions shown in Figure 6.4, the system will ask you to specify a subnetwork number (1 to 255):

```
ENTER SUBNETWORK # TO EDIT (CURRENT MAX = 1) ==>
```

You must specify a subnetwork number. That is, once you begin modifying networks with PREDYNGR, you must choose and modify one subnetwork at a time. There is no default subnetwork number. A <RETURN> without specifying a subnetwork number is interpreted as an exit.

6.4.4 Selecting Which and How Input Data Will Be Modified

At this point, you are ready to specify which data that you want to modify (what data form) and how you want to modify it (add, delete, modify, etc.).

— Selecting a Type of Data Form

After you specify a subnetwork number, the system will prompt you:

Enter record type to create/edit, menu, graphics (#|M|G) ==>

This prompt indicates that you have the following three options for choosing the type of data to be modified and one option to quit (EXIT).

Note: The GRAPHICS option is NOT available (and will not appear) in "Text" mode (see Section 6.3.1).

- Enter a record type: Type the identification number of the specific data input record type. Then press <RETURN>. (Record types and descriptions are listed in Figure 6.5.)
- Select a MENU that lists the record types: Type M for MENU. Then press <RETURN>. The system then displays the menu of corresponding data input form types shown in Figure 6.5. Sections 6.4.5 through 6.4.21 describe the 19 types of forms.

Select a form type from the menu: Type the number of the form type. Then press <RETURN>. (If you just press <RETURN>, the system will assume form type 00.)
- Select the GRAPHICS mode of data input: Type G for GRAPHICS. Then press <RETURN>. This mode of data input enables you to edit the traffic network graphically by manipulation of graphic elements on a map of the area (see Section 6.5). The system automatically presents data input forms that you must change to accompany your graphic modifications.
- EXIT from data input: Type <CTRL>-Z to exit. The system will ask you, "SAVE THE NETWORK CHANGES? (Y/N) [Y] ==>" (This same prompt occurs whenever you finish modifying data, as described at the end of Section 6.4.2. If you enter Y for "yes," the system will save all the changes you have made to the input data for the current case. If you enter N for "no," then the system will NOT save any changes made during the data input session. In either case, the system will then return to the Model menu.)

MENU OF FORM TYPES

FORM TYPE	DESCRIPTION
00	Title
01	Identification
02	Run control
03	Time period specification
04 (05)	Time-step data, output options
06	Origin control
11	Link Characteristics
21	Link turn movements
35 (36)	Sign or signal control data
49	Evacuation data
50	Entry link volumes
51	Source/sink volumes
52	Load factors
175	Traffic assignment parameters
176	Origin-trip production volumes
177	Internal centroids
178	Destination capacities

Enter form type

Figure 6.5. Menu of Form Types for Evacuation Model Input

— Selecting How the Data Will Be Modified (FUNCTION TYPE)

If you select any form but form type 00, the system will ask you what you want to do to the form. The following prompt serves as a menu of the functional options that follow:

Enter Function Type (Add, DElete, Modify, Block change, DOne):

Select the function type: Type A (add), DE (delete), M (modify), B (block changes), or DO (done). Then press <RETURN>.

- A: Displays an empty data form for you to fill out. Each time you press <RETURN>, the system will supply another data form.
- DE: Displays the designated form and gives you an opportunity to delete each one of that type.
- M: Displays the designated form and gives you an opportunity to modify each one of that type.

Dynamic Evacuation Model

- **B:** Presents the Block Changes Options menu shown in Figure 6.6.
 - **COPY Time Period:** This option copies all records of a given form type within a specified time period and inserts them in the current time period.
 - **DELETE Time Period:** This option deletes all records of a selected type.
 - **MERGE Time Period:** Used for the time period 1 only, this option can combine all the turn movement data created as a result of a traffic assignment execution into the current case -- necessary before a **traffic simulation** can be executed. You must specify the case number of a completed traffic assignment in response to a prompt for the number of a traffic assignment case to be merged.
- **DO:** When you select DO, the system checks to see if you are finished with this data input session:

Enter Record Type, Menu, new Subnetwork, new TP, or Graphics (#|M|S|T|G) ==>

- If you enter nothing or enter (CTRL>-3, then the system will present the ending sequence of prompts, starting with, "Save the network changes? (Y/N) [Y] ==>" (Section 6.4.2).

BLOCK CHANGE OPTIONS	
Option code	Option
C	COPY Time Period
D	DELETE Time Period
M	MERGE Time Period
E	Exit from the block change mode

Enter OPTION CODE corresponding to option of your choice:

Figure 6.6. Menu of Block Change Options

Table 6.2 shows the data input requirements for executing traffic assignments and traffic simulations. Sections 6.4.5 through 6.4.21 discuss each of the form types shown in Table 6.2. These discussions contain more detailed explanations of the data input requirements. Note that default data are available for certain data inputs. These default data values need not be altered unless, for some reason, these are unsatisfactory for your purposes.

After these form types discussions, the graphics mode of data input (with automatic forms selection) is described in Section 6.5.

Table 6.2. Data Input Requirements for Traffic Assignment and Simulation

Form Type	Description	Traffic Assignment TP=1 only	Traffic Simulation	
			TP=1	TP>1
00	Title	Req'd	Req'd	-
01	Identification	Req'd	Req'd	-
02	Run control	Req'd	Req'd	-
03	Time period specification	Req'd	Req'd	-
04 (05)	Time step data, output options	Req'd	Req'd	-
06	Origin control	Req'd	Req'd	-
11	Link characteristics (a)	Req'd	Req'd	Opt'l
21	Link turn movements (a)	Opt'l	Req'd	Opt'l
35 (36)	Sign or signal control data	Req'd	Req'd	-
49	Evacuation data (b)	-	Opt'l	Opt'l
50	Entry link volumes (c)	-	Opt'l	Opt'l
51	Source/sink volumes (c)	-	Opt'l	Opt'l
52	Load factors	-	Opt'l	Opt'l
175	Traffic assignment parameters	Opt'l	-	-
176	Origin-trip production volumes	Req'd	-	-
177	Internal centroids	Opt'l	-	-
178	Destination capacities	Req'd	-	-

TP = Time period Req'd = Required Opt'l = Optional

- (a) Information can be input for later time periods to reflect changes over time.
- (b) If form 49 is not present, the default is a full evacuation. Form 49 is required for any other situation in which sector statistics are required.
- (c) These data are required for each time period when such traffic movement is simulated.

6.4.5 Title (Type 00)

Use the Title form to enter any identifying information that you wish. The form is a eight-line window that scrolls a set of eight lines as you enter lines (using <RETURN>) or use the arrow keys (F1, F2, F3, F4). You can enter up to 50 separate lines of information. At a minimum, you should enter a brief case description (indicating traffic assignment or simulation), your name, and the current date.

6.4.6 Identification (Type 01)

The Identification form has spaces for

- * User name
- * Date: (month, day, year)
- * Name of agency
- * Run identification number: A run number, not a case number.

These items are for identification only. They have no effect on running the evacuation model. However, it is critical that each case has a unique title and identification data with clear differentiation of traffic assignment and simulation data sets.

6.4.7 Run Control (Type 02)

The items on this form specify the type of run and several control parameters.

- * Run type: Enter "1" for simulation or "2" for traffic assignment. A negative number will cause IDYNEV to perform several input diagnostic tests before terminating further execution of the model.
- * Fill time: Enter a number of minutes. If this number is negative, the evacuation model will exit at the end of this time if equilibrium (traffic input to the network = traffic output from the network) has not been attained. If this number is positive, the model will continue processing, even if equilibrium is not attained during this period.
- * Factor by which Link Capacity is reduced during saturated conditions:
A percent.
- * Clock time at simulation start: Enter hours and minutes.

- * **TRAD Model selection:** Enter "1" to select the TRAD trip distribution and traffic assignment model (leave blank when trip table is input directly to bypass TRAD or when performing simulation).

6.4.8 Time Period Specification (Type 03)

Use the Time Period Specification form to indicate the durations of the time periods (TPs). A TP is a length of time during which all input data remain constant. For example, if traffic loads onto a network at one rate for 15 minutes and another rate for the next 30 minutes, TP 1 would be 15 minutes and TP 2 would be 30 minutes.

- * **Simulation time period:** Enter the length of each TP in units of tenths of minutes.

For example, in the above case you would enter 150 for TP 1 and 300 for TP 2.

- Note:**
1. A Traffic Assignment run must have only one TP. A simulation run can have up to 19 TPs but must include a final TP in which all entering traffic volumes are zero (0).
 2. Each TP must be a multiple of the time interval (TI) specified on form type 04. For example, if the TI is 80 seconds (1-1/3 minutes), then permissible TP entries would include 40 (4 minutes), 80 (8 minutes), 120 (12 minutes), etc.

6.4.9 Time Step Data (Type 04) and Output Options (Type 05)

The Time Step Data form and Output Options form appear together on the screen.

Time Step Data

- * **Time interval length:** Enter a number of seconds within the range of 60 to 600. The default is 60 seconds.

The time interval (TI) is used together with the number of time slices (see the following item) to govern how finely the model computes its results.

- * **Number of time slices per time interval:** Enter a number where the ratio (time interval/number of slices) is within the range of 2 to 5 seconds.

The model divides each TI into **time slices**. The time slice defines the resolution of the histograms used by the IDYNEV logic. The number of slices in each TI must be set so the length of each slice is in the range of 2 to 5 seconds. For example, if the TI is 80 seconds, the number of time slices

should be between 16 and 40. If this entry is left blank, the model will calculate a suitable number of time slices.

Output Options

- * Number of time intervals between outputs of cumulative simulation statistics: Enter a number. A blank means cumulative output only at the end of each time period.
- * Number of time intervals between outputs of intermediate simulation statistics: Enter a number. A blank means no intermediate simulation statistics.
- * Sector output code: Enter "1" for statistics.

A sector output code of "1" **must** be entered to obtain sector statistics. These statistics pertain to vehicle activities for each time step during the simulation. These statistics can be used by the met/dose model Mesorad to estimate doses for people traveling within the evacuation area around a nuclear power plant. Two sets of statistics are provided: one set for a 20 x 20 sector Cartesian grid and one set for a 240-sector Polar grid. A sector output code of "1" **must** be entered for IDYNEV to honor the data provided in form type 49 (see Section 6.4.14).

6.4.10 Origin Control (Type 06)

The Origin Control form enables you to specify an evacuation case origin (longitude and latitude) that is independent of site origin. The case origin specified in this form is used to define the lower left corner of the model space. Offsets from this case origin are then used to spatially locate the graphic representation of the traffic network on background maps. This origin control is an enhancement to the evacuation model for this system. Before this enhancement, the evacuation model automatically assumed the site center was located a standard distance northeast of the model case origin. The addition of the case origin allows multiple IDYNEV cases with different origins to be specified in the same site area.

- * Longitude: Enter a number of degrees of longitude in three parts:
 - Left of decimal point: an integer number of degrees (-180 to 180)
 - 1st four decimal places: an integer number of 1/10,000 deg
 - 2nd four decimal places: an integer number of 1/100,000,000 deg
- * Latitude: Enter a number of degrees of latitude in three parts:
 - Left of decimal point: an integer number of degrees (-90 to 90)
 - 1st four decimal places: an integer number of 1/10,000 deg
 - 2nd four decimal places: an integer number of 1/100,000,000 deg

Using Old Evacuation Cases. The current system implementation of the evacuation model requires this case origin: if you attempt to use a case that does not have a record type 06, the system displays default longitude and latitude values calculated from the site center and prompts you to change or accept these values as a case origin for record type 06.

6.4.11 Link Characteristics (Type 11)

When you select the Link Characteristics form and the "Modify" option, the system will present a form with the following parameters. Specifying items on this form would normally be the task of a traffic engineer, but if the data is available, anyone could enter it.

- * Upstream node number
- * Downstream node number
- * Link length: Enter a number in units of miles multiplied by 100.
- * Number of full lanes
- * Number of lanes in left- and right-turn pockets
- * Grade percent: Enter a grade percent in the range -9% to 9%.
- * Lane channelization: Enter 0 for unrestricted, 1 for left-turn vehicles only, 2 for buses only, 3 for closed lanes, or 4 for right-turn vehicles only for each lane in the link.
- * Receiving nodes: Enter node numbers of downstream receiving nodes for through traffic and for left-, right-, and diagonal-turning traffic. (Enter negative numbers for left-diagonal-turning traffic.)
- * Opposing left turn node: Enter the number of the upstream node of the link that opposes traffic on current link making a left turn.
- * Start-up lost time: Enter a start-up lost time (experienced by the first vehicle in a queue) in units of 1/10 seconds. The default entry is 25 to specify a value of 2.5 seconds.
- * Mean queue discharge headway: Enter a time in units of 1/10 seconds. The default entry is 22 to specify a value of 2.2 seconds.
- * Free flow speed: Enter speeds in miles per hour (10 to 65).
- * RTOR (Right turn on red) code: Enter 0 for right turn permitted, or 1 for right turn prohibited.

- * Pedestrian traffic code: Enter a code in the range of 0 (no traffic); 1 (100 to 250 pedestrians per hour); 2 (250 to 500); 3 (greater than 500 pedestrians per hour).

6.4.12 Link Turn Movements (Type 21)

Use the Turn Movements form to specify turn parameters for both simulation and traffic assignment runs.

- * Upstream node number
- * Downstream node number
- * Turn volumes (for simulation runs): Enter percents or counts for left, right, through, and diagonal. This information can be input for each time period to reflect changes over time. (Executing a traffic assignment creates this data for time period 1; practically speaking, you would probably never enter this data manually for a simulation.)
- * Turn prohibition code (for traffic assignment runs): Enter 1 to prohibit a turn for left, right, through, and diagonal. This information is input only for time period 1 to indicate where movements allowed by geometry are prohibited by policy.
- * Blockage factor: Enter the percent of the time period that one lane is blocked. This can be input for each time period of a simulation run or for time period 1 if a traffic assignment run.

6.4.13 Sign or Signal Control Data (Types 35 & 36)

The input on the Sign or Signal Control Data form is required for all internal nodes.

- * Node number: The node numbers for nodes and links have the following conventions --
 - 0001 - 1999 = internal nodes and links
 - 2000 - 2999 = internal centroid
 - 3000 - 6999 = internal nodes and links
 - 7000 - 7999 = interface nodes and links
 - (8000 - 8999 = entry or exit nodes and links) never specified here
 - 9000 - 9999 = internal nodes and links
- * Offset: Enter a number of seconds. The offset is the time gap between a network-wide reference time and the initialization of interval 1 defined for the signal at the referenced node. It provides a means of synchronizing signals in the network.

- * **X and Y coordinates:** Enter location coordinates X, Y (EW, NS) in units of centimiles (1/100 of a mile) with reference to an origin 0, 0 at the model origin (see Section 6.4.10, **Origin Control**).
- * **Interval duration:** Enter a number of seconds for signal durations during each interval. The sum of all signal durations must equal the desired cycle length (the time it takes for the signal to complete a full cycle of changes).
- * **Signal Control Codes:** Enter codes defining the signal control for each signal interval and for each of up to five different approaches to the subject node. Uncontrolled intersections should be coded for green on the major approaches and yield on the minor ones.

0 = yield sign or amber
 1 = green
 2 = red
 3 = red, green right arrow
 4 = red, green left arrow
 5 = stop sign
 6 = red, green diagonal arrow
 7 = no turns - green through arrow
 8 = red, left, and right green arrows
 9 = no left, green through and right

6.4.14 Evacuation Data (Type 49)

The parameters in the Evacuation Data form control which portions of the area around a nuclear power plant are evacuated. If this data is not present, the simulation run will produce a 100% radial evacuation without voluntary features. **This data is used only in context with a traffic simulation that will be used in conjunction with a subsequent Mesorad met/dose modeling run.** IDYNEV will ignore any form type 49 data designed to limit the evacuation area around a nuclear power plant unless the sector output code is set to 1 in form type 05 (Section 6.4.9).

- * **Mandatory evacuation factor:** A percent.
- * **Voluntary evacuation factor:** A percent.
- * **Areas to be evacuated:** Enter 1 to designate each of the following areas for evacuation. An entry of 0 designates sheltering instead of evacuation.
- * **Rings:** Designate 0 or 1 for ring areas that are identified by size of ring -- 0 to 2 miles, 2 to 5 miles, 5 to 10 miles (boundary of EPZ -- Emergency Planning Zone).
- * **Sectors:** Designate 0 or 1 under the sector number (1 to 48). Sixteen regular sectors are designated in each ring area. The sectors are numbered from the

inner ring to the outer ring, clockwise from north: the north sector in the 0-to-2-mile zone is sector number 1, and the north-northwest sector in the 5-mile-to-boundary zone is sector number 48.

These sectors are not the sectors for which sector statistics are produced by setting the **sector output code** to 1 in form type 05.

6.4.15 Entry Link Volumes (Type 50)

Use the Entry Link Volumes form to specify flow rates through an entry link when such traffic movement is sought. Entry links store vehicles until conditions at the downstream node permit vehicle discharge. No vehicle statistics are collected until the vehicles reach an internal receiving link.

Entry link volume data can be specified for each time period to reflect changes in flow rates over time. This data is omitted during traffic assignment runs or for any network that receives all its inputs from source nodes. Otherwise, the data is optional.

- * Upstream node of entry link: This node number must be in the range of 8000 to 8999.
- * Downstream node of entry link
- * Flow rate: Number of vehicles per hour. Continuous flow rate is the default.
- * Truck Volume: Percentage (of the flow rate).

6.4.16 Source/Sink Volumes (Type 51)

The Source/Sink Volumes form contains the following items to indicate vehicles entering or leaving the network at a source or sink centroid:

- * Optional source/sink centroid number: Enter a centroid number in the range of 2000 to 2999.
- * Upstream node number
- * Downstream node number
- * Source/sink rate: Enter vehicles per hour. Negative numbers indicate a sink rate.

6.4.17 Load Factors (Type 52)

The Load Factors form contains optional input data for indicating the number of people within vehicles. This form need not be used unless the default values are unsatisfactory. Do not specify more than one load factor record per time period.

The auto, carpool, and truck entries are in terms of the number of people per vehicle, multiplied by 100.

- * Average occupancy of each auto: The default is 130 (1.3 people).
- * Average occupancy of each carpool vehicle: The default is 350 (3.5 people).
- * Average occupancy of each truck: The default is 120 (1.2 people).
- * Average occupancy of each bus: Enter the number of people multiplied by 10. The default is 250 (25 people).

6.4.18 Traffic Assignment Parameters (Type 175)

Use the Traffic Assignment Parameters form for data input to traffic assignment runs. This record can appear for the first time period only. Use this form only when the default values are unsatisfactory. Changing items on this form would normally be the task of a traffic engineer with a clear understanding of the model. Refer to the IDYNEV references for explanations of the terms and values noted in this form. Blank entries will be treated as default values.

- * Exponent value defining threshold for objective function: A value in the range of 2 to 5 (default = 2).
- * Maximum number of inner iterations: A value in the range of 3 to 20 (default = 3).
- * Coefficient A in BPR formula: A positive coefficient value, multiplied by 100 (default entry = 15 to specify a value of 0.15).
- * Coefficient B in BPR formula: The coefficient value, multiplied by 10 (default entry = 40 to specify a value of 4.0).
- * Maximum number of outer iterations: A value in the range of 3 to 20 (default = 3).
- * Kalman filter applied to calculated values of capacity: A value in the range of 10 to 90 (default = 50).

6.4.19 Origin Trip Production Volumes (Type 176)

Depending on your selection (1 or the default 0) in the TRAD Model selection field in form 02, you will see one of two versions of Form 176. The Origin Trip

Production Volumes form is required in the first time period in traffic assignment runs.

Entering 1 will display this version:

- * Origin node number: Enter a node code number (2000 to 2999 or 8000 to 8999).
- * Total Volume of Traffic emitted from origin: Enter a number of vehicles per hour.
- * Candidate Destination nodes: Enter a node code number (8000 to 8999).

Entering the default value (0) will display this version:

- * Origin node number: Enter a node code number (2000 to 2999 or 8000 to 8999).
- * Destination node receiving traffic from origin: Enter a node code number (2000 to 2999 or 8000 to 8999).
- * Volume from origin to destination: Enter a number of vehicles per hour.

6.4.20 Internal Centroids (Type 177)

Use the Internal Centroids form to specify internal centroids of the network. This data is required when internal centroids are used as traffic sinks.

- * Internal centroid number: Enter a centroid number code (2000 to 2999).
- * Upstream node of link: Enter a node number for the node of the link providing access to the centroid (1 to 1999, 3000 to 6999, 9000 to 9999).
- * Downstream node of link: Enter a node number for the node of the link providing exit from the centroid (1 to 1999, 3000 to 6999, 9000 to 9999).

6.4.21 Destination Capacities (Type 178)

Use the Destination Capacities form to specify all the destinations (hosting areas) in the network. These data are required for traffic assignment runs.

- * Destination node number: Enter a node number code (8000 to 8999).
- * Maximum number of trips that may be assigned to the node: Enter a number of vehicle trips in vehicles per hour.

6.5 Graphics Mode of Modifying Input to the Evacuation Model

To use the graphics mode, type G at the prompt,

Enter record type to create/edit, menu, or graphics (#|M|G) ==>

After you enter G for graphics, the terminal display will clear and the system will present a menu of map display functions, which are described in the IBS Utilities Guide discussion of M P Display. You will now choose features (topographies) that will be displayed on the site map as you modify input to the evacuation model:

1. Use your terminal's graphics input device to pick the SELECT TOPOGRAPHIES option on the menu.

You will see a submenu that lists a set of topographies.

Note: The graphic subsystem does not support default topographies such as those used in the IBS program. Reselect your topographies from the list.

The map features listed are not necessarily available for every site. To be available for display, the features must already be a part of the site data files. (For more information on the contents of the site databases, refer to the IBS Data Management Guide).

2. Pick the map features that you want to display while modifying the evacuation model input.

Be sure to pick ROADS and POPULATION or some other map overlays containing this sort of data.

3. After selecting the desired topographies, pick EXIT.

You will return to the main menu of map display functions.

4. Pick EXIT on map display menu. The chosen topographies will appear on the map display screen, accompanied by a new graphics menu for working with the evacuation model data.

The graphics mode of modifying input to the evacuation model uses a subset of the features of the Map Display software described in the IBS Utilities Guide. More detailed information about the operating characteristics of Map Display functions can be found there.

A displayed map and the graphics input menu are shown in Figure 6.7.

Figure 6.7. Map Display with Modify Input Data Menu

— Using the Graphics Selection Menu

After you have picked topographies and exited the Map Display menu, you will see the site map and a menu for graphic modification of the IDYNEV case data (Figure 6.8). Note that the nodes in the link node diagram will only be displayed if the zoom area is small enough to make the nodes identifiable.

This part of the system uses interactive graphics to accomplish some of the same things that you would do by entering data in the evacuation model input forms. You will still need to enter data in some of the input forms, but now the system will select many of these forms automatically to complete the operations that you accomplish graphically. Some items, such as the "Modify Trip Table" item, are not represented graphically but provide convenient access to the data input form. For

some operations, you will use the graphics cursor (cross hairs) several times during the operation to carry out several steps.

Figure 6.8. Menu for Graphically Modifying IDYNEV Input Data

— Manipulating the Map

Detailed functional explanations of the map manipulation functions summarized here can be found in the IBS Utilities Guide.

The following summary descriptions assume that you have picked the menu item by placing the cross hairs over the item and tapping the <SPACE BAR>. (If you are using a mouse instead of the joydisk, press the left button on the mouse instead of the <SPACE BAR>.)

- **Exit:** You can use this option to exit back to the time step form for selecting another time step.
- **Help:** This option displays information about the menu.
- **Continue:** This option continues or finishes some data modification operations. Refer to the following discussion on modifying data.

- **Zoom in:** 1) Place the cross hairs over a point on the map that is the center of the area you want to enlarge, and tap the <SPACE BAR>. 2) Place the cross hairs over a second point that will define the edge of the enlarged area, and tap the <SPACE BAR>.
- **Zoom out:** 1) Place the cross hairs over a point on the map that is the center of the area that you want to reduce, and tap the <SPACE BAR>. 2) Place the cross hairs over a second point that will define a square to which the current map will be reduced.
- **Restore base plot:** This option restores the map to the size and detail at which it started the session.
- **Refresh screen:** This option redraws the map and menu, erasing any interfering items, such as text prompts, that can appear.
- **Coordinate Pick:** Place the cross hairs over a point for which you want to know the coordinates, and tap the <SPACE BAR>. The system will display the longitude and latitude corresponding to the point that you picked.
- **Map Zoom:** This option uses the standard IBS zoom function that will move the map so that it centers on a location of your choosing. The system will provide a menu of options on how to specify a map center. Select one of the first three options as PREDYNGR has no known points information available. The system then prompts for a new radius:

Enter the radius (in kilometers) [53.097] ==>

The units used in the prompt (miles or kilometers) are dependent on the content of your job environment. The default radius shown in the prompt is your current map zoom radius.

— Modifying Evacuation Model Input Data

Consider the following example for adding a link on a traffic network. (If you are using a mouse, press the mouse button instead of the <SPACE BAR> to point and to pick options.)

Adding a link between two existing nodes:

1. Add link on the graphics menu. (That is, move the cross hairs over the menu until they cross over the Add link menu item. Then press the <SPACE BAR> to select Add link.)

The Add link lettering should change color to indicate selection and blink to indicate that the operation is continuing.

2. Place the cross hairs over the upstream node and press the <SPACE BAR>. The upstream node will flash. You can change your mind at this point and pick another node. The new pick will start flashing, and the old pick will stop.

3. Pick Continue on the menu.

4. Place the cross hairs over the downstream node and press the <SPACE BAR>.

The downstream node will flash also. You have the same opportunity to change your mind.

5. Pick Continue on the menu.

The system will draw in the new link and automatically present the data input forms relevant to creating the link: Type 11 (Link Characteristics), Type 21 (Link Turn Movements), Types 35 and 36 (Sign or Signal Control Data).

6. Fill out the required forms, pressing <GOLD> <Z> to finish each form.

Adding a link between two nodes can affect other links having either node as a downstream node. You need to identify and modify these links, using the appropriate forms. In a similar manner, you can perform the other operations listed on the graphics menu and obtain automatic access to the data input forms of the evacuation model.

6.6 Using the IDYNEV Function Selection Menu

This menu is only applicable to IBSSH users who selected IDYNEV from the Models menu. Users of the IBS program can perform all of the following functions, as well as other CSEPP functions involving IDYNEV, by selecting them from the IBS Emergency Activities menus. For further information, see the *IBS User Guide*.

When you select IDYNEV from the Models, you will be turned over to the IDYNEV Function Selection menu. Sections 6.6.1 through 6.6.8 will explain the

Dynamic Evacuation Model

various functions that can be requested from the IDYNEV function selection menus shown in Figures 6.9 and 6.10.

IDYNEV FUNCTION SELECTION	
Selection Designator	Selection
A	Execute Evacuation - Interactive
B	Execute Evacuation - Batch
C	Print/Display Reports
D	Change Case/Site
E	Perform File Maintenance
X	Exit

Please Enter Selection Designator.

Figure 6.9. IDYNEV Function Selection Menu - Text Mode

EXIT
HELP
CHANGE CASE/SITE
EXECUTE EVAC. INT.
EXECUTE EVAC. BATCH
DISPLAY RESULTS ON MAP
DISPLAY BASE MAP
PRINT/DISPLAY REPORTS
PERFORM FILE MAINTENANCE

Figure 6.10. IDYNEV Functional Selection Menu - Graphic Mode

6.6.1 Execute Evacuation - Interactive

IDYNEV Function Selection Text Menu: Select A (Execute Evacuation - Interactive)

This option displays a check list for the current evacuation case (Figure 6.11) along with a prompt: **Execute IDYNEV for this case? (Y/N) [N] == >.**

Model Check List. The check list is displayed for your information only: it is NOT a form that you can change. Read the check list to be sure that the case is set up to run as you wish. Then respond to the prompt:

- If you answer N (no), you will return to the IDYNEV function selection menu without running the model.
- If you answer Y (yes), the model will execute in interactive mode. In this mode, the model is run as part of your current process, and while the model is running, you cannot interact with any other IBS function. The system will return to the menu when the model has completed its run.

You can display the model results graphically by selecting the Display Results On Map function (see Section 6.6.3).

Stopping the Model. The evacuation model cannot be cleanly halted while it is running. Pressing <CTRL Y> is a standard system interrupt command and will halt model execution, but presents no guarantee where you will regain control. The model can be restarted only from the beginning of its execution.

Simulation Summary Report. If the model is executed for a traffic simulation, and if one or more time steps of the simulation are completed, the system displays a report of the evacuation following model completion. This report summarizes various statistics extracted from the last simulation time period reported by the evacuation model: vehicle miles, vehicle minutes, vehicle trips, average speed, percentage of vehicles that stopped, average number of vehicles per queue, average delay per vehicle, etc.

The current IDYNEV case has the following attributes:

CASE: 700
Type: SIMULATION
Site: JCB1
Name: My development site.

Selection Statistics WILL be generated.

Start time: 22:00
periods: 10
Time span: 0:03:00

Time period breakdown:

150	150	150	150	150	150	150	150	150	450
0	0	0	0	0	0	0	0	0	

Execute IDYNEV for this case: (Y/N) [N]==>

Figure 6.11. Check List for IDYNEV Model Execution (Example)

6.6.2 Execute Evacuation - Batch

IDYNEV Function Selection Text Menu: Select B (Execute Evacuation - Batch)

This option displays the check list (Figure 6.11) for the current evacuation case and displays the prompt: **Execute IDYNEV for this case? (Y/N) [N] == >**

Read the check list to be sure that the case is set up to run as you wish. Then respond to the prompt at the bottom of the check list:

- If you answer N (no), you will return to the IDYNEV function selection menu without running the model.
- If you answer Y (yes), the model will execute in batch mode. Batch execution of the model is convenient for larger cases. In batch mode, the system executes the model separate from your current interactions. While the model is running in batch mode, you will return to the IDYNEV function selection menu. You can then use other system functions, but do NOT select any IDYNEV interactions while the model is running: these could interfere with the model run.

After the evacuation model has finished executing, you can display the model results graphically by selecting the Display Results On Map function.

6.6.3 Display Results On Map - Graphics Mode Only

Pick this option on the IDYNEV graphics menu to graphically display the results of the current evacuation case.

When you first pick DISPLAY RESULTS ON MAP, you will see a menu of Map Display functions, which are fully described in the IBS Utilities Guide. The main purpose of this menu here is to enable you to choose background topographic features for the site map:

1. Use the SELECT TOPOGRAPHIES option to pick topographies for inclusion in the current display.

Each menu item will change color as you pick it. Be sure to include ROADS among the selected features.

2. Exit from the topography selection and Map Display menus.

The system will draw the site and present the graphics menu shown in Figure 6.12.

3. Use the graphics input device attached to your workstation (the joydisk on your keyboard, or a mouse):

To pick a menu item: Use the joydisk (or mouse) to place the graphics cursor cross hairs (or cross) over the menu item and tap the keyboard <SPACE BAR>.

The item will change color to indicate selection. Some items will blink, indicating that further interaction is needed to continue the operation.

Site Model Case #	
EXIT	<-- Site and case number identification
HELP	
CONTINUE	
DESCRIBE NODE	
DESCRIBE LINK	
DESCRIBE EXIT NODE	
DESCRIBE INT. CENTROID	
DISPLAY FLOOD WATER	
RETURN POPULATION	
DETERMINE AREA	
ZOOM IN	
ZOOM OUT	
RESTORE PLOT	
RUN	
RESTART	
STEP FORWARD	
STEP BACKWARD	
SELECT TIME STEP	
## HOURS, ## MINUTES	<-- Time Period label appears when model results are run or stepped.
TEXT DISPLAY ENABLED	
ICON DISPLAY ENABLED	
REFRESH SCREEN	

Figure 6.12. Graphics Menu for Display Results on Map

The following descriptions explain the functions of the graphics menu terms (Figure 6.12). Some functions can require pressing the <DIALOG> key to view text information.

- **EXIT:** This option exits the Display Evacuation Results menu.
- **HELP:** This option displays information about the menu.
- **CONTINUE:** This item is used to continue or finish another operation.
- **DESCRIBE NODE:** Use the graphics cursor to pick a node on the traffic network (the selected node will blink) and a report will be displayed. This option will display signal interval durations and node numbers of approaching links.
- **DESCRIBE LINK:** Pick a link on the network (the selected link will blink); then pick CONTINUE.

This option will display link information, including Level of Service (LOS) codes that indicate traffic conditions from free-flow (code A) to unmoving (code F). The link information is part of a set of dynamic statistics that are

accumulated as the model steps through its execution. To display link information, the model output must first be stepped at least through the first time step. (See the RUN, STEP FORWARD, and STEP BACKWARD functions in following paragraphs.)

- **DESCRIBE EXIT NODE:** Pick an exit node on the evacuation boundary (the selected node will blink); pick CONTINUE.

Exit nodes are just described by an associated internal node number.

- **DESCRIBE INTERNAL CENTROID:** Pick an internal centroid that points into a link of the network (the selected centroid will blink), and a report will be displayed.

This option displays a report on the centroid, including the time period, centroid number, and traffic flow rate in vehicles per hour.

- **DISPLAY FLOOD WATER:** This option will display flood water contours if you have SLOSH flood data available.
- **RETURN POPULATION:** This option will display the population associated with a selected area -- 1) Outline an area on the screen by picking points that define the sides of a polygon around the desired region. 2) Pick CONTINUE to draw the final side of the polygon and return a population value.
- **DETERMINE AREA:** This option will display the area within a polygon that you draw on the map display -- 1) Outline an area on the screen by picking points that define the sides of a polygon around the desired region. 2) Pick CONTINUE to draw the final side of the polygon and return area and parameter values.
- **ZOOM IN:** 1) Pick a point at the center of an area where you want to zoom in (enlarge). 2) Pick a second point that defines the edge of a square with the first point at the square's center.

The square area defined will be rescaled to fill the map display grid.

- **ZOOM OUT:** 1) Pick a point at the center of an area where you want to zoom out (scale down). 2) Pick a second point that defines the edge of the square with the first point at the square's center.

The display area will be scaled down to the size of the defined square with the display centered at the first point chosen.

- **RESTORE PLOT:** This option restores the map display to its initial scale and appearance.

- **RUN**: This option displays the model output, showing changes in the traffic network as the output steps from the first time step to the last.

Red bars on the network links (street segments) indicate the extent of traffic line ups near nodes (intersections). These bars have a constant width, no matter what the scale of the map display.
- **RESTART**: This option resets the model output at the first time step.
- **STEP FORWARD**: This option displays the traffic network results for the next sequential time step.
- **STEP BACKWARD**: This option displays the traffic network results for the previous time step.
- **SELECT TIME STEP**: Type a time step number in response to the prompt, and press <RETURN>.

This option enables you to reset the model time step for displaying results.

- **TEXT DISPLAY ENABLED**: This option enables/disables the display of map-related text if any such text is associated with the map data.
- **ICON DISPLAY ENABLED**: This option enables/disables the display of map icons (constant-size graphic figures that represent the location of items on the map).
- **REFRESH SCREEN**: This option clears and then redraws the map display.

6.6.4 Display Base Map - Graphics Mode Only

This graphics menu option produces the standard menu of Map Display functions. For a complete description of these functions, refer to Map Display in the IBS Utilities Guide.

6.6.5 Print/Display Reports

IDYNEV Function Selection Text Menu: Select C (Print/Display Reports)

This option displays several options for printing or displaying the IDYNEV evacuation report, as shown in Figure 6.13. This large report on the modeled evacuation is the only report for the evacuation model. The output is designed for a high-speed line printer, so you may need to set your terminal screen for a greater width.


```

=====
P. Print file on printer
D. Display file on terminal
X. Exit
=====
Choice ==> :

```

Figure 6.13. Menu for IDYNEV Print/Display Reports Option

6.6.6 Change Case/Site

IDYNEV Function Selection Text Menu: Select D (Print/Display Reports)

Selecting this function from the IDYNEV Function Selection menu will display the site/case selection form (example shown in Figure 6.14). The Site Name in the form refers to the current site. The first of the two case numbers shown is the number of the model's Current Case. The second, or Based on, case number is the case from which the current case was originally created.

Use the up, down, left, and right arrow keys (<F1> - <F4>) to move the input cursor to the field you want to change. When you are finished changing fields, press <GOLD> <Z> to accept the changes and exit the form (or <GOLD> <Y> to exit, restoring the original contents of the fields).

If you change Site Name, you may wish to consider changing the current elevation and population data sets as well. (A message screen will remind you of this and provide information about the defaults that are applied by the system when you change sites.)

IDYNEV - Evacuation Model

Case Selection Form V2000

Site Name: MY DEVELOPMENT SITE

IDYNEV Model Case: 700

IDYNEV Model Base: 700

Figure 6.14. Form for Change Case/Site (Example)

6.6.7 Perform File Maintenance

IDYNEV Function Selection Text Menu: Select E (Perform File Maintenance)

Selection of this function from the IDYNEV Function Selection menu will display the menu shown in Figure 6.15. The menu options enable you to list the directories of evacuation cases for the current site. You also can delete selected input and output files from the system (if you own the files). Exercise care here, especially in deleting input files, because re-creation of files can be time consuming.

PERFORM FILE MAINTENANCE	
Selection Designator	Selection
A	Display Directory of Cases
B	Delete Case Output for a Specified Case
C	Delete Input and Output for a Specified Case
X	Exit to Previous Menu
Enter selection designator:	

Figure 6.15. Menu for Perform File Maintenance

6.6.8 Exit

Selection of this function from the IDYNEV functional selection menu will return you to the subsystem from which IDYNEV was selected.

6.7 If You Have Problems

The IBS is designed to provide as much help to the user as possible in the form of data range checking, and status and diagnostic messages to make the system's use independent of computer systems specialists. If you have questions, try the following:

1. Obtain on-line help with data forms and fields, press the HELP (<F6>) key on your terminal keypad.
2. Contact the System Manager for assistance with problems related to the operation of the system.

3. Refer to your equipment service contract for the name of your vendor representative, if problems seem to be related to hardware.

6.8 Status and Diagnostic Messages

The system provides a number of status messages that keep you informed about system activity. These messages generally do not require any reciprocal action on your part. They will be generated automatically under certain circumstances, such as the start of a process which requires substantial computing time and which does not involve user interaction.

The following diagnostic messages will appear under certain circumstances. Associated actions are recommended where practical.

- YOU MUST ENTER 3 DIGITS TO SPECIFY THE EVACUATION CASE TO DELETE. FOR EXAMPLE "001" TO SPECIFY CASE 1. THIS IS FOR YOUR PROTECTION.
- YOU MUST USE DIGITS ONLY TO SPECIFY THE EVACUATION CASE TO DELETE. FOR EXAMPLE "001" NOT "C1".
- THE EVACUATION CASE NUMBER THAT YOU SPECIFIED FOR A BASE CASE DOES NOT EXIST.
- PLEASE TRY AGAIN.
- I'm sorry, but with the current design of IBS, you'll have to wait until it completes before you can start another run.
- AN UNANTICIPATED ERROR HAS BEEN ENCOUNTERED WITH THE EVACUATION BASE CASE - PLEASE REPORT TO YOUR SYSTEM MANAGER.
- I'm sorry, but you have requested that IBS draw an evacuation link-node network diagram, and there doesn't seem to be any evacuation model input data for the evacuation model case and site currently selected.
- The link-node diagram will be excluded from the background map.
- I'm sorry, but you have requested that IBS draw a background site map, and there doesn't seem to be any background map data for the site.
- The background map will not be drawn.

Section 7

Model Library - Outdoor Sound Propagation Model

IBSSH Menu: Select Models

Models Menu: Select OSPM (Outdoor Sound Propagation Model)

This section describes the **Outdoor Sound Propagation Model (OSPM)**, a computer model that helps in evaluating existing and planned placements of warning sirens that are used in the vicinity of nuclear power plants or other industrial sites. This section covers the following topics:

- 7.1 Description of the Outdoor Sound Propagation Model
- 7.2 What You Need to Know to Use the Outdoor Sound Propagation Model
- 7.3 Using the OSPM Function Selection Menu
- 7.4 Change Case
- 7.5 Display Base Map
- 7.6 Contour Results
- 7.7 Report Decibel (dB)
- 7.8 Edit Elevation File
- 7.9 Update Impedance
- 7.10 Place Site
- 7.11 Update Siren
- 7.12 List Site Specs
- 7.13 Execute OSPM - Interactive
- 7.14 Execute OSPM - Batch
- 7.15 Perform File Maintenance
- 7.16 Delete Contours

Model Library - Outdoor Sound Propagation Model

- 7.17 Create a New Siren Type
- 7.18 Change Site

7.1 Description of the Outdoor Sound Propagation Model

In the past, alert sirens were modeled individually and manually to determine the overall sound output levels at locations of interest. The OSPM simultaneously models many sirens in a given geographic area and reports sound pressure output levels at dBa, dBC, and 8 octave-band frequencies (63 Hz through 8 kHz) for a gridded set of points around the center of the area of interest. The OSPM was prepared by International Energy Associates, Ltd., now ERC International, Washington, D.C.

The OSPM is used by FEMA to evaluate existing and planned configurations of sirens that are used to alert the populace in the vicinity of nuclear power plants. Within FEMA, OSPM is the accepted model for the evaluation of alert systems. The OSPM subsystem implemented in IBS includes input data preparation, data display, and file management of relevant information for the model.

Data Preparation. The OSPM subsystem enables input to the model through various data input forms and through graphic manipulations as well. For example, the locations of sirens are stored in an IBS map graphics file. Sirens can be added, deleted, and updated by picking siren icons on the graphics display. A set of impedance (acoustic attenuation) data is maintained in an IBS gridded cell file, where cell values can be accessed by picking points on the site map and modified.

Data Display. The data display features gained by integrating this model into the IBS include 1) the ability to graphically visualize the sound pressure levels in the form of line contours (isopleths of sound pressure level) that are drawn over background maps and 2) the ability to determine the population count in those associated areas to get an idea of the effectiveness of the siren sound spread in alerting the necessary population. For example, it may be that in certain areas the sound does not carry well, but if the area is unpopulated, effective sound propagation would not be very important.

Data File Management. The data file management aspect of OSPM integration is quite similar to other models integrated into IBS; that is, you have the ability to specify the site of interest and the current cases (input and output data sets) using a case and site specification forms. You have the ability to perform file maintenance, which consists of taking a directory of cases in the OSPM, including a case description that you can provide to describe the overall purpose of a particular case.

Use of Elevation Data. Another significant feature of the OSPM integration into IBS is that elevation data, which is a key input set to the OSPM model, is provided through the Geographic Information System (GIS) functionality underlying IBS. That is, once you have obtained an appropriate elevation data set for the site, you do not have to prepare individual elevation input values for a large number of locations in order to run the model: that is done for you automatically.

7.2 What You Need to Know to Use the OSPM

With the information supplied in this section, you will be able to execute the OSPM and obtain model results. To use the OSPM in IBS, just:

1. Modify the input data for the current OSPM case.
2. Execute the model (Sections 7.13 and 7.14).
3. Graphically display the results (Section 7.6).

The following overview of the model can be helpful in understanding the model inputs.

7.2.1 Overview of the Model

The model calculates the siren sound level at various receiver or listener locations (referred to as **receptors**) caused by the operation of a single siren under a single set of meteorological conditions. The siren sound level depends upon three basic factors: 1) the sound output power of the siren; 2) the topography of the siren location; and 3) the meteorological conditions. OSPM requires, as input, detailed information about each of these factors.

7.2.2 Obtaining and Using Elevation Data

Application of OSPM requires a special group of TAC data sets for each IBS site. (TAC = Terrain Analysis Cell, the cell file format used by IBS to store regular gridded data, chiefly elevation data. A TAC data set consists of a .TAC file of header information and a .TAA file of cell values.) These files are stored in the top-level site directory of each site: `SITE$DIR disk:[username.SITES.site]` where *site* is the four-character site code. OSPM requires the following files, which have a common *filename* root:

- *filename*.TAC and *filename*.TAA -- These are the standard IBS elevation data files for a site, and *filename* is the name of the Current Elevation Set specified for the current IBS **job environment**. The elevations are for longitude-latitude locations.
- *filename*_UTM.TAC and *filename*_UTM.TAA -- These are the elevation data files used directly by the model. They contain Universal Transverse Mercator (UTM) projected data converted from the longitude - latitude locations in the *filename*.TAC and *filename*.TAA files.

- *filenameAT.TAC* and *filenameAT.TAA* -- These are cell files containing OSPM impedance (acoustic attenuation) information that indicates the acoustical hardness of the terrain at each cell location. These files are specifically created to contain this information.

The standard TAC elevation data set can be created from the regional map database (if the database contains elevation data for the site area). This procedure is explained in the IBS Utilities Guide, Section 2.5.7, **MPDISPLAY STORE REGION**.

The other two TAC data sets are specific to OSPM. They are created by using two special TAC utility programs, **LATLONXUTM** and **CREATETAC**. **LATLONXUTM** converts a TAC data set in longitude-latitude coordinates to a TAC data set having a UTM projection. **CREATETAC** just creates a TAC data set having a specified size and uniform cell values.

Creating the UTM files for the Model. You will use the standard longitude-latitude elevation data as input to the **LATLONXUTM** utility for creating UTM elevation data:

1. Outside of IBS, set your default directory to the site directory and start **LATLONXUTM**:

```
$ SET DEF SITE$DIR
```

(If this does not place you in the proper site directory, use **IFS** to set to the correct site and try again.)

```
$ LATLONXUTM
```

2. Respond to the following series of prompts as shown in the following example. Note that the prompts show default responses in brackets []; pressing <RETURN> accepts the default value. For this example, the *filename* of the standard elevation data set is assumed to be **ABCDEL** (consisting of **ABCDEL.TAC** and **ABCD.TAA**).

By convention, the standard name of any site's elevation file is **siteEL**, where **site** is the four-character site code. Similarly, the UTM copy of the site elevation file is **siteEL_UTM** and attenuation data file is **siteELAT**.

<u>LATLONXUTM Prompt</u>	<u>Your Response</u>
Enter Input File (.TAC):	ABCDEL.
Enter Output File (.TAC):	ABCDEL_UTM
Do you wish to convert Lat-Long TAC file to UTM [y]?	y or <RETURN>
Enter Min X of Output Map [value from input file]:	<RETURN>
Enter Max X of Output Map [value from input file]:	<RETURN>
Enter Min Y of Output Map [value from input file]:	<RETURN>
Enter Max Y of Output Map [value from input file]:	<RETURN>
Pad type (0=none, 1=User supplied, 2=Nearest Neigh) [2]:	2 or <RETURN>

Model Library - Outdoor Sound Propagation Model

3. Check the site directory to see that the UTM elevation files have been created.

For the example above, these would be ABCDEL_UTM.TAC and ABCDEL_UTM.TAA.

Creating the Impedance TAC Files. The initial goal of creating the OSPM impedance files is to generate a cell data file that has the same extents and grid size as the regular elevation data file but which contains zero (0) values instead of elevation values. Impedance values are then entered using the UPDATE IMPEDANCE function of the IBS OSPM subsystem (explained in Section 7.9).

1. Determine the longitude-latitude extents and the cell grid size (rows x columns) of the standard elevation data set.

You can do this by using the DISPLAY ELEVATION STATISTICS option of the IBS Map Display software. This option is available through the EDIT ELEVATION FILE function on the OSPM menu (or on the Miscellaneous Functions menu of the IBS Map Edit software).

2. Outside of IBS, set your default directory to the site directory and start CREATETAC:

```
$ SET DEF SITE$DIR or disk:[username.SITES.site]
$ CREATETAC
```

where *disk* is the device name, *username* is your username, and *site* is the IBS four-character site code.

3. Respond to the CREATETAC prompt for information as shown in the following example. For this example, the name of the standard elevation data set is assumed to be ABCDEL (which would consist of ABCDELAT.TAC and ABCDELAT.TAA).

<u>CREATETAC Prompt</u>	<u>Your Response</u>
ENTER NAME OF TAC FILE :	ABCDELAT
ENTER MINIMUM LONGITUDE :	] Enter the longitude-latitude extents and grid size that you found for the standard elevation data set in Step 1.
ENTER MAXIMUM LONGITUDE :	
ENTER MINIMUM LATITUDE :	
ENTER MAXIMUM LATITUDE :	
ENTER NUMBER OF COLUMNS :	
ENTER NUMBER OF ROWS :	
ENTER CELL VALUE :	0 (sets all cell values to zero)
ENTER SIZE OF DATA TYPE :	1 (selects byte as the data type)

4. Enter DIR to check the site directory and see that the attenuation (impedance) files have been created.

For the example, these would be ABCDELAT.TAC and ABCDELAT.TAA.

Following the previously described convention for naming site elevation and other site TAC files ensures that they can be found by the system when needed.

7.3 Using the OSPM Function Selection Menu

Models Menu: Select OSPM (Outdoor Sound Propagation Model)

Depending on your selection of Models Graphic or Models Non-Graphic, you will see one of the OSPM function selection menus shown in Figures 7.1 and 7.2.

Using the Graphic Menu. The graphic menu contains all IBS OSPM functions, including the graphics-specific functions, such as displaying the base map and drawing contours of sound pressure level.

Note: The first time that you pick any graphics display function, IBS automatically drives you through the DISPLAY BASE MAP function before continuing: this is mostly to give you the opportunity to select background topographies for the map display. Remember that OSPM does not support default topographies. These must be selected each time OSPM is accessed.

EXIT	
HELP	
CANCEL	
CONTINUE	
CHANGE CASE	7.4
DISPLAY BASE MAP	7.5
CONTOUR RESULTS	7.6
REPORT DECIBEL (dB)	7.7
EDIT ELEVATION FILE	7.8
UPDATE IMPEDANCE	7.9
PLACE SIREN	7.10
UPDATE SIREN	7.11
LISTS SITE SPECS	7.12
EXECUTE OSPM - INTER	7.13
EXECUTE OSPM - BATCH	7.14
PERFORM FILE MAINTENANCE	7.15
DELETE CONTOURS	7.16
CHANGE SITE	7.18

Figure 7.1. OSPM Top-Level Menu - Graphic Mode

Non-Graphic OSPM Main Menu

```
0 : EXIT
1 : List/Update Site Specs
2 : Execute OSPM - Interactive
3 : Execute OSPM - Batch
4 : Change Site
5 : Change Case
6 : Perform File Maintenance
7 : Create New Siren Type
    Menu Choice (? = Help) == >
```

Figure 7.2. OSPM Function Selection Menu - Text Mode

Using the Text Menu. The text menu contains everything except the graphics specific functions. To start an OSPM function from the text menu, type the corresponding number and press <RETURN>.

The OSPM functions are described in Sections 7.4 through 7.18, in the order that they appear on the graphics menu. The graphics specific functions are marked Graphic Menu Only. The option to create a new siren type occurs as option 7 on the text menu; this function is part of PLACE SIREN on the graphic menu.

Exiting from the OSPM Function Selection Menu(s). When you exit the OSPM subsystem, the system enables you to choose whether you want to save the changes and updates that you have made:

Save the OSPM updates? (Y/N) [Y] == >

- If you answer N (no), the system will ignore any changes that you have made during the update session.
- If you answer Y (yes), the system will save all the new data files that were created by your updating activities.

7.4 Change Case

OSPM Text Menu: Select 5 (Change Case)

When you pick CHANGE CASE from the OSPM menu, the system displays the form shown in Figure 7.3 used for switching to a different OSPM case number. The site name in the form refers to the current site.

```
OSPM - OUTDOOR SOUND PROPAGATION MODEL      Case Selection Form V2.00

Site Name: _____

OSPM      Model Case:  _____
OSPM      Model Base:  _____
```

Figure 7.3. Change Case/Site Form

Use the arrow keys and <RETURN> to edit the data in the following fields. When finished, press <GOLD> <Z> to accept the data and exit the form.

Site Name: A four-character site code corresponding to a site that has been created for IBS (for reference only).

- * OSPM Model Case: Enter a three-digit case number from 000 to 999. Always enter all three digits of the case number into the case number field. This is the current case number (the one to be worked with).

When you use OSPM for the first time, this case number will be 000, the number of the standard OSPM **base case**. Case 000 has no particular significance except that it contains an executable set of data inputs. If you enter a case number for which no input data exists, the system will create the new case based on the case listed in the Based on Case: field.

- * OSPM Model Base: If you want to create a new case, enter the number of an existing case that can be used as the basis for creating another OSPM case. Always enter all three digits into the field.

Initialization of Graphics Functions. When you change the case or site, the system takes a moment to re-initialize all aspects of the graphics data. You will need to reselect topographies, icons, etc. for the current site or case.

7.5 Display Base Map - Graphic Menu Only

This option produces a menu of standard graphics functions for IBS map display. For a complete description of these functions, refer to the MPDISPLAY utility in the *IBS Utilities Guide*.

Siren Icon Locations in the Graphics Edit File. If you are familiar with using the IBS Map Display software, then you know that it maintains an **edit file** for storing any graphic elements added when you use the EDIT FUNCTIONS during map display. The OSPM subsystem makes special use of this edit file as part of the input data set for the OSPM system: For each OSPM case, the system maintains an edit file that contains the locations of graphic icons that represent the sirens on the site.

These case-specific edit files are located in the OSPM input directory for the site:

disk:[username.SITES.site.INPUT.OSPM]CnnnOSPM.DMS

where *site* is the four-character site code and *nnn* is the OSPM case number.

This approach makes it possible to examine the attributes of these sirens by picking the icons in the DISPLAY BASE MAP function. This also means that you could use the EDIT FUNCTIONS to edit the siren location file, but any editing of the siren icons could cause a mismatch between this file and a different data file that contains further siren information. Siren location editing should be done only through the OSPM functions on the OSPM graphic menu.

Note: If you use the EDIT FUNCTIONS on the Map Display menu, you will be editing the graphics file that contains the siren icon locations for the current case. Although you might add unrelated items, **DO NOT EDIT THE SIREN LOCATIONS** in this way: This could cause discontinuities in the OSPM case data.

7.6 Contour Results - Graphic Menu Only

This option enables you to graphically visualize the sound pressure levels of a modeled siren by drawing line contours on a site map background. The contours represent isopleths of the sound pressure level at a specific frequency. Contouring requires an OSPM output file, which is obtained by executing OSPM (in either interactive or batch mode). To define the contour display, you must specify a frequency and set of contour values (sound pressure levels) for the contours.

Specifying a Frequency. When you pick CONTOUR RESULTS from the OSPM menu, the system first displays a menu for selecting the sound frequency that you wish to contour (Figure 7.4). Choose one of the frequency values.

Specifying Contour Values. After you choose a frequency, the system displays a form for specifying the sound pressure value associated with each contour (Figure 7.5). Beginning with Contour 1 (the lowest level), enter values for a number of contours, and then press <GOLD> <Z> to finish. The contour value fields can contain values that have been used in a previous contour display.

Displaying the Contours. The system then attempts to create contours on the graphics screen for the selected frequency and sound pressure levels. If the results are lower than any of the specified sound pressure levels, no contours will be drawn. When you are through viewing the contours, delete them with the DELETE CONTOURS option.

CONTOUR FREQUENCY SELECTION	
Frequency Designator	Frequency
A	dBa
B	dBc
C	63 Hz
D	125 Hz
E	250 Hz
F	500 Hz
G	1 kHz
H	2 kHz
I	4 kHz
J	8 kHz
X	Exit

Enter Frequency Designator Corresponding to Frequency You Wish to Contour:

Figure 7.4. Contour Frequency Selection Menu

CONTOUR SPECIFICATIONS	
TITLE: OSPM CONTOUR CONTROL	
Contour Number	Contour Value (Units)
1	0.2000E+02
2	0.4000E+02
3	0.6000E+02
4	0.8000E+02
5	0.1000E+03
6	0.0000E+00
7	0.0000E+00
8	0.0000E+00
9	0.0000E+00
10	0.0000E+00

Figure 7.5. Contour Specification Form (Example)

Contour Graphics Files. The graphic data represented by the contours are stored in case-specific Digital Mapping Structure (DMS) files in the OSPM output directories:

disk:[username.SITES.site.OUTPUT.OSPM]CnnnCONTOUR.DMS

where *site* is the four-character site code and *nnn* is the OSPM case number. These contour graphics files are standard DMS files and can be examined or edited with the IBS Map Display and Map Edit programs.

Note: Do NOT attempt to edit contours using the EDIT FUNCTIONS under the BASE MAP DISPLAY option of the OSPM menu. If you do so, the contours will be written to the current edit file, which is intended to contain the location information for displaying **siren icons**. If you wish to edit the contour graphics files, do so with IBS map editing functions outside of the OSPM subsystem.

7.7 Report Decibel (dB) - Graphic Menu Only

Use this function to obtain a report of sound pressure levels calculated at a point selected by a graphic pick. The report includes all 10 of the OSPM output types: each of the 8 octave bands, the dBa, and the dBc. Figure 7.6 shows an example report.

Point data reporting requires OSPM output, which is obtained by executing OSPM (in either interactive or batch mode).

To obtain a report at a point:

1. Pick REPORT DECIBEL (db).
2. Pick a point on the map within the modeled area.

The system will display the sound level report similar to the following one on the dialogue screen for a moment. Pick any point in the graphic menu area to exit from Report Decibel and restore graphic menu.

Sound Levels for location Lon,Lat		-69.586	44.008
89.00	DBA		
91.70	DBC		
0.373E-4	64Hz		
0.259E-4	128Hz		
0.932E-5	256Hz		
91.30	512Hz		
81.60	1 KHz		
67.80	2 KHz		
47.50	4 KHz		
0.000E=00	8 KHz		

Figure 7.6. Example Sound Level Report from REPORT DECIBEL (db)

7.8 Edit Elevation File - Graphic Menu Only

This function enables you to modify the elevation file used directly by OSPM to provide an elevation profile for the current site. When editing the elevation file, you are making changes to a grid of elevation points from which the model selects certain points to obtain the elevation profile. (The elevation file is *disk:[username.SITESite]siteEL UTM.TAC* in the site directory, where "site" is the IBS site code -- see Section 7.2.2.)

After choosing this function, you will see a submenu of elevation-related functions. These functions are explained under Map Edit in the *IBS Utilities Guide*. (Almost all of the functions are the same or similar to the functions of the UPDATE IMPEDANCE menu explained in Section 7.9. In that section, the same functions are applied to impedance cell values instead of elevation cell values.)

A rectangular box containing a list of menu options for editing elevation files. The options are arranged in several groups: a top group with EXIT, QUIT, HELP, CANCEL, and CONTINUE (where EXIT and QUIT have sub-explanations in parentheses); a group with MODIFY ELEVATION and MODIFY GROUPED ELEVATION; a group with RETURN ELEVATION, RETURN SLOPE, ENABLE ELEVATION RES, and DISABLE ELEVATION RES; a group with DISPLAY ELEVATION STATS and DISPLAY ATTRIBUTES; a group with ZOOM IN, ZOOM OUT, and RESTORE BASE AREA; and a bottom group with REFRESH SCREEN and CONTOUR ELEVATION. The text is left-aligned within the box.

```
EXIT (SAVE TAC FILE CHANGES)
QUIT (DEL TAC FILE CHANGES)
HELP
CANCEL
CONTINUE

MODIFY ELEVATION
MODIFY GROUPED ELEVATION

RETURN ELEVATION
RETURN SLOPE
ENABLE ELEVATION RES
DISABLE ELEVATION RES

DISPLAY ELEVATION STATS
DISPLAY ATTRIBUTES

ZOOM IN
ZOOM OUT
RESTORE BASE AREA

REFRESH SCREEN
CONTOUR ELEVATION
```

Figure 7.7. Graphic Menu for EDIT ELEVATION FILE

7.9 Update Impedance - Graphic Menu Only

Selecting this function displays a submenu (Figure 7.8) that enables you to modify the data file containing impedance (acoustic attenuation) data for the current site. The file contains gridded data; the value of each cell (location) in the grid indicates whether that location is acoustically hard or soft. Upon completion of your modifications, IEMIS interprets the file and provides OSPM with the coordinates of acoustically hard locations.

Impedance Data Files. Unlike regular gridded elevation data that are probably available from the regional map database, the impedance data are needed only by OSPM and must be generated outside of IBSSH. See Section 7.2.2.

Impedance Functions. The principal model-related functions of the UPDATE IMPEDANCE submenu shown in Figure 7.8 are described in the following paragraph. Standard Map Display functions (groups at the top and bottom of the menu) are explained in detail under MPDISPLAY in the *IBS Utilities Guide*.

- EXIT (SAVE TAC FILE CHANGES): Returns to the OSPM menu, saving a new version of the impedance file. This new version will include any changes that you make to the impedance data.
- QUIT (DEL TAC FILE CHANGES): Returns to the OSPM menu but does NOT save changes and does NOT create a new version of the impedance file.

A rectangular box containing a list of menu options for the UPDATE IMPEDANCE function. The options are arranged in several groups, with some groups separated by blank lines. The text is in all caps and a monospaced font.

```
EXIT (SAVE TAC FILE CHANGES
QUIT (DEL  TAC FILE CHANGES
HELP
CANCEL
CONTINUE

MODIFY IMPEDANCE
MODIFY GROUPED IMPEDANCE

RETURN IMPEDANCE

ENABLE IMPEDANCE RES
DISABLE IMPEDANCE RES

DISPLAY IMPEDANCE STATS
DISPLAY ATTRIBUTES

ZOOM IN
ZOOM OUT
RESTORE BASE AREA
REFRESH SCREEN
```

Figure 7.8. Graphic Menu for UPDATE IMPEDANCE

- **MODIFY IMPEDANCE**: Enables you to find and modify an impedance data value.

1. Pick MODIFY IMPEDANCE.
2. Pick a point on the screen.

A yellow asterisk (*) will mark the selected location. Depending on the screen density of the impedance data, a box representing the borders of the impedance data cell can also appear.

3. Pick CONTINUE.

The dialogue screen will display the current impedance cell value and a prompt asking you to enter a new impedance value. 0: = soft
1: = hard

4. Enter a new value (or press <CTRL Z> to skip entering a new value).

- **MODIFY GROUPED IMPEDANCE**: Displays the following submenu (Figure 7.9), for modifying all cell values within a specified polygon area or within the current screen area:

EXIT
HELP
CANCEL
CONTINUE
CURRENT SCREEN
DRAW POLYGON
SELECT POLYGON
EDIT POLYGON CELL VALUES

Figure 7.9. Menu for MODIFY GROUPED IMPEDANCE

The principal options of this submenu are explained as follows.

- **CURRENT SCREEN**: Selects the currently displayed screen area as a polygon within which you can change impedance cell values. The menu item will blink in green as long as the current screen is the selected polygon. (See EDIT POLYGON CELL VALUES.)
- **DRAW POLYGON**: Enables you to define a polygon within which you can change elevation values.

1. Pick DRAW POLYGON.

2. Pick three or more points that define the sides of a polygon.

The program will draw segments between each new point and the previous point.

3. Pick CONTINUE to complete the polygon.

The program will draw the final segment between the first and last points picked. The polygon will blink in green to indicate the polygon is selected. The polygon can then be used for EDIT POLYGON CELL VALUES.

- **SELECT POLYGON**: Enables you to pick an existing polygon within which you can change impedance values.

1. Pick SELECT POLYGON.

The menu item will blink in green to indicate that further input is required.

2. Pick any point on a polygon displayed on the screen.

The polygon will blink in white to indicate a pick. If the wrong polygon is highlighted, pick again on the desired polygon until it is highlighted.

3. Pick CONTINUE to confirm the polygon selection.

The polygon will blink in green to indicate that the polygon was properly selected. The menu option will stop blinking to indicate that no further input is required for polygon selection. The polygon can then be used with EDIT POLYGON CELL VALUES.

- **EDIT POLYGON CELL VALUES**: Enables you to enter one impedance value for all cells within the currently selected polygon (which can be the entire screen area).

1. Identify an area by using one of the three menu options: CURRENT SCREEN, DRAW POLYGON, or SELECT POLYGON.

2. Pick EDIT POLYGON CELL VALUES.

You will see this prompt:

ENTER NEW CELL VALUE (CTRL-Z TO ABORT) :

3. Enter an impedance value of 0 (for soft) or 1 (for hard), or press <CTRL Z> to cancel the request.

If you enter an impedance value, that value will apply to all cells within the selected polygon.

- RETURN IMPEDANCE: Displays the impedance at a selected point.

1. RETURN IMPEDANCE.
2. Pick a point on the screen.

A yellow asterisk (*) will mark the selected location. Depending on the screen density of the impedance data, a box representing the borders of the impedance data cell can also appear.

3. Pick CONTINUE.

The impedance of the selected point (center point of the cell) will be displayed.

- ENABLE IMPEDANCE RES: Shows the resolution of the impedance data by drawing white grid lines around the impedance data cells.

To show the impedance data, first pick ENABLE IMPEDANCE RESOLUTION. If the current screen requires more than 50 rows or columns of cells in the impedance data grid, you will see a warning message and a count of the number of rows and columns:

* * * WARNING * * *

TAC DATA IS VERY DENSE

COLUMNS : 224
ROWS : 175

Do you want to continue? (Y/N) == >

Because displaying a dense impedance data grid can effectively obliterate the other displayed data, you should consider answering No and zooming in to a smaller area before displaying the impedance grid.

- DISABLE IMPEDANCE RESOLUTION: Erases the display of a display of a line grid of impedance data cells.

- **DISPLAY IMPEDANCE STATS:** Displays information about the impedance data with respect to 1) the base map and 2) the current view (screen display). The information includes the longitude-latitude extents, the number of rows and columns in the cell grid, and the X and Y step sizes (east-west and north-south cell sizes).

TAC DATA STATISTICS

PROJECTION TYPE: Geographic (Lat-Long)
 MINIMUM X COORDINATE -112.833000
 MAXIMUM X COORDINATE -111.871800
 MINIMUM Y COORDINATE 39.948000
 MAXIMUM Y COORDINATE 40.673000
 COLUMNS: 1135
 ROWS : 872
 X STEP SIZE: 0.000848
 Y STEP SIZE: 0.000832

TAC DATA IN THE CURRENT VIEW

MINIMUM X COORDINATE -112.459464
 MAXIMUM X COORDINATE -112.270333
 MINIMUM Y COORDINATE 40.237602
 MAXIMUM Y COORDINATE 40.382797
 COLUMNS: 224
 ROWS : 175

Figure 7.10. Example Impedance Stats Display

- **DISPLAY ATTRIBUTES:** Displays the attributes associated with a selected item on the graphics screen. For example, you can display the attributes associated with a particular siren icon.
1. Pick DISPLAY ATTRIBUTES.
 2. Pick a point on or near the desired point or composite string.

The data item will blink, and if it has attributes associated with it, they will be displayed in a table. For example, this table displays information on two attributes:

#	MAJOR	MINOR	PARAM	DESCRIPTION
1	8	1400	0	Seaplane ramp or Landing area
2	8	7	6	NO ENTRY IN ATTRIBUTE DICTIONARY

where # is the index number and the following four columns list each attribute (the Major code, the Minor code, and the Parameter value) plus a text description of the data item, if a description exists in the attribute

dictionary. Notice that the second set of attributes has NO ENTRY IN ATTRIBUTE DICTIONARY: that is, the attributes 8;7;6 might have some meaning for the person who created the data item, but they are not in the master list of attributes currently used by the map database.

3. Repeat Step 2 for any other data item.

As you continue to pick data items, old attribute information is retained on the text screen. To clear the screen and make it easier to read, press the <D ERASE> key.

4. Pick any other submenu item to end the DISPLAY ATTRIBUTES function.

The following four standard viewing functions are included in the EDIT ELEVATION FILE menu for convenience. Their functions are described in detail as options of the Map Edit main menu (see the *IBS Utilities Guide*).

- ZOOM IN: Enlarges a square portion of the displayed area.
- ZOOM OUT: Reduces the currently displayed area to a square area within a larger displayed area.
- RESTORE BASE PLOT: Redraws the **base plot** in the display area.
- REFRESH SCREEN: Redraws the current display.

7.10 Place Siren - Graphic Menu Only

This option enables you to specify where sirens will be located on the site for the current case. When you pick PLACE SIREN from the OSPM menu, the system displays the menu shown in Figure 7.10.

The menu features options for placing or deleting sirens on the site for the current OSPM case. The graphic mode enables you to place or delete sirens by picking the location of siren icons on the site map. Before you place a siren, you may need to specify what type of siren is being placed (the **default siren**).

The principal options of this menu are described in the following sections.

Figure 7.11. Menu for Place Siren - Graphic Mode

7.10.1 Change Default Siren

This option enables you to change your **default siren**--the type of siren that is inserted when you use the PLACE SIREN function. When you select CHANGE DEFAULT SIREN, the system displays a menu similar to the one shown in Figure 7.11. This menu shows your current library of siren types, which can contain up to 30 types of sirens. (You can add siren types to this menu: see Section 7.17, **Create a New Siren Type**.)

To change the default siren type:

1. Use the arrow keys to move the cursor down the column of siren numbers.
The siren names will scroll up to reveal more sirens (if any are present).
2. When you have found the desired siren type, enter its number in the field at the top of the screen.

3. Enter <GOLD> <Z>.

You will then return to the PLACE SIREN menu. The next siren that you place will be the selected type.

CHANGE DEFAULT SIREN			
Current default siren name: ACA PENETRATOR			
Enter siren number (1-30): <u>1</u>			
Sirens available in the Library:			
	Type		Power
1	ACA PENETRATOR	R	120.0
2	ACA PENETRATOR DUAL	R	117.0
3	ACA BANSHEE	N	111.0
4	ACA CYCLONE	N	123.0
5	FS 110	R	113.0
6	FS STH10	R	113.0
7	FS THUNDERBOLT	R	125.0
8	FS THUNDERBOLT DUAL	R	122.0
9	FS THUNDERBEAM	R	124.0
10	WS 3000	R	120.0

Figure 7.12. Form for Change Default Siren

7.10.2 Add Siren

This option enables you to add a siren by 1) picking a location and 2) specifying the height of the siren. The added siren will be of the default siren type. You can pick any location on the map.

Picking the Siren Location in Graphic Mode:

1. Pick PLACE SIREN on the menu in Figure 7.10.

The menu item will start blinking.

2. Pick a point for the siren location.

A white location marker will start blinking.

A siren icon will replace the blinking location marker. (If the siren is not in your siren library, the system will require that you Create a New Siren Type by entering information on a siren definition form. Refer to Section 7.17.)

Specifying the Siren Mounting Height:

After specifying the siren location, the system displays a form for specifying the mounting height of the new siren (Figure 7.12). You will be able to add only the siren height on this form. After entering this value, you will return to the Place Siren menu.

ADD A SIREN							
SIREN SPECIFICATION							
Siren Model Name: ACA PENETRATOR							
Latitude	44.00904389			(deg)			
Longitude	-69.53766394			(deg)			
Mounting Height:	22.00			(ft)			
Siren Type: ROTATING							
Siren Reference Sound Pressure Values at 100 ft.							
0.00	0.00	0.00	120.00	119.00	116.00	111.00	105.00
63 Hz	125 Hz	250 Hz	500 Hz	1KHz	2KHz	4KHz	8KHz

Figure 7.13. Form for Specifying Siren Height (Example)

7.10.3 Delete Siren

This option enables you to delete an existing siren by 1) picking the siren location and 2) verifying that the siren found at that location should be deleted. You can just pick a location on the map.

Picking the Siren Location in Graphic Mode:

1. Pick DELETE SIREN on the menu in Figure 7.10.

The menu item will start blinking.

2. Pick a point on or near a siren icon on the site map display.

A white asterisk positioned on the siren icon will start blinking.

3. Pick CONTINUE.

The system then displays a form for verifying siren deletion (Figure 7.14).

DELETE A SIREN							
SIREN SPECIFICATION							
Siren Model Name: ACA PENETRATOR							
Latitude 44.00904389 (deg)							
Longitude -69.53766394 (deg)							
Mounting Height: 22.00 (ft)							
Siren Type: ROTATING							
Siren Reference Sound Pressure Values at 100 ft.							
0.00	0.00	0.00	120.00	119.00	116.00	111.00	105.00
63 Hz	125 Hz	250 Hz	500 Hz	1KHz	2KHz	4KHz	8KHz
Is this the siren to DELETE? (Y/N) <u>Y</u>							
Do you want to look for other sirens at this location? (Y/N) <u>N</u>							

Figure 7.13. Form for Verifying Siren Deletion (Example)

Verifying the Siren Deletion:

A prompt at the bottom of the form will ask whether this is the siren that you want to delete.

1. Answer Y (to delete) or N (if this siren should not be deleted).

A second prompt will ask whether you want to look for other sirens at this location.

2. Answer Y or N.
 - If you answer N, then you will return to the Place Siren menu.
 - If you answer Y, the system checks for more sirens at the same location and (if it finds one) displays another deletion verification form. If no other sirens can be found, then you will return to the Place Siren menu.

7.11 Update Siren - Graphic Menu Only

This option enables you to update an existing siren by 1) picking the siren location and 2) entering siren information on an Update Siren specification form (Figure 7.14).

Picking the Siren Location. You can pick a siren icon on the map, just as you would pick a siren for deleting it.

Updating the Siren. After specifying the siren location, the system displays a form for updating the siren information (Figure 7.14). Only two fields on the form are updatable:

- * **Siren Model Name:** Enter the name of a siren that is in your library of siren types (Section 7.10.1). If you enter a different siren name, it can be represented by a different icon. The system will automatically delete the existing icon and draw the new one when finished displaying forms. It will display a new form with updated values on exit.
- * **Mounting Height:** Enter a number of feet above the ground.

A prompt at the bottom of the form will ask whether you want to update other sirens at this location. If you answer N, then you will return to the OSPM menu. If you answer Y, the system checks for more sirens at the same location and (if it finds one) displays another Update Siren form. If no other sirens can be found, you will return to the OSPM menu.

UPDATE A SIREN							
SIREN SPECIFICATION							
Siren Model Name: <u>ACA PENETRATOR</u>							
Latitude 44.00904389 (deg)							
Longitude -69.53766394 (deg)							
Mounting Height: <u>22.00</u> (ft)							
Siren Type: ROTATING							
Siren Reference Sound Pressure Values at 100 ft.							
0.00	0.00	0.00	120.00	119.00	116.00	111.00	105.00
63 Hz	125 Hz	250 Hz	500 Hz	1KHz	2KHz	4KHz	8KHz
Do you want to look for other sirens at this location? (Y/N) <u>N</u>							

Figure 7.15. Form for Updating a Siren (Example)

7.12 List Site Specs

OSPM Text Menu: Select C (List/Update Site Specs)

This option displays a form (Figure 7.15) that enables you to enter the various inputs required by the model. The top portion contains various meteorological data for the model, and the lower portion contains the grid definition information for the model. The grid definition information is important because if you specify a larger number of grid points in each direction, you will get a more detailed set of output.

METEOROLOGICAL DATA:			
Wind Direction	232.0	(degrees clockwise from north)	
Ground-Level Wind Speed	3.6	(mph)	
Reference Higher-Elevation Wind Speed	4.5	(mph)	
Ground-Level Temperature	74.5	(degrees F.)	
Higher-Elevation Temperature	73.4	(degrees F.)	
Ground-Level Elevation	35.0	(feet)	
Reference Higher-Elevation	197.2	(feet)	
Relative Humidity	64.0	(%)	
Barometric Pressure	764.000	(mm Hg)	
GRID DEFINITION:			
Center of interest	-69.6964000	Longitude	43.95150000 Latitude
Radius of interest	17000	meters	
Number of grid points in each direction	0020		
CASE DESCRIPTION:			
FIRST TEST OF OSPM FOR MAINE YANKEE. 17 KM RADIUS 20X20 GRID			

Figure 7.15. List Site Specs Form

7.13 Execute OSPM - Interactive

OSPM Text Menu: Select 2 (Execute OSPM Interactive)

Selection of this function from the OSPM top-level menu causes the Outdoor Sound Propagation Model to run using data for the current case. This will then enable you to graphically display the results through the Map Display software.

You will first see a checklist (Figure 7.16) and a prompt asking whether you wish to run the model. The checklist is displayed for your information only: it is NOT a form you can change. Read the checklist to be sure the case is set up to run as you wish. Then respond to the prompt:

- If you answer N (no), you will return to the menu without running the model.
- If you answer Y (yes), the model will run. While the model is running, you cannot interact with any other IBS function. The system will return to the OSPM menu when the model has completed its run.
- If you have made changes to the input data, the system will prompt

Save the OSPM updates? (Y?N) [Y] == >

If you answer Y (yes) or just <RETURN>, the system will display the following message:

Saving Updates.

Press <RETURN> to continue....

Current OSPM Case Attributes

Case: 2C001

Site: JCB1

Name: My Planning Site

Min Longitude: -112.45946448

Max Longitude: -112.27033288

Min Latitude : 40.23760238

Max Latitude : 40.38279710

Sirens : 11

Execute OSPM for this case? (Y/N) [N] == >

Figure 7.17. Checklist for OSPM Model Execution

7.14 Execute OSPM - Batch

Batch execution is convenient for large modeling runs. This function is identical to Execute OSPM - Interactive except:

- If you answer Y (yes) to the check list prompt, you will see the modeling job has been added to a system batch queue. While the model is running (or waiting to run), you will be returned to the OSPM top-level menu. At this point, you can exit the system or perform other IBS functions. However, DO NOT interact with OSPM while the model is running: this could cause problems with the input data sets.

7.15 Perform File Maintenance

When you select PERFORM FILE MAINTENANCE from the OSPM top-level menu, the system displays the menu of file maintenance functions shown in Figure 7.17. The menu options enable you to list the OSPM cases, delete case outputs, or delete entire cases by deleting both inputs and outputs.

```

                                PERFORM FILE MAINTENANCE

Selection
Designator      Selection

    A      Display Directory of Cases
    B      Delete Case Output for a Specified Case
    C      Delete Case Input and Output for a Specified Case
    X      Exit to Previous Menu

                                Enter Selection Designator: __

```

Figure 7.18. File Maintenance Functions Menu

7.16 Delete Contours - Graphic Menu Only

Selecting this option will remove from the graphics display any sound pressure contours that have been drawn with the CONTOUR RESULTS option.

7.17 Create a New Siren Type

OSPM Text Menu: Select 7 (Create a New Siren Type)

The capability to create a new siren type for use by OSPM is available through both the graphic and text menus.

Text Menu Option 7. When you select this function from the OSPM top-level text menu, the system will display the Create a New Siren Type form (Figure 7.18). This form enables you to define a new siren type and then places it in your siren library (Figure 7.11), where you can select it for adding sirens.

Graphic Menu PLACE SIREN Option. Creating a new siren type through the graphic menus requires special use of the PLACE SIREN option on the top-level OSPM menu:

1. Pick PLACE SIREN on the OSPM graphic menu.
2. Pick CHANGE DEFAULT SIREN on the resulting PLACE SIREN submenu.
3. Enter a siren number that is currently unused in your siren library (see Section 7.10.1, **Change Default Siren.**)
4. Pick PLACE SIREN on the PLACE SIREN submenu.
5. Pick a location for the new siren on the site map.

The system will display the following message:

Siren Name Not Found. Do you want to add this siren to
your library? (Y/N)

6. Answer Y (yes).

The system then displays the Create a New Siren Type form shown in Figure 7.18.

7. Enter a siren name, type, and sound pressure values.

The system then places the siren description into the siren library and locates a siren icon at the selected location.

CREATE A NEW SIREN TYPE

SIREN SPECIFICATION

Siren Model Name: ACA PENETRATOR

Siren Type: ROTATING

Siren Reference Sound Pressure Values at 100 ft.

<u>0.00</u>	<u>0.00</u>	<u>0.00</u>	<u>120.00</u>	<u>119.00</u>	<u>116.00</u>	<u>111.00</u>	<u>105.00</u>
63 Hz	125 Hz	250 Hz	500 Hz	1KHz	2KHz	4KHz	8KHz

Figure 7.19. Siren Specification Form for Creating a New Siren Type
(Example)

The fields of the form are:

- * Siren Model Name: Enter a siren name. This name will be added to the siren library at the next available siren number.
- * Siren Type: Enter either "R"otating or "O"nni-Directional.
- * Reference Sound Pressure Values: For each octave-band frequency, enter a value for sound pressure at 100 ft.

7.17 Create a New Siren Type

Text menu select 4 (Change Site)

You will be presented with a menu similar to the following example from which to select a site.

Select New Site

```
.....  
0: EXIT  
1: JCB1 [Plan] == > TEAD [Real]      (Current)  
2: ALAB (Plan) == > SALA [Real]  
.....
```

Example of Change Site Menu

Select a site by entering a number at the left. You will see a message indicating the system is resetting to the new site and you will then be placed in the top level OSPM menu (either text or graphic).

Section 8

Model Library - CHEMS Models

IBSSH Menu: Select Models (Graphics) or Models (New Graphic) Library)
Model Library Menu: Select CHEMS (Chemical Model)

This section describes the **CHEMS Models**, a set of three computer models (Explosion, Fire, and Dispersion) that help in assessing the explosion shock wave, thermal radiation, and vapor dispersion hazards arising from the accidental releases of chemicals. This section covers the following topics.

8.1 Description of the CHEMS Models

- Explosion Model
- Heat Radiation Model
- Dispersion Model

8.2 Using the CHEMS Models

- Overview of the Explosion Model
- Overview of the Heat Radiation Model
- Overview of the Dispersion Model

8.3 Using the CHEMS Model Menu

8.4 Update Case Data

8.5 Select Spill Location

8.6 Change Case

8.7 Base Map Display

8.8 Execute Model Inter and Execute Model Batch

8.9 Display/Print Reports

8.10 Perform File Maintenance

8.1 Description of the CHEMS Models

The CHEMS Models are a set of computer models that utilize a chemical specific property database and a series of physical and chemical behavior models to calculate the potential areas of hazard resulting from chemical release events. Explosion, fire, and vapor dispersion models are supported. Each model produces text output and topographic output consisting of polygons describing the hazard zones computed. The polygons have attributes that may be viewed using standard IBS graphic picks to obtain additional information on the hazard zones calculated.

8.1.1 Explosion Model

The Explosion Model is a computer model that calculates the overpressure and impulse of the shock wave produced if a vapor cloud is ignited and detonates. The model is based on the mass of vapor exploding, the heat of combustion of the chemical exploding, and TNT-equivalent damage of such an explosion. The mass of chemical exploding is assumed to be 10% of the vapor released in an accident. Experiments and investigations of accidents show that where a chemical can detonate in semiconfined or unconfined geometry, the efficiency of explosion (that is, the fraction of the released vapor mass participating in detonation) ranges between 3% and 10%. The heat of combustion for the chemical is obtained from the CHEMS chemical property database. The TNT-equivalent damage areas are computed for various types of damage, each of which is provided as an output polygon. The output from the model is accurate only in the far field from the exploding vapor cloud.

8.1.2 Heat Radiation Model

The Heat Radiation Model is a computer model that calculates the thermal radiation field and areas of potential hazard (**hazard zones**) around a **liquid pool fire**. (A liquid pool fire occurs when a pool of combustible liquid or a flammable solid burns.) The model is based on the laws of thermal radiation exchange between objects. Hazard zones are defined as areas within which the thermal radiative heat flux is above a specified value for an observer or target. IBS shows the results of the model as heat flux level isopleths (contours) overlaid on a map of the fire site. The model provides the constant heat flux contours for specified orientations of the heat-receiving target. The object plume may be horizontal, vertical, or oriented to receive the maximum heat flux at a particular position.

8.1.3 Dispersion Model

The Dispersion Model calculates the areas of potential hazard (**hazard zones**) arising from the atmospheric dispersion of vapors of a hazardous chemical. The model calculates the vapor concentration distributions both horizontally and vertically above ground at different downwind locations from the release point. Both instantaneously released vapor mass dispersion (puff dispersion) and the dispersion of vapors released continuously (plume dispersion) are modeled. Heavy

gas effects in the dispersing fluid arising from the higher-than-air density of the cloud/plume are taken into consideration. The higher-than-air density can result from the high molecular weight of the chemical vapor, or from the low temperature of the vapor, or from the entrainment of liquid aerosol droplets in the dispersing cloud or plume. The model results are presented as the isopleths of specified concentrations within the cloud or the plume. These isopleths are superposed on a local area map.

The nature of the source can be described in terms of the geometry of the type and geometry of the container releasing the chemical and the initial condition of the chemical in the container before release. Three classes of chemical initial conditions are modeled, namely, liquified compressed gas, high-vapor-pressure liquid, and cryogenic liquid.

8.2 Using the CHEMS Models

The CHEMS Model subsystem is a software product of Technology & Management Systems, Inc., Burlington, Massachusetts.

With the information supplied in this section, you will be able to execute the CHEMS Models and obtain model results. To use the CHEMS Models in IBS, modify the data for the current CHEMS case (Section 8.4), select spill location (Section 8.5), execute the model (Section 8.8), and print or display the numerical results (Section 8.9) or display the results graphically (Section 8.7).

Section 8.4 (**Modify Case Data**) describes the various inputs required by the models. The inputs include the source location, chemical properties, atmospheric conditions, heat flux levels and critical concentrations (of hazard zones), and the characteristics (such as height or orientation) of the source. Some input is common to the CHEMS Models. Some, such as the orientation of observer or the target receiving thermal radiation, is specific to a particular model. The following overview of the models may be helpful in understanding the model inputs.

8.2.1 Overview of the Explosion Model

When a flammable and detonable vapor is released from a container into the atmosphere, it forms a vapor cloud that mixes with the air in the atmosphere. Certain parts of the cloud will have concentrations of the vapor/air mixture that will be in the detonable range. If the cloud is ignited and the flame propagating into the cloud transits to a detonation wave, a shock wave is created, which then propagates at supersonic speeds in the atmosphere. Not all flammable clouds detonate when ignited. In most cases, the vapor cloud will burn with a deflagration flame front. The explosion model assumes that the vapor cloud will explode.

The shock wave created of a detonating vapor cloud can cause damage to human beings and structures. The strength of the shock wave diminishes almost inversely

as the square of the distance (at far distances) from the source of detonation. Damage can be caused by either the overpressure or the impulse of the shock wave or both. The extent of damage depends on the distance of the object from the source and the mass of vapor that detonates.

The TNT-Equivalent Model is used to relate the shock-wave pattern arising from detonation of an equivalent mass of TNT to various types of blast damage. If the mass of the flammable vapor in a vapor cloud that participates in detonation is known, and if this mass can be related to an equivalent mass of TNT, the overpressure field (at least in the far field) arising from detonation of the gas in the vapor cloud can be calculated. The distance to a particular type of damage can then be calculated, knowing the overpressure for damage and knowing the pressure field. This model is explained in more detail in the following paragraph.

(i) **TNT Blast Overpressures:** The variation of direct peak overpressure as a function of distance is scaled with the third power of the mass of TNT. This is because the distance to a given overpressure is proportional to the cube root of the mass of TNT detonating.

(ii) **TNT Equivalent Mass:** The equivalent mass of TNT of an exploding vapor is calculated using the equation

$$m_{\text{TNT}} = m_{\text{Vapor}} * \frac{(\text{Heat of Combustion of Chemical})}{(\text{Heat of Combustion of TNT})} \quad (8.1)$$

where

m_{TNT} = equivalent mass of TNT for the exploding chemical

m_{Vapor} = mass of chemical vapor detonating with air.

The heat of combustion of TNT = 1120 cal/gm = 4.688×10^6 J/kg.

(iii) **Mass of Vapor Exploding:** The mass of vapor in a flammable vapor cloud that can explode (assuming the chemical is detonable in the open environment) is very difficult to calculate. The calculation of this parameter requires the description of the dispersion of vapor and its mixing with the air. Then the mass of vapor in the detonable range within the cloud has to be calculated. Finally, it has to be assumed that all of this vapor with concentration in the detonable range participates in the detonation. The three-dimensional calculations that are involved are not included in the model.

However, it is known from experiments and investigations of accidents involving gas explosions in the open that between 3% and 10% of the mass of vapor released in the accident participates in the explosions phenomenon. Based on this, a conservative factor of 10% is included in the model for the amount of vapor in a vapor cloud that explodes, if the chemical is capable of detonating in the open.

(iv) **Blast Damage Criteria (P_{haz}):** The damage to structures and people depends on the peak overpressure, impulse of shock, and the duration over which the blast effects last. For risk analysis purposes, the model considers only the effects of peak overpressures. Table 8.1 lists the level of peak overpressures at which the indicated types of damage occur. It is noted that glass panels can break at relatively low pressure pulses (of 0.75 psi). To evaluate the hazard area arising from potential gas explosions, it is suggested glass breakage is used as the damage criterion.

Table 8.1. Overpressure Damage Criteria Due to Blast Wave

<u>Type of Damage</u>	<u>Overpressure, psi</u>	<u>Overpressure, pa (N/m²)</u>
Glass window breakage	0.75	5,170
Wood siding panel collapse	1.5	10,340
Ear drum rupture, concrete block collapse	2.5	17,240
Lung damage	6.0	41,370

(v) **Hazard Distance Calculation:** The salient steps of the procedure by which the blast or explosion hazard area is calculated are shown below:

1. With the known chemical that is released and the mass released, the mass of vapor that may participate in an explosion is calculated by using the formula

$$m_{\text{exploding}} = m_{\text{released}} \times \text{fraction detonated}$$

where the fraction is generally set equal to 0.1.

2. The equivalent TNT mass is calculated using this Equation.
3. The scaled distance to a specified overpressure is obtained from the TNT blast overpressure versus distance correlations.
4. The actual hazard distance is then calculated for each of the five damage peak overpressures.

8.2.2 Overview of the Heat Radiation Model

The software models a large turbulent pool fire that is described by a base diameter, a time-averaged plume length, and a tilt caused by wind. This geometric representation of a turbulent fire as an equivalent, tilted, circularly cylindrical fire is illustrated in Figures 8.1 and 8.2.

Figure 8.1. Geometric Representation of a Turbulent Fire by an Equivalent Tilted Cylindrical Fire

Figure 8.2. Heat Radiation Model

The model calculates the distance at which a target at a specific height, orientation, and radial position (relative to the tilted fire) will receive a specified heat flux. These distances are calculated over a range of radial positions to generate hazard zones and contours defined by the specified heat flux. Model parameters are illustrated in Figure 8.2. For details on model equations and assumptions, refer to the model documentation produced by Technology & Management Systems (see Section 8.2.4).

Limitations of the Model. The Heat Radiation Model has two limitations that should be observed. The first limitation is the model is generally designed to describe a liquid pool fire, such as a liquefied natural gas fire. The model has been validated in field-scale tests with liquefied natural gas fires (Report # NTIS AD-A077073). Application of the model to other types of fires, such as a fire that starts within a structure, would be questionable, even if efforts were expended to relate the fire to an equivalent liquid pool fire.

A second limitation of the Heat Radiation Model in IBS is the model is designed to describe heat fluxes from fires of relatively small scale, compared to the resolution of most of the data now in IBS. You probably would want to apply the Heat Radiation Model only in areas where the included data are detailed enough for the heat flux contours to be meaningful (as in 1:00K or TIGER data).

8.2.3 Overview of the Dispersion Model

Dispersion of toxic or flammable vapors in the atmosphere occurs when compressed liquified gas, a liquid chemical of high vapor pressure, or compressed gas is released and forms a vapor cloud or a plume of vapor. Depending on the severity of the breach of chemical containment, the physical state in which the chemical is stored or transported, and the properties of the chemical, several different types of dispersion phenomena can occur, are discussed in following paragraphs.

Release of Liquified Compressed Gas (LCG): When a LCG is released, a part of the liquid coming out of the hole flashes directly into vapor and the other part remains in the liquid phase. This latter liquid, which will be at the saturation temperature corresponding to ambient pressure, may be entrained in the vapor cloud as fine aerosol droplets, or it may form a puddle on the ground, or both. The vapor cloud formed, for most LCG releases, is generally heavier than the air because of liquid droplet entrainment and, in some cases, because of the density of the vapor at saturation temperature being heavier than that of air. The vapor cloud is then moved downwind by the prevailing wind and becomes diluted. The dispersion of this heavy gas is modeled by heavy gas models.

Release of a High Vapor Pressure Liquid: When a chemical that is normally a liquid at ambient temperature and pressure is released from the container onto the ground, it first spreads and forms a pool on the ground. The size of the pool will depend on whether or not there is any kind of natural diking in the vicinity of the spill. The liquid at the same time evaporates because of heat transfer from the environment (ground and air). The vapors produced by this liquid pool evaporation then disperse into the atmosphere.

Release of a Cryogenic Liquid: When a cryogenic liquid (that is, a liquid transported or stored at ambient pressure but at considerably below the ambient temperature) is released, the liquid boils on the ground surface and liberates vapor. The rate of the liberation of vapor from the boiling pool is high initially because of the warm ground beneath the liquid. However, as the ground cools, the rate of heat transfer into the liquid pool diminishes. Calculation of the dispersion of vapors under this condition requires the knowledge of conditions and properties of the ground, the atmosphere, and the chemical.

The vapor dispersion model included in the CHEMS Models in IBS is based on a heavy gas dispersion model called the Air Force Dispersion Assessment Model (ADAM) developed for the U.S. Air Force. This model takes into account the effects of possible higher-than-air density of the released cloud, reaction, if any, of the cloud with the atmospheric water vapor, presence of liquid aerosols that evaporate or react during the dispersion process, and the combined effects of these on the thermodynamic state of the cloud at all positions downwind of the release point. The ADAM also has several source models to simulate the release of various types of chemicals (pressurized liquified gases, cryogenic liquids, high vapor pressure liquids, compressed gases) and the behavior of the released chemicals on various types of substrates. Jet releases are also modeled. Details of the ADAM

model can be found in the U.S. Air Force publication #G-89-55 listed in Supplementary References.

8.2.4 Supplementary References

Pacific Northwest Laboratory (PNL). Liquid Pool Fire Thermal Radiation Model. A Computer Code For Integration into the Integrated Emergency Management Information System (IEMIS). A Report and User Manual. Prepared by Technology & Management Systems, Inc. for the Pacific Northwest Laboratory, Richland, WA.

U.S. Coast Guard. 1979. Experiments Involving Pool and Vapor Fires from Spills of Liquefied Natural Gas on Water. NTIS AD-A077073.

Raj, P.K., and Morris, J. A. 1988. Source Characterization and Heavy Gas Dispersion Models for Reactive Chemicals. NTIS# ADA 200-121, TMS Report # G-85-42 to the U.S. Air Force, Air Force Systems Command, Hanscom AFB, MA, January 1988.

Raj, P.K. 1990. Hydrogen Fluoride and Fluorine Dispersion Models Integration into the Air Force Dispersion Assessment Model (ADAM). TMS Report # G-89-55 to the Air Force Geophysical Laboratories (LYA), Hanscom AFB, MA.

8.3 Using the CHEMS Models Menu

Models Menu: Select CHEMS (Chemical Model)

Depending on your selection of Graphic or Non-Graphic Models, you will see one of the menus of the CHEMS Models subsystem (Figures 8.3 or 8.4).

EXIT	
HELP	
CANCEL	
CONTINUE	
UPDATE CASE DATA	8.4
SELECT SPILL LOCATION	8.5
CHANGE CASE	8.6
BASE MAP DISPLAY	8.7
EXECUTE MODEL INTER	8.8
EXECUTE MODEL BATCH	8.8
DISPLAY/PRINT REPORTS	8.9
PERFORM FILE MAINT	8.10

Figure 8.3. CHEMS Models Menu - Graphic Menu

```

CHEMSHAZ MODEL

Selection
Designator      Selection
A               Modify Current CHEMSHAZ Case Data
B               Execute Current CHEMSHAZ Case
C               Change Case Number
D               CHEMSHAZ Case Directory and Optional Delete
E               Print/Display Report
X               Exit

CHEMSHAZ Model Case Number:    000  Based on: 000
Site Name:MAIN

Please Enter Selection Designator:  _

```

Figure 8.4. CHEMSAZ Models Menu - Text Menu

The CHEMS Models graphic menu contains all the subsystem functions, including display of the site map and model results. All other functions are also available on the CHEMS Models text menu.

To select a function from the CHEMS Models menus:

Graphic Menu - Pick the desired function.

Text Menu - Type the corresponding selection designator (letter) and press <RETURN>.

The CHEMS Models functions are described in the following sections.

Standard CHEMS Base Case. Whenever a site is created, the input data for the CHEMS base case number 000 are also created. The base case contains some test data for a gas cloud explosion, but has no other particular significance. The base case can be used to demonstrate execution of the CHEMS Explosion Model and generate explosion damage contours.

8.4 Update Case Data

CHEMS Text Menu: Select Modify Current CHEMSHAZ Case Data

Graphics Menu: Select Update Case Data

When you pick this option from the CHEMS Models menu, IBS displays a text menu of case data modification selections as shown in Figure 8.5.

CHEMSHAZ MODELS CHANGE MODEL PARAMETERS	
Selection Designator	Selection
A	CHEMICAL SPILLED
B	TIME OF SPILL / CURRENT TIME
C	ENVIRONMENTAL INPUTS
D	MODE OF OPERATION
E	MODEL SELECTION / MODEL DEPENDENT INPUTS
F	SOURCE INPUTS
G	OUTPUT SPECIFICATIONS
X	Exit
Please Enter Selection Designator: _	

Figure 8.5. CHEMSHAZ Models - Change Model Parameters Menu

8.4.1 Chemical Spilled

When you select this option (Selection A), a form is presented providing you five options for specifying a Chemical Identification field:

.....
1..... CHRIS Chemical Code
2..... Chemical Name
3..... #
4..... UN / DOT #
5..... STCC #

Enter a Selection Number :__

Enter Chemical Identification :__

The Chemical Hazard Response Information System (CHRIS) chemical code is a 3-letter alphabet code used for each of approximately 1200 chemicals by the U.S. Coast Guard in its CHRIS. The Chemical Abstracts Service (CAS) number is the number assigned for each chemical. The UN/DOT number is a 4-digit number by which several chemicals are identified by the U.S. Department of Transportation (DOT). These numbers for the chemicals are spelled out in DOT Regulations in 49 CFR. The STCC number is a number designation used by the railroad industry.

Tank Specification Form

After entry of the chemical to be studied, or after completion of other forms described later if not completed previously, CHEMS presents a form allowing you to provide TANK SPECIFICATION information (Figure 8.6). A default Tank Gauge Pressure is provided; you may override it if you wish. If you elect to override the value by placing a "Y" in the first field on the form, a small window will appear in the lower right corner of the form showing units available for specifying tank pressure.

TANK SPECIFICATION	
Override tank pressure value shown (Y/N)	—
Tank Gauge Pressure:	_____ Pascal
Override tank temperature value shown (Y/N)?	—
Tank Temperature :	_____ K
	<div style="border: 1px solid black; width: 80px; height: 25px; margin-left: 100px;"></div>

Units Available	Unit Codes
psi	1 psi
inches of water	2 inches of water
mm Hg	3 mm Hg
atm	4 atm
bars	5 bars
Pascal	6 Pascal

==>

Figure 8.6. Tank Specification Form

The second line on the form contains two fields. The first is a floating point number that, when entered, causes the lower right window to be updated with numeric values for selection of units available to be specified in the second field, as shown. Entry of the unit code updates the form with the associated units and advances the cursor to the next field. This method of providing a variety of input units for specification of data to be used by CHEMS models is found frequently in forms described in this section.

The next field allows you to override tank temperature value shown. If you elect to override the value by placing a Y in this field, a small window will appear in the lower right corner of the form showing units available for specifying tank temperature. Fahrenheit, Celsius, and Kelvin units are supported. Interaction with this field and unit code specification is similar to corresponding fields for tank pressure. On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

8.4.2 Time of Spill / Current Time

When you select this option (Selection B), a form is presented that shows the current date and time and allows you to specify the date and time of the accident. On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

8.4.3 Environmental Conditions

When you select this option (Selection C), a menu is presented as shown in Figure 8.7.

ENVIRONMENT CONDITIONS	
Selection Designator	Selection
A	Atmospheric Conditions
B	Ground Conditions
C	Water Conditions
X	Return to Change Model Params Menu
Please Enter a Selection Designator: _	

Figure 8.6. CHEMSHAZ Models - ENVIRONMENT CONDITIONS Menu

Atmospheric Conditions

When you select this option (Selection A), two forms are presented in sequential fashion. They show a number of atmospheric parameters that can be specified. Some of the parameters have options for unit specification, which are provided in a window as described previously.

The first form allows you to enter:

Air Temperature as Fahrenheit, Celsius, or Kelvin.

Relative Humidity as percent.

Wind Speed as mph, knots, ft/s, m/s, or km/hr.

Height at which wind speed is measured in meters.

Wind Direction in degrees.

The second form allows you to enter:

Atmospheric stability as a numeric equivalent of Pasquill stability classes.

Wind statistical data:

Sigma Theta in degrees

Averaging time in minutes

Cloud Transmittance as a decimal value in the range or 0. to 1.

Sigma Theta represents the standard deviation in degrees of the wind direction. That is, it is a statistical measure of the variability in the wind direction; the smaller this number, the more steady the direction from which the wind is blowing. This value is related to the overall stability of the atmosphere and, hence, the dispersion characteristics of the atmosphere.

On completion of this form, you are returned to the ENVIRONMENT CONDITIONS Menu.

Ground Conditions

When you select this option (Selection B), a form is presented showing a number of ground condition parameters that may be specified. Some of the parameters have options for unit specification, which are provided in a window as described previously.

The form allows you to enter:

Temperature of Ground in Fahrenheit, Celsius, or Kelvin.

Surface Roughness in the dispersion region (a menu of six options is provided, relating descriptions of ground conditions to ground roughness measurements)

Surface Roughness at Met Tower (a menu of six options relates descriptions of ground conditions to ground roughness measurements)

Ground condition - Dry, Wet, or Snow may be specified.

On completion of this form, you are returned to the ENVIRONMENT CONDITIONS Menu.

Water Conditions

When you select this option (Selection C), the form presented shows a number of water condition parameters that may be specified. Some of

the parameters have options for specifying units. These are provided in a window as described previously.

The form allows you to enter:

Description of Water Body as River/Urban, River/Rural, Coastal Zone, Port Area, or Open Water.

River Environmental Conditions:

Water Temperature as Fahrenheit, Celsius, or Kelvin.

Current Speed as mph, knots, ft/s, m/s, or km/hr.

Depth in feet or meters

Width in feet or meters

Spill location as Mid-River, Near Shore, or Shoreline.

On completion of this form, you are returned to the ENVIRONMENT CONDITIONS Menu.

8.4.4 Mode of Operation

When you select this option (Selection D), a menu is presented as shown in Figure 8.8.

OPERATION MODE

Choose Mode of Operation

1. Emergency Mode
2. Contingency Mode

Choice: _

Figure 8.8. Form for Selecting Mode of Operation

The Emergency Mode requires less accurate field data to run. It recognizes that in an emergency not all of the model parameters will be available and, even if available, only approximate values may be found. For example, the diameter of a hole in a tanker leaking a chemical in an accident may not be available accurately. Hence, in Emergency Mode, only a qualitative descrip-

tion of the parameter (in this case, a small, medium, or large hole) is sought. Also in the Emergency Mode, some of the parameter values are built into the calculation and may not even be sought from the user.

In the Contingency Mode, you have the option to change all parameter values with exact numerical data. Of course, even in the case of Contingency Mode, all parameters are initially set to the default values stored in the default file. Any unchanged parameter will have the default value.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

8.4.5 Model Selection/Model Dependent Inputs

When you select this option (Selection E), a menu is presented showing the current chemical to be modeled and allowing you to choose one of three models

(1 = FIRE RADIATION, 2 = VAPOR DISPERSION, and 3 = EXPLOSION). Depending on the model selected, other forms will be displayed, allowing input of model-specific parameters.

Selection 1 (FIRE RADIATION)

When you select this model, a TARGET SPECIFICATIONS form is presented. You can enter the following parameters:

Target Height in feet or meters

Target Orientation as Horizontal, Vertical, or To Receive Max. Flux

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

Selection 2 (VAPOR DISPERSION)

When you select this model, you are returned to the CHANGE MODEL PARAMETERS menu. No model-specific input is required.

Selection 3 (EXPLOSION)

When you select this model, a menu entitled TYPE OF EXPLOSION is presented with two options (1 ... Tank Explosion, 2 ... Gas Cloud Explosion).

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

8.4.6 Source Inputs

When you select this option (Selection F), a menu entitled CHEMICAL RELEASE ENVIRONMENTS is presented with seven options as shown in Figure 8.9.

CHEMICAL RELEASE ENVIRONMENTS

1. Barge
2. Ship
3. Rail Tanker
4. Road Tanker
5. Fixed Storage Tank
6. Bullet Tank
7. Pipeline

Enter the Release Environment Number : _

Figure 8.9. Form for Selecting Chemical Release Environment

Selection of any of the Release Environment Numbers causes an environment-specific form to be displayed:

Selection 1 (Barge)

A form entitled RELEASE FROM A BARGE is displayed, allowing entry of several parameters specific to a barge release.

Tank Specifications:

Volume in gallons (U.S.), barrels, ft³, m³, liters, or gallons (U.K.)

Length in feet or meters

Width in feet or meters.

Distance from Tank Top to

Hole in feet or meters

Waterline in feet or meters.

Hole Size in inches or centimeters

Release Type as Instantaneous or Continuous

Depth of Chemical in feet or meters.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

Selection 2 (Ship)

A form entitled RELEASE FROM A SHIP is displayed, which allows entry of several parameters specific to a ship release. This form is identical to RELEASE FROM A BARGE, described previously, except for its title.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

Selection 3 (Rail Tanker)

A form entitled RELEASE FROM A RAIL TANKER is displayed, enabling you to enter several parameters specific to a rail tanker release.

Tank Volume in gallons (U.S.), barrels, ft³, m³, liters, or gallons (U.K.)

Tank Diameter in feet or meters

Hole size in inches or centimeters

Release Type as Instantaneous or Continuous

Hole Height from Tank Bottom in feet, meters, inches or centimeters.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

Selection 4 (Road Tanker)

A form entitled RELEASE FROM A ROAD TANKER is displayed, allowing entry of several parameters specific to a road tanker release. This form is identical to RELEASE FROM A RAIL TANKER, described above, except for its title.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

Selection 5 (Fixed Storage Tank)

A form entitled **RELEASE FROM A FIXED STORAGE TANK** is displayed. It allows entry of several parameters specific to a fixed storage tank release.

Tank Volume in gallons (U.S.), barrels, ft³, m³, liters, or gallons (U.K.)

Tank Diameter in feet or meters

Hole size in inches or centimeters

Release Type as Instantaneous or Continuous

Hole Height from Tank Bottom in feet, meters, inches or centimeters

Dike? (Y/N)

Dike Diameter in feet or meters.

On completion of this form, you are returned to the **CHANGE MODEL PARAMETERS** menu.

Selection 6 (Bullet Tank)

A form entitled **RELEASE FROM A BULLET TANK** is displayed, which allows entry of several parameters specific to a bullet tank release. This form is identical to **RELEASE FROM A FIXED STORAGE TANK**, described above, except for its title.

On completion of this form, you are returned to the **CHANGE MODEL PARAMETERS** menu.

Selection 7 (Pipeline)

A form entitled **RELEASE FROM A PIPELINE** is displayed, allowing you to enter several parameters specific to a pipeline release.

Pipe Diameter in feet, meters, inches or centimeters

Pipe Pressure in psi, inches of water, mm Hg, atm, bars or Pascal

Pipe Temperature in degrees Fahrenheit, Celsius, or Kelvin

Pipe flow rate in ft/s or m/s

Chemical State as gas or liquid

Hole size in inches or centimeters.

On completion of this form, you are returned to the CHANGE MODEL PARAMETERS menu.

8.4.7 Selection G (Output Specifications)

When you select this option, a form will be presented allowing you to specify certain preferences for output. The items which may have been specified are different for different models. If you have chosen the explosion model, the form will allow you to specify the units to be used for tabular reporting of hazard distances. You may select feet, meters, kilometers, or miles.

If you have chosen the fire radiation model, a form entitled HEAT FLUX SPECIFICATION is displayed which allows you to enter three heat flux levels to be reported on. Each level can be specified using units of kW/m^2 or $\text{BTUs}/\text{sq. ft.}$ A table of suggested values for certain uses is also provided for reference.

If you have chosen the vapor dispersion model, a form entitled DISPERSION OUTPUT SPECIFICATIONS is displayed. Here you can specify Threshold Limit Value (TLV) in units of 1000 ppm or Threshold Limit Value Time Weighted (TTLTWA) in ppm or Lower Flammability Limit (LFL) or OTHER in units of ppm, kg/m^3 , or mg/m^3 , as well as units for output displays (English - fps or Metric - mks).

8.4.8 Exit

On exit (Selection X), you will be prompted to change the case description.

- * **Case Description:** Enter up to 80 characters for a title to describe the case. It may be helpful to include the author's name and the date of creation or update. This title is displayed on the case file directory, which you can select from the CHEMS Models menu.

Table 8.2. Example Limit Values for Incident Radiant Flux

Off-Site Target Description	Incident Flux, kW/m^2
1) Outdoor areas occupied by 20 or more persons during normal use, such as beaches, playgrounds, outdoor theaters, and other recreation areas or other places of public assembly	5.0
2) Buildings that are used for residences or are occupied by 20 or more persons during normal use	12.6

3) Buildings made of cellulosic materials (or that are not fire-resistant or that do not provide durable shielding from thermal radiation) that:	12.6
i) have exceptional value, or contain objects of exceptional value, based on historic uniqueness described in federal, state, or local registers	
ii) contain explosive, flammable, or toxic materials in hazardous quantities	
iii) could result in additional hazard if exposed to high levels of thermal radiation	
4) Structures that are fire-resistant, provide durable shielding from thermal radiation, and have the characteristics described in subdivisions 3 (i) through 3 (iii)	21.1
5) Public streets, highways, and main lines of railroads	21.1
6) Other structures, or the property line of the facility	31.5

Source: Technology & Management Systems, Inc. (User Manual),
derived from 49 CFR Part 193

8.5 Select Spill Location

When you pick this option (SELECT SPILL LOCATION) from the CHEMS menu, IEMIS first checks to see you have displayed a base map. If you have not, you will be notified with the message base map not plotted. Press the <Return> key and you will be returned to the CHEMS menu. If you have a base map displayed, you can pick a spill location anywhere on the map by placing the cross-hairs over the point you wish to pick and pressing the space bar. The menu will then be redrawn, indicating the point you picked has been recorded.

When executed to locate the resulting contours, the spill location picked is then used by the models. Reselecting spill location after model execution will not relocate the contours because they are recorded as topographic overlays. It is necessary to reexecute the model to relocate a contour.

8.6 Change Case

CHEMS Model Text Menu: Select C Change Case Number

When you pick this option (CHANGE CASE) from the CHEMS Model menu, IBD displays the case selection form (example shown in Figure 8.10). The site name in the form refers to the current site. The case number shown for the model is the "current case."

```

CHEMS - Chemical Hazard Model      Case Selection Form  V2.00

Site Name: _____

CHEMS      Model Case:  _____
CHEMS      Model Base:  _____
  
```

Figure 8.10. Form for Changing Case Number

Use the arrow keys, <RETURN>, and <GOLD> <Z> to edit the data in the following fields of the form:

CHEMS Model Case: Enter one of 1000 possible case numbers from 000 to 999.

When you enter the CHEMS subsystem for the first time, this number will be 000, the number of the standard CHEMS model base case, which has no particular significance except that it contains an executable set of data inputs. If you enter a case number for which no input data exists, the system will create the new case based on the case listed in the "Model Based: field.

- * Model Base: If you want to create a new case, enter the number of an existing case that can be used as the basis for creating another CHEMS case.
- * Site Name: Your current site name - for reference only.
Press <GOLD> <Z> to accept the data and exit the form.

8.7 Display Base Map - Graphic Menu Only

This option produces a menu of standard graphics functions for IEMIS map display.

Each of the CHEMS models (Explosion, Fire, and Dispersion) produces its own topographic overlays of polygonal output. These topographies must be included in your site's map input subdirectory version of SETUP.COM and USRTOPO.FIL so they will be selected as topographies for display within the Base Map Display Select Topography submenu. For further information on topographies, refer to the *IBS Database Guide*.

Each of the contours placed in these topographies by the CHEMS models contains adequate attributes to describe the contours and distinguish them from other graphic objects.

Note that if you do not select a given model's topographic output from the Select Topographics menu you will not see the model output graphics on your map display. For a complete description of map display functions, refer to the *IBS Utilities Guide*, MPDISPLAY.

8.8 Execute Model Inter or Execute Model Batch

CHEMS Model Text Menu: Select A (Execute Current CHEMSHAZ Case)

Graphic Menu: Execute Model INTER
 Execute Model BATCH

The graphic menu options execute the model for the current case as a interactive subprocess or a batch job, respectively. The text menu option (Execute Current CHEMSHAZ Case) provides a menu allowing you to select either interactive or batch process execution of the current model case.

— Executing the Model

Model execution for the current CHEMS case is automatic.

Error Messages. If problems are encountered during execution of the CHEMS model, your terminal may display diagnostic error messages. For example, if the target is located within the plume of the fire for a specific hazard distance calculation, you will see an error message to that effect.

8.9 Print/Display Reports

CHEMS Model Text Menu: Select E (Print/Display Reports)

Graphic Menu: DISPLAY/PRINT REPORTS

When you select this option, the system prompts you to choose among options for printing the results of the current CHEMS Model case or displaying them as a report on your terminal. As shown in Figure 8.11, you have five options that are selected by pressing the appropriate key.

Print Display Menu	
Selection Designator	Selection
A	Print Current Model Text Output
B	Screen Display Current Model Text Output
C	Print Chemical Property Data
D	Screen Display Chemical Property Data
X	Exit
Please Enter Selection Designator _	

Figure 8.11. Menu for IDYNEV Print/Display Reports Option

8.10 Perform File Maintenance

Chems Model Text Menu: Select D (CHEMSHAZ Case Directory and Optional Delete)

Graphic Menu: **PERFORM FILE MAINT**

When you select **PERFORM FILE MAINT.** from the **CHEMS Model** menu, IEMIS displays the menu of file maintenance functions shown in Figure 8.12. The menu options allow you to list the **CHEMS Model** cases, delete case outputs, or delete the cases by deleting both inputs and outputs.

PERFORM FILE MAINTENANCE

Selection Designator	Selection
A	Display Directory of Cases
B	Delete Case Output for a Specified Case
C	Delete Input and Output for a Specified Case
X	Exit to Previous Menu

Enter selection designator:

Figure 8.12. Menu for Performing File Maintenance

Section 9

Model Library - Integrated Modeling (EESF)

IBSSH Menu: Select Models (Graphic) or Models (Non-Graphic)
Models Menu: Select EESF (Integrated Model)

This section describes the integrated modeling capabilities of the Exercise Evaluation and Simulation Facility (EESF). EESF is a computer-based system developed to improve FEMA's capability for evaluating radiological emergency plans, exercises, and preparedness near nuclear power plants. A key feature of EESF is the capability for concurrent execution of the IBS meteorology/dose model (MESORAD) and evacuation model (IDYNEV). EESF also features concurrent display of the map-related results of the two models.

This section covers the following topics.

- 9.1 Description of EESF
- 9.2 Using EESF
- 9.3 Obtaining On-Line HELP with Data Input
- 9.4 Starting an EESF Work Session
- 9.5 Obtaining and Using the Top-Level EESF Menu
- 9.6 Display Base Map - Graphics Menu Only
- 9.7 Modify Meteorology/Dose Model Input Data
- 9.10 Modify Evacuation Start Times for Dose Calculations
- 9.13 Print Reports
- 9.14 Display Reports - On Terminal
- 9.15 Display Map-Related Graphics - Graphics Menu Only
- 9.16 Case File Directory and Optional Delete
- 9.17 Change Case Numbers
- 9.18 Concurrent Execution of MESORAD and IDYNEV
- 9.19 If You Have Problems
- 9.20 Status and Diagnostic Messages
- 9.21 References

9.1 Description of EESF

The Exercise Evaluation and Simulation Facility (EESF) is an integration of computer-based tools for simulating and reporting the events of a radiological emergency: evacuation of populations, wind and weather transport of effluents released to the atmosphere, and resulting radiological doses. EESF is essentially a combination of two IBS models: the Interactive Dynamic Evacuation Model (IDYNEV) and the meteorology/dose model (MESORAD). EESF combines these computer modeling capabilities with map information, interactive color graphics displays, and a simple user interface to form a powerful system for evaluating emergency preparedness.

EESF can be used to

- Generate scenarios for emergency preparedness exercises around commercial nuclear power plants
- Model radiological releases, atmospheric dispersion, and dose levels according to test plan objectives
- Model diffused toxic gases (no gravity effects) to estimate concentrations at distances from the release point
- Evaluate the timing of emergency responses made during exercises
- Evaluate the dose effects of evacuation or sheltering decisions made during exercises
- Evaluate the feasibility and acceptability of evacuation plans
- Simulate the progress of radiological emergency scenarios for use in emergency management training.

EESF produces tailored results from three computer models: an evacuation model, a meteorology model, and a dose model. The meteorology and dose models are integrated in a single meteorology/dose (met/dose) package. Model results can be viewed and printed as text, or displayed graphically as overlays on maps derived from the national map database. Modeled cases that specify the site, the type of release, and the conditions of the release can be saved for further reference or modification. More detailed information on these Models can be found in Section 4 (MESORAD Model) and Section 6 (Dynamic Evacuation Model).

The following sections supply summary background on the evacuation model, the meteorology/dose model, and the display of map-related model results.

9.1.1 Evacuation Model (IDYNEV)

The Interactive DYnamic Network EVacuation (IDYNEV) model predicts the progress of area evacuations by indicating the changing conditions of population movement over a traffic network. The model produces several measures of evacuation effectiveness, including travel times, vehicle counts, queues and delays, and person throughput over the links (street segments) and nodes (intersections) of the network. Many model results can be viewed; some results can serve as input data to other models, such as the dose model.

IBS enables easy data input to the evacuation model, either by completion of data input forms or by graphic manipulation of some parameters on a map display (see Section 9.4).

9.1.2 Meteorology/Dose Model (MESORAD/FEMA)

The meteorology and dose models in EESF are derived from a computer code called MESORAD. MESORAD is an extension of MESOI, an atmospheric dispersion model that includes deposition and decay of radiological materials. The MESOI model was enhanced with dose modeling logic to create MESORAD. The implementation of MESORAD in IBS combines a subset of MESORAD capabilities with added capabilities for creating special dose reports and for calculating doses that are due to ingestion pathways. This manual refers to the IBS implementation of MESORAD as MESORAD/FEMA (or only as MESORAD).

Generally, this manual describes the use of these models without explaining their theoretical bases. Meteorologists and other technical experts concerned with creation of specific cases can refer to the source documents for detailed explanations of the models (see Section 9.21, **References**). Section 4.15 is a reference discussion of IBS modifications to the original MESORAD model.

— Meteorology Model

The meteorology portion of MESORAD estimates the transport, diffusion, deposition, and decay of radioactive materials released to the atmosphere. The model is capable of treating simultaneous releases from as many as four different elevations at the point of release (site center). Puffs generated at the release points are affected by a three-dimensional, horizontal wind field that takes into account the topology of the area. The concentrations within the released puffs are initially assumed to be distributed normally in both the horizontal and vertical directions. Horizontal concentration distribution is modified by the wind field. Vertical concentration distribution is modified by assuming reflection at the ground and at the top of the atmospheric mixing layer.

Inputs to the model include parameters such as time intervals, atmospheric stability, wind speeds and directions, height of the mixing layer, surface and upper layer temperatures, dose check point locations, terrain and topographic features,

and locations of weather stations. Section 4.6 describes data input forms for modifying input to the met/dose model.

The meteorology model calculates an integrated, two-dimensional wind field in the simulation area and creates a file defining the concentration and deposition fields over time. This file is subsequently used by the dose part of the model.

— Dose Model

The dose model calculates radiological doses simultaneously with the atmospheric dispersion and depletion calculations for as many as 50 radionuclides. Dose calculations include current and cumulative values for external doses to the whole body and for internal doses to critical organs (lungs, thyroid) and the whole body.

The model interacts with the evacuation model so that the options of evacuation and sheltering can be compared. Sheltering factors for portions of the population can be entered, and the model will account for exposures during the evacuation.

EESF provides an interactive graphic display environment for viewing the results both during and after model execution. Section 9.18 describes the concurrent execution and display of results from the EESF models.

9.1.3 Display Map-Related Modeling Results

EESF calls on the IBS map database and Map Display software for graphic display of modeling results overlaid on background maps. For example, you can display met/dose model results over the area for which the model was run. You can specify what static features to display: political boundaries, waterways, etc. Then you can customize the model results to be displayed by selecting dose type and other information such as the windfield and the path of the effluent plume(s). This interactive graphics display is a powerful tool for understanding the results of the technical models.

9.2 Using EESF

The materials in this section are intended for any user of the EESF. The section includes the steps you must take in using EESF to simulate radiation release scenarios that might be used in emergency response exercises and training. EESF is designed to step you through the creation of these scenarios.

EESF Functions. You can use this section as a step-by-step guide through the EESF functions. After you have selected EESF from the Model Library menu, EESF can supply background information with its HELP features (described in Section 9.3). Section 9.21 lists documentation references for the models used in EESF.

Meaningful Use of EESF. Although any user can use the computer models contained in EESF, the model results will be only as meaningful as the input. Some static details, such as the physical characteristics of a site, would rarely be changed by a typical user. Other inputs would require the technical expertise of a meteorologist or traffic engineer. For example, consultation with a traffic engineer is suggested for specifying road network and traffic parameters. Generally, a system manager and technical consultants will set up standard cases the typical user can modify in less fundamental ways.

Use of Computer Memory and Resources. EESF's concurrent modeling capabilities require a good portion of your computer's resources: both models plus the graphics display software must be loaded into memory. Other users of the system can expect to see the system run more slowly, especially if you are running large cases. If you are mostly interested in the concurrent display of results, it may be more efficient to run the models independently (perhaps in batch mode during off hours) and then use the EESF display options to depict the results after the models have concluded.

Graphic Mapping Options. The EESF displays of map graphics are based on the IBS mapping functions described in the *IBS Utilities Guide*, MPDISPLAY.

9.3 Obtaining On-Line HELP with Data Input

If you have a question about a particular item on a data input form, you can press the HELP key <F6> above the numeric keypad.

- Pressing <HELP> the **first** time produces help information pertaining to the **current input field**.
- Pressing <HELP> a **second** time produces help information pertaining to the **entire** data form or menu.

In both cases, more than one page (screenful) of help information may be presented. Press <RETURN> to page through the available information and get back to the data form or menu.

9.4 Starting an EESF Work Session

Models Menu: Select EESF (Integrated Model)

An EESF work session is a continuous period of time during which you use EESF. The session starts when you select EESF from the IBSSH Model menu and ends when you return to the IBSSH Model menu.

9.4.1 Selecting Graphic or Text Mode

- If you select EESF from the models (Graphic) menu, the succeeding menus (for selecting EESF functions or for modifying IDYNEV input data) will include options for graphically displaying (and modifying) site and modeling information.
- If you select EESF from the models menu, the succeeding menus will NOT include any graphic options. Thus, a subset of EESF tasks could be accomplished on a nongraphic terminal.

9.5 Obtaining and Using the Top-Level EESF Menu

When you select EESF from the Models menu, you will be turned over to one of the EESF top-level menus shown in Figures 9.2 and 9.3. Sections 9.5 through 9.18 explain how to use the various functions that may be requested from the EESF top-level menu(s).

The text mode menu in Figure 9.3 contains the basic IDYNEV and MESORAD execution functions. The graphics mode menu in Figure 9.2 includes graphics functions: Display Base Map, Display Map-Related Graphics, and Concurrent Execution of MESORAD and IDYNEV. This last feature enables the concurrent graphic display of MESORAD and IDYNEV results at the end of coinciding time steps/periods.

Both menus include a display of the model case numbers and the site name (the Maine Yankee test site in these examples).

To select a function (from either menu): Enter the function designator (letter) of the desired function and press <RETURN>.

EESF		
Function Designator	Function	Explained in Section
A	Display Base Map	9.6
B	Modify Meteorological/Dose Model Input Data	9.7
E	Modify Evacuation Start Time for Dose Calculations	9.9
H	Print Reports	9.13
I	Display Reports	9.14
J	Display Map-Related Graphics	9.15
K	Case File Directory and Optional Delete	9.16
L	Change Case Numbers	9.17
M	Concurrent Execution of MESORAD And IDYNEV	9.18
X	Exit	
Evacuation Model Case Number: <u>101</u> Based on: <u>0</u>		
Met/Dose Model Case Number: <u>817</u> Based on: <u>0</u>		
Site Name: <u>MAINE YANKEE</u>		
Enter Function Designator Corresponding to Function You Wish to Execute. <u> </u>		

Figure 9.2. EESF Top-Level Menu - Graphic Mode

```

                                EESF

Function
Designator   Function

A           Modify Meteorological/Dose Model Input Data
B           Model Evacuation - Batch
C           Model Evacuation - Interactive
D           Modify Evacuation Start Time for Dose Calculations
E           Execute MESORAD - Batch
F           Execute MESORAD - Interactive
G           Print Reports
H           Display Reports
I           Case File Directory and Optional Delete
J           Change Case Number
X           Exit

Evacuation Model Case Number:101   Based on: 0
Met/dose   Model Case Number:817   Based on: 0
Site Name:MAINE YANKEE

Enter Function Designator Corresponding to Function You Wish to Execute. _

```

Figure 9.3. EESF Top-Level Menu - Text Mode

It is normal to begin a session by selecting the case number before selecting any of the other listed options. This avoids inadvertent creation of new cases. See Section 9.17, **Change Case Numbers**, for a discussion of site names, case numbers, and their data files.

After making sure that you are working with the correct site and case data, you can select any of the other MESORAD, IDYNEV, display, or concurrent functions. The functions of the EESF top-level menu are explained in Sections 9.6 through 9.18.

9.6 Display Base Map - Graphics Menu Only

EESF Top-Level Menu: Select A (*graphic mode menu*)

This option produces the standard IBS menu of Map Display functions. For a complete description of these functions, refer to the *IBS Utilities Guide*, MPDISPLAY.

9.7 Modify Meteorology/Dose Model Input Data

EESF Top-Level Menu: Select A (*text mode menu*) or B (*graphic mode menu*)

When you select this function, IEMIS activates a second menu for selecting data forms for updating MESORAD case input. This second menu can be either a graphic menu or a text menu, depending on which option you selected when starting EESF. The example in Figure 9.4 is the text mode menu.

To select the data input form that you want to inspect or change, pick the appropriate item on the form selection menu. For a complete discussion of using the data input forms to modify Mesorad model input, including certain graphics functions not shown in Figure 9.4. Please also refer to Section 4.6, *Update Case Data* in the MESORAD Model section.

Note the option to Change Output Options. This option enables you to limit the types of output that will be produced when the current MESORAD case is executed. Limiting the output may reduce the time needed to execute the model or reduce the disk space needed for output reports.

Choosing Exit on the form selection menu will return you to the EESF top-level menu.

Meteorological/Dose Model Parameter Change

The change option menu is as follows:

- A Change Control Parameters
- B Change Source Details
- C Change Weather Data
- D Change Checkpoints
- E Change Station Definitions
- F Change Terrain
- G Change Cartesian Grid
- H Change Polar Grid
- I Display weather Directory
- J Change Output Options
- X Exit

Enter option selection: ____

Figure 9.4. Menu for Selecting Forms for Changing Parameters in the Meteorological/Dose Model - Text Mode

9.8 Model Evacuation - Batch

EESF Top-Level Menu: Select B (*text mode menu*).

This option displays a check list for the current IDYNEV case (case number, site name, run type, etc.) and the following prompt:

Execute IDYNEV for this case? (Y/N) [N] = = >

Read the checklist to be sure the case is set up to run as you wish. Then respond to the prompt shown above:

- If you answer N (no), you will return to the EESF menu without executing the model.
- If you answer Y (yes), the model will execute in batch mode, using data defined for the current site and IDYNEV case number.

Batch execution of the model is convenient for larger cases. In batch mode, IBS executes the model separate from your current interactions with IBS. While the model is running in batch mode, the system returns control to the EESF top-level menu. You can then use other EESF functions, or even exit. However, do NOT select any IDYNEV interactions while the model is running: these could interfere with the model run.

Refer to Section 6 and to the IDYNEV source documentation (listed in Section 9.21) for specific information on the evacuation model.

9.9 Model Evacuation - Interactive

EESF Top-Level Menu: Select C (*text mode menu*).

This option displays a checklist for the current IDYNEV case (case number, site name, run type, etc.) and the following prompt:

Execute IDYNEV for this case? (Y/N) [N] = = >

Read the checklist to be sure that the case is set up to run as you wish. Then respond to this prompt:

- If you answer N (no), you will return to the EESF menu without executing the model.
- If you answer Y (yes), the model will execute in interactive mode, using data defined for the current site and IDYNEV case number.

In interactive mode, IBS executes the model as part of your current process; you will have to wait until the model is finished before you can continue any further operations. When the model finishes running, the system will inform you that IDYNEV execution is complete and return control to the EESF top-level menu. For large cases, execution in batch mode (Section 9.8) may be more convenient.

Stopping the Model. The evacuation model cannot be cleanly halted while it is running. Pressing <CTRL Y> is a standard system interrupt command and will halt model execution; however no guarantee is presented as to where you will regain control (probably at the IBS main menu). The model can be restarted only from the beginning of its execution.

Summary Report. If the model is executed for a traffic simulation, and if one or more time steps of the simulation are completed, EESF displays a summary report of the evacuation following model completion. This report contains various statistics extracted from the last simulation time period reported by the evacuation model. Figure 9.5 is an example of this report.

```

                DYNEV   SUBNETWORK STATISTICS
VEHICLE-MILES =      247221.75   VEHICLE-MINUTES =      1480384.88
VEHICLE-TRIPS(EST.) =    32432
PCT OF VEHs THAT STOPPED = 21.747   MOVING/TOTAL TRIP TIME = 0.207
AVG. SPEED(MPH) = 10.02
AVG. QUEUE CONTENT = 3769.5 VEH.   AVG DELAY/VEH = 268.06 SEC.
TOTAL DELAY =      1173755.6 MIN.
DELAY/VEH-MILE =  4.75 MIN/V-MILE   TRAVEL TIME/VEH-MILE = 5.99 MIN/V-MILE
IDYNEV COMPLETED PROCESSING THRU 4 37 30, Press <Return> to continue

```

Figure 9.5. Example of an Evacuation Summary Report

9.10 Modify Evacuation Start Time for Dose Calculations

EESF Top-Level Menu: Select D (*text mode menu*) or C (*graphic mode menu*)

If the evacuation model is being run in conjunction with a simulated radiological release, this option enables you to consider some of the effects of the evacuation on dose calculations. When you choose this function, you will see the input form shown in Figure 9.6. This form enables you to specify 1) an **evacuation offset time** (a period between the start of the release scenario and the start of the evacuation) and 2) **sheltering factors** to account for the protection afforded by vehicles and other shelter during an evacuation.

```
Enter evacuation offset time (99 99) for static population: 0 0
Apply shelter factors (Y or N): _
Vehicle Shelter Factor: 0.00
Sector Shelter Factor: 0.00
```

Figure 9.6. Input for Modifying Evacuation Start Times and Applying Shelter Factors

The input fields of this form are explained below.

- * Evacuation offset time: Enter a number of hours (0 through 24) in the first field and a number of minutes (0 through 60) in the second field.

For example: 01 09 indicates an offset of 1 hour plus 9 minutes.

This offset time is the period between the start of the release scenario and the start of the evacuation. As indicated by the prompt, an entry of 99 99 means that the evacuation never started as far as the dose calculations are concerned: the population never moved from their shelters. Negative values (indicating the evacuation started before the start of the release) are NOT supported.

- * Apply shelter factors (Y or N): Enter Y to include the following shelter factors in the dose calculations, or enter N to ignore the following shelter factors.
- * Vehicle Shelter Factor: Enter a factor (0. to 1.) to indicate the fraction of the dose that was NOT received because of sheltering by vehicles. A 0 zero signifies NO sheltering; 1 signifies total sheltering.

9.11 Execute MESORAD - Batch

EESF Top-Level Menu: Select E (*text mode menu*).

This option displays a check list for the current MESORAD case (site name, case numbers, simulation start and end times, etc.) and the following prompt:

Execute MESORAD for this case? (Y/N? [N]) = >

Read the checklist to be sure the case is set up to run as you wish. Then respond to the prompt shown above:

- If you answer N (no), you will return to the EESF menu without executing the model.
- If you answer Y (yes), the model will execute in batch mode, using data defined for the current site and met/dose case number.

Batch execution of the model is convenient for larger cases. In batch mode, the system executes the model separate from your current interactions with EESF. While the model is running in batch mode, the system returns control to the EESF top-level menu. You can then use other EESF functions, or even exit the system. Do NOT try to change MESORAD input data while the model is running: errors can result.

Refer to Section 4 and to the MESORAD source documentation (listed in Section 9.21) for specific information on the met/dose model.

9.12 Execute MESORAD - Interactive

EESF Top-Level Menu: Select F (*text mode menu*).

This option displays a checklist for the current MESORAD case (site name, case numbers, simulation start and end times, etc.) and the following prompt:

Execute MESORAD for this case? (Y/N) [N] = >

Read the check list to be sure the case is set up to run as you wish. Then respond to the prompt shown above:

- If you answer N (no), you will return to the EESF menu without executing the model.
- If you answer Y (yes), the model will execute in interactive mode, using data defined for the current site and met/dose case number.

In interactive mode, the system executes the model as part of your current process but still returns control to the EESF top-level menu. You can then use other EESF functions, but do NOT try to change MESORAD input data while the model is running: errors can result. When the model is through running, the system will inform you that IDYNEV execution is complete. For large cases, execution in batch mode (Section 9.11) may be more convenient.

Refer to Section 4 and to the MESORAD source documentation (listed in Section 9.21) for specific information on the met/dose model.

9.13 Print Reports

EESF Top-Level Menu: Select G (*text mode menu*) or D (*graphic mode menu*)

When you select Print Reports, the system displays six options for which met/dose or evacuation data reports to print, as shown in Figure 9.7. (Refer also to Section 9.14, **Display Reports**.)

- * **Met/Dose Case Debug Output:** This first option is of interest only to the very experienced user or programmer and will tie up the printer for some time printing tables of figures. The report describes the current input to the met/dose model and output for the model time steps.
- * **Evacuation Report:** This option prints a large report on the modeled evacuation. This is the only report for the evacuation model, IDYNEV. The output is best suited for a high-speed line printer. You may wish to display this output without printing it (Section 9.14).
- * **Dose Report:** This third option prints the standard MESORAD dose report, usually 3 to 10 pages.
- * **PAG Dose Summary Report:** This option prints a report that relates the modeled doses to Protective Action Guides (PAG) established by various government agencies. PAG are recommended dose commitment levels that require some sort of protective action. The report indicates the grid sector and time at which a PAG was exceeded during a met/dose run. The report also indicates the distance from the release point, total doses, and the active nuclides from the sources.
- * **Ingestion Pathway Report:** This option prints a report that shows the whole body (WB) and thyroid (Thy) dose commitments expected through ingestion of vegetables, meat, and milk. The doses are also related to PAG. The report indicates the distance from the release point at which emergency and preventative PAG were exceeded because of ingestion of radioactive materials.
- * **Field Team Summary Release Report:** This option prints tabular reports that relate field readings of iodine concentrations, gamma dose rates, and gamma + beta dose rates to isopleth numbers for different time periods.

For more information on the PAG dose summary report, ingestion pathway report and the field team summary release report, refer to Section 4.16 on MESORAD/FEMA.

To select Print Report options: Type Y in the input field of each report

that you want to print. Press <GOLD> <Z> to print the selected reports or press <GOLD> <Y> to escape to the EESF menu. You will then be prompted as to whether you wish to Print, Display, or Exit.

Figure 9.8 is an example of EESF (MESORAD) printed output.

EESF Report Options	
(N)	Met/Dose Case Debug Output
(N)	Evacuation Report
(N)	Dose Report
(N)	PAG Dose Summary Report
(N)	Ingestion Pathway Report
(N)	Field Team Summary Release Report

Figure 9.7. Menu for EESF Print Reports (and Display Reports) Options

MAINE YANKEE MULTIPLUME WITH 10:00 EVACUATION				
20-OCT-86 12:15:20				
DOSE TO POPULATION				
Time of Day at End of 15 minutes Period	CURRENT 15 MINUTE PERIOD		CUMULATIVE	
	THYROID REM	WHOLE BODY REM	THYROID REM	WHOLE BODY REM
20-OCT-86 10:15	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 10:30	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 10:45	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11: 0	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:15	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:30	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 11:45	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 12: 0	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
20-OCT-86 12:15	1.30178	0.03212	1.30178	0.03212
20-OCT-86 12:30	39.3930	0.94264	40.6948	0.97477
20-OCT-86 12:45	70.7735	1.66823	111.468	2.64300

Figure 9.8. Example Page of EESF Dose Report Printed Output

9.14 Display Reports - On Terminal

EESF Top-Level Menu: Select H (*text mode menu*) or E (*graphic mode menu*)

When you select Display Reports, IEMIS displays six options for which met/dose or evacuation data reports to display on your terminal screen, as shown in Figure 9.7. The report options are identical to those for printing reports. (For descriptions of the reports, refer to Section 9.13, **Print Reports**.)

Note: The Evacuation Report (the only report of the evacuation model) is designed for a high-speed line printer, so it may be too wide to display usefully.

To select Display Report options: Type Y in the input field of each report that you want to display on your screen. Press <GOLD> <Z> to display the selected reports or press <GOLD> <Y> to escape to the EESF menu.

The system will prompt you to choose between options for printing or displaying the report(s). Select an option (Figure 9.9) by pressing the appropriate key:

<u>Key Press</u>	<u>Function</u>
P	Print file on printer
D	Display file on terminal
X	Exit

Figure 9.9 Print Options Menu

Figure 9.10 is an example of EESF (MESORAD) displayed output.

MAINE YANKEE MULTIPLUME CASE WITH 10:00 EVACUATION
20-OCT-86 12:15:20

ALL TIMES THAT DO NOT APPEAR HAVE VALUES OF ZERO

DOSE AT CHECKPOINT BOOTHBAY

Time of Day at End of 15 minute Period	CURRENT 15 MINUTE TIME PERIOD		
	I-131 micro Ci/cc	PLUME mrem/hr	GROUND mrem/hr
20-OCT-86 13:15	1.169146E-07	.320722	.016055
20-OCT-86 14: 0	2.421295E-05	62.2961	3.25024
20-OCT-86 14:15	2.745193E-04	666.396	36.0906
20-OCT-86 14:30	7.320443E-04	1690.78	94.5235
20-OCT-86 14:45	9.793648E-04	2169.62	124.548
20-OCT-86 15: 0	9.279536E-04	1971.88	116.196
20-OCT-86 15:15	4.346244E-04	882.184	53.4811
20-OCT-86 15:30	1.109445E-04	215.269	13.4149
20-OCT-86 15:45	1.782245E-05	33.0999	2.11789

Figure 9.10. Example Page of EESF Dose Report Displayed Output

9.15 Display Map-Related Graphics - Graphics Menu Only

EESF Top-Level Menu: Select F (*graphic mode menu*)

This option enables you to graphically display the results of the met/dose and evacuation models on the same screen:

- Evacuation Graphics. You can display evacuation graphics only after the evacuation model has executed and produced output.
- Met/Dose Graphics. You can display met/dose graphics only after the met/dose model runs.

Picking DISPLAY MAP-RELATED GRAPHICS from the EESF graphic-mode menu will display a menu of EESF Graphical Mapping Options (Figure 9.11) for choosing output. The menu offers a choice of what static and dynamic mapping features can be displayed. (For a summary of how to use this and related menus to display model results, refer to Displaying Map-Related Graphics, on the page following this menu description.)

```

EESF Graphical Mapping Options

_ Specify Limits for Graph Data Selection ?

Dynamic Features:                                Curr Cumul:
- Windfield                                     - - Relative Plume Concentration
- Initial Particle Path                         - - Relative Deposition
- Traffic Data                                 - - Dose - Total Whole Body
                                                - - Dose - Thyroid
Grids:                                           - - Dose - Inhalation/Whole Body
- 10 Mile EPZ (Polar)                          - - Dose - Deposition
- 16x16 Meteorolog.(Square )                   - - Dose - Overhead Plume
- 20x20 Evacuation (Square )                   - - Dose - Lung
- 31x31 Dose (Square )                         - - Iodine
- 3x36 Dose                                     - - Gamma
                                                - - Gamma + Beta

Static Features:
_ Link-Node diagram
  
```

Press GOLD <F5> followed by 'Y' to return to EESF menu

Figure 9.11. Menu for EESF Graphical Mapping Options

The principal types of input fields for selecting items for display are described here:

- * Specify Limits for Graph Data Selection: Entering a Y in this field will cause a form of MESORAD Global Graphics Parameters to appear after you finish the current menu. The use of this form is explained in the MESORAD section, Section 4.4, where the form appears as Figure 4.4.
- * Dynamic Features: Select a maximum of two dynamic features by entering Y in the entry field in front of the feature name. Select Traffic Data for viewing evacuation results.
- * Grids: Select any of the grids by entering a Y in the entry field in front of the grid name. The several types of grids are static graphic overlays and do not change, after they are drawn.
- * Static Features: A Y in the Link Node Diagram field selects a representation of the links and nodes of the evacuation traffic network.
- * Curr(ent) or Cumul(ative statistics): Select a maximum of two types of MESORAD output statistics for display and reporting by entering a Y in one or both of the two entry fields in front of each item. Curr(ent) refers to the current time step; Cumul(ative) refers to totals since the start.

Displaying EESF Map-Related Graphics. To display map-related graphics:

1. Select the appropriate options on the EESF Graphical Mapping Options menu (Figure 9.11) and then press <GOLD> <Z>.

EESF then calls on the IBS Map Display software so that you can create a background over which the model results will be displayed. This occurs only once during a graphics display session: if you have already selected map background features, you skip to Step 4. (The *IBS Utilities Guide*, MPDISPLAY, explains in detail how to use the map display software and graphic input devices.)

2. Use the SELECT TOPOGRAPHIES function on the Map Display menu to choose the topographic features for your background map.
3. Exit the topography selection and Map Display menus.

IBS then reads the necessary model output files, draws the map background features, and presents the EESF graphics display menu shown in Figure 9.12.

Note: If you do not select Traffic Data on the EESF Graphical Mapping Options menu, the system displays the regular MESORAD graphics display menu instead of the menu shown in Figure 9.12.

<pre> Site Met/Evac ###/### EXIT HELP CONTINUE DESCRIBE NODE DESCRIBE LINK DESCRIBE EXIT NODE DESCRIBE INT. CENTROID DISPLAY FLOOD WATER RETURN POPULATION DETERMINE AREA ZOOM IN ZOOM OUT RUN EVAC OUTPUT (EO) RESTART EO STEP EO FORWARD STEP EO BACKWARD SELECT EO TIME STEP REFRESH EO NETWORK ## HOURS, ## MINUTES TEXT DISPLAY ENABLED ICON DISPLAY ENABLED STEP MET/DOSE OUTPUT REPORT DOSE DISPLAY LEGEND DD/MMM/YY HH:MM </pre>	<pre> <-- Site and Met/Evac case number identification <-- IDYNEV Time Period label appears when model output results are run or stepped. <-- Start date and time step time display for MESORAD </pre>
--	---

Figure 9.12. EESF Graphics Display Menu

4. Use the graphics input device attached to your workstation (the joydisk on your keyboard, or a mouse) in selecting options on the EESF graphics display menu.
5. After you have displayed the desired model results for the current case, pick EXIT on the EESF graphics display menu to return to the EESF Graphical Mapping Options menu.
6. Repeat Steps 1 through 5 with different options and parameters, or press <GOLD> <Y> to return to the main EESF menu.

The following descriptions of functions on the EESF graphics display menu assume that you have picked the menu item. Some functions may require pressing the <DIALOG> key to view text information.

- **EXIT:** This option exits the EESF graphics display menu and returns to the EESF Graphical Mapping Options menu.

- **HELP:** This option displays information about the menu.
- **CONTINUE:** This item is used to continue or finish another operation.
- **DESCRIBE NODE:** Use the graphics cursor to pick a node on the traffic network (the selected node will blink); and a screen report will display. This option will display signal interval durations and node numbers of approaching links.
- **DESCRIBE LINK:** Pick a link on the network (the selected link will blink) and a screen report will display.

This option will display link information, including Level of Service (LOS) codes that indicate traffic conditions from free-flow (code A) to unmoving (code F). The link information is part of a set of dynamic statistics that are accumulated as the evacuation model steps through its execution. To display link information, the model output must first be stepped at least through the first time step. (See the RUN EVAC OUTPUT (EO), STEP EO FORWARD, and STEP EO BACKWARD functions.)

- **DESCRIBE EXIT NODE:** Pick an exit node on the evacuation boundary (the selected node will blink).

Exit nodes are described by an associated internal node number.

- **DESCRIBE INTERNAL CENTROID:** Pick an internal centroid that points into a link of the network (the selected centroid will blink).

This option displays a report on the centroid, including the time periods, associated traffic flow rates, and upstream and downstream nodes.

- **DISPLAY FLOOD WATER:** This option will display flood water contours if you have SLOSH flood data available. (For information on SLOSH flood data, see Section 5.3.1 of Section 5, **Generate Evacuation Network**).
- **RETURN POPULATION:** This option will display the population associated with a selected area -- 1) Outline an area on the screen by picking points that define the sides of a polygon around the desired region. 2) Pick CONTINUE to draw the final side of the polygon and return a population value.
- **DETERMINE AREA:** This option will display the area within a polygon that you draw on the map display -- 1) Outline an area on the screen by picking points that define the sides of a polygon around the desired region. 2) Pick CONTINUE to draw the final side of the polygon and return an area value and circumference.

- **ZOOM IN:** 1) Pick a point at the center of an area that you want to zoom in on (enlarge). 2) Pick a second point that defines the edge of a square with the first point at the square's center.

The square area defined will rescale to fill the map display grid.

Note: Because of the way that the MESORAD dynamic plume data are drawn, the ZOOM IN, ZOOM OUT, and REFRESH EO NETWORK functions may cause all MESORAD dynamic animation plumes to be displayed instead of just the current one. To avoid this problem, use these functions only BEFORE selecting STEP MET/DOSE OUTPUT.

- **ZOOM OUT:** 1) Pick a point at the center of an area that you want to zoom out from (scale down). 2) Pick a second point that defines the edge of the square with the first point at the square's center.

The display area will be scaled down to the size of the defined square with the display centered at the first point chosen.

- **RUN EVAC OUTPUT (EO):** This option displays the model output, showing changes in the traffic network as the output steps from the first time step to the last.

Red bars on the network links (street segments) indicate the extent of traffic line ups near nodes (intersections). These bars have a constant width, regardless of the scale of the map display.

- **RESTART EO:** This option resets the evacuation model output to the start of the simulation.
- **STEP EO FORWARD:** This option displays the traffic network results for the next sequential time step. Cumulative network statistics are available at each time step.
- **STEP EO BACKWARD:** This option displays the traffic network results for the previous time step.
- **SELECT EO TIME STEP:** Type a time step number in response to the prompt, and press <RETURN>.

This option enables you to reset the model time step for displaying results.

- **REFRESH EO NETWORK:** This options redraws the evacuation link-node diagram, which may be overdrawn by other display features. (Also, see the note at ZOOM IN.)

- Evacuation Time Periods (display only): As the evacuation model output is run or stepped, the current time since the evacuation start is displayed at this position in the menu area.
- TEXT DISPLAY ENABLED/DISABLED: This option enables/disables the display of map-related text if any such text is associated with the map data.
- ICON DISPLAY ENABLED/DISABLED: This option enables/disables the display of map icons (constant-size graphic figures that represent the location of items on the map).
- STEP MET/DOSE OUTPUT: This option displays MESORAD model results one step at a time (that is, for the next sequential time step).
- REPORT DOSE: This option enables you to pick a point on the map display, pick continue, and return dose report output associated with that location.

Dose report output for a point may be more than one screen of text. Be prepared to use <CTRL S> and <CTRL Q> to stop and restart the text as it scrolls by on your screen.

- DISPLAY LEGEND: This option momentarily clears the menu area and writes "map legend" information about the current model results and other map display items. After a few seconds, this legend information is erased and replaced with the menu options.
- Met/Dose Model Start Date and Time Step (display only): As the met/dose model output is run or stepped, the model start date and current time step is displayed at this position in the menu area. When this is done, you will also see the scale of the map display briefly displayed.

9.16 Case File Directory and Optional Delete

EESF Top-Level Menu: Select I (*text mode menu*) or G (*graphic mode menu*)

This option will display the menu shown in Figure 9.13. The menu options enable you to list the directories of evacuation cases and of met/dose cases for the current site. You also may delete selected input and output files from the system (if you own the files). Exercise care here, especially in deleting input files, because re-creation of files may be time consuming.

Figure 9.13. Menu for Case File Directory and Optional Delete

9.17 Change Case Numbers

EESF Top-Level Menu: Select J (*text mode menu*) or H (*graphic mode menu*)

This option displays the case selection form (example shown in Figure 9.14). The evacuation and met/dose models each have a case number because the two models have independent inputs. The case numbers that appear when you first enter the EESF system will be the ones last used in the last work session under your username and password. To find out what cases exist before you alter model case numbers, you can refer to the case file directory (see Section 9.16, **Case File Directory and Optional Delete**).

The data fields of the form in Figure 9.14 follow.

Site Name: Current site name for reference only.

Evacuation Model Case and Met/Dose Model Case: The first of the two case numbers shown for each model is the current case. When you select other options from the EESF main menu, you are operating with the data for the current case. You may change this case number to either switch to a different (existing) case or create a new case (by copying from the Model Base).

Model Base: The second or Model Base number shows the number of the case from which the current case was initially created. You would change this number only when you are creating a new set of case data.

Evacuation Model and Met/Dose Model		Case Selection Form V2.00
Site Name: MY DEVELOPMENT SITE		
Evacuation	Model Case: <u>4</u>	
Evacuation	Model Base: <u>4</u>	
Met/Dose	Model Case: <u>29</u>	
Met/Dose	Model Base: <u>29</u>	

Figure 9.14. Form for Change Site Name, Case Numbers (Example)

9.18 Concurrent Execution of MESORAD and IDYNEV

EESF Top-Level Menu: Select I (*graphic mode menu*)

This option enables you to concurrently execute the met/dose and evacuation models (MESORAD and IDYNEV) while depicting results graphically overlaid on a map of the site. You can execute the models one step at a time (between breakpoints where the MESORAD time step and the IDYNEV time period coincide) or from the beginning to end (with graphics displays at the end of each breakpoint). In order to use other functions, you must specify creation of sector statistics in the IDYNEV case (record types 5 and 49).

9.18.1 Initial Sequence of Operations

When you select Concurrent Execution on the EESF top-level menu, EESF does several things; some require your response. In the following sequence, EESF's actions are bulleted, and actions required of you are numbered.

- EESF displays a checklist for the current evacuation case (case number, site name, run type, etc.) and a prompt.

Execute IDYNEV for this case? (Y/N) [N] = >

1. Read the checklist to be sure the case is set up to run as you wish. Enter Y to continue (or N to return to the EESF top-level menu).

- EESF displays a similar checklist for the current met/dose case and the same prompt.

Execute MESORAD for this case? (Y/N) [N] = >

2. Read the checklist to be sure the case is set up to run as you wish. Enter Y to continue (or N to return to the EESF top-level menu).

- EESF then displays a message ***** MSG - INITIALIZING MESORAD**, and takes a few minutes to set up the models for starting concurrent execution and display.
- EESF presents a form for modifying the evacuation start time and entering shelter factors that could affect dose calculations. This same form and its input fields are described in Section 9.10.

3. Enter an evacuation offset time and shelter factors, as desired.

The system then creates some files that you will need later and runs the evacuation model through the first time period.

After completing this initial sequence, EESF displays a menu of Concurrent Execution Options. These options are explained in the following section.

9.18.2 Using the Concurrent Execution Options Menu

The Concurrent Execution Options menu appears as shown in Figure 9.15. Its chief functions are to enable you to start the concurrent model execution and to specify when the concurrent execution will begin--from the original start time or from a **breakpoint** in the course of execution and when it stops. The menu also contains an option for making limited interactive changes to the met/dose model between breakpoints.

The functions and options of the Concurrent Execution Options menu are explained here and on the following pages.

Overview of Using the Menu and Concurrent Execution. In general, during concurrent execution and display you will follow a cyclic sequence similar to the following:

1. On the Concurrent Execution Options menu, select Restart or Continue in order to choose the *Next breakpoint*. (Or you can exit and return to the EESF top-level menu.)

A **breakpoint** occurs when the end of a met/dose model time step coincides with the end of an evacuation model time period. The **Next breakpoint** is the breakpoint at which you wish concurrent model execution to stop.

2. Select Execute with modified parameters to begin model execution.
3. On the EESF Graphical Mapping Options menu, specify which output results are to be displayed. (Or exit and return to Step 1.)
4. View the site map and the graphic display of model results, using the EESF Graphics Display menu. (Or exit and return to Step 3.)

This sequence is explained in more detail in Section 9.18.3.

Concurrent Execution Options

Selection Designator	Function
A	Restart
B	Continue
C	Modify Met/Dose Parameters
D	Execute with modified parameters
X	Exit

Enter Selection Designator: _

Figure 9.15. Menu for Concurrent Execution of MESORAD and IDYNEV

The individual functions of the Concurrent Execution Options menu are listed here:

- **Restart and Continue:** Selection of either Restart or Continue will display the Describe Concurrent Execution form shown in Figure 9.16. The only input field on this form is the Next Breakpoint #.
- **Start Date/Time:** This is an informational field only. The start date and time can be changed only within the MESORAD and IDYNEV models themselves. The start time for concurrent execution will be the earlier of either the MESORAD or IDYNEV start times.
- **End Date/Time:** This is an informational field only.
- * **Next breakpoint #:** Enter a number that corresponds to one of the valid breakpoints listed in the lower half of the form. The next concurrent execution will stop at this breakpoint.
- **VALID BREAKPOINTS:** A valid breakpoint occurs when the end of a MESORAD time step coincides with the end of an IDYNEV time period. Valid breakpoints are displayed in two columns as shown in Figure 9.16.

DESCRIBE CONCURRENT EXECUTION

Start Date/Time 10/05/87 09:00:00 ←(display only)
 End Date/Time 00/00/00 00:00:00 Next breakpoint #: ____
 MESORAD time step size (hh:mm) 00:15 (currently disabled)

VALID BREAKPOINTS ARE:

1) 10/05/88 10:00:00	11) 00/00/00 00:00:00
2) 10/05/88 11:00:00	12) 00/00/00 00:00:00
3) 10/05/88 03:30:00	13) 00/00/00 00:00:00
4) 00/00/00 00:00:00	14) 00/00/00 00:00:00
5) 00/00/00 00:00:00	15) 00/00/00 00:00:00
6) 00/00/00 00:00:00	16) 00/00/00 00:00:00
7) 00/00/00 00:00:00	17) 00/00/00 00:00:00
8) 00/00/00 00:00:00	18) 00/00/00 00:00:00
9) 00/00/00 00:00:00	19) 00/00/00 00:00:00
10) 00/00/00 00:00:00	20) execute to end of information

Figure 9.16. Form for Controlling Concurrent Execution of the MESORAD and IDYNEV Models (Example)

For example: If the MESORAD time step is 15 minutes and the IDYNEV time periods are 3 hours, 3 hours, and 2 hours, the valid breakpoints would be at 3, 6, and 8 hours:

When Restart is selected, the concurrent execution will begin at the original start time, using the specified MESORAD time step size, and then continue until the next breakpoint is reached.

When Continue is selected, the concurrent execution will continue from the current valid breakpoint (the end time displayed on this form before you specify any new *next breakpoint #*), using the specified MESORAD time step size, and then continue until the next breakpoint is reached.

- Modify Met/dose Parameters: Selection of this option will display the menu shown in Figure 9.17. These menu options are explained here:
 - Change Weather Data: This option enables modification of weather records on the met/dose model Change Weather Data form (Section 4.6.3). No other input modification can be made without restarting the simulation.
 - List Weather Directory: This display option lists the time stamps of the weather records on the screen. (You cannot modify these records).
 - Exit: Exit returns you to the Concurrent Execution Options menu.

MODIFY MET/DOSE PARAMETERS	
Selection Designator	Function
A	Change weather data
B	List weather directory
X	Exit

Enter Selection Designator: _

Figure 9.17. MESORAD Menu for Modifying Met/Dose Parameters

- Execute with Modified Parameters: This option starts the concurrent execution of the models, beginning either from the start (Restart) or from the previous breakpoint (Continue), and ending at the current breakpoint. After the models reach the current breakpoint, EESF will:
 - display the EESF Graphical Mapping Options menu for input. This menu is shown and explained in Section 9.15, **Display Map-Related Graphics**. (Exiting from this menu with <GOLD> <Y> will return to the Concurrent Execution Options menu in Figure 9.15.)
 - display the Map Display menu for choosing background topographies (if that has not already been done during the current session).
 - display the selected topographies with a graphics menu for controlling the display of the currently selected model results. This menu is the EESF Graphics Display Menu shown in Figure 9.12 in Section 9.15, **Display Map-Related Graphics**. (Exiting from this menu will return to the EESF Graphical Mapping Options menu.)
 - display the selected model results for each time step selected in the execution until the selected Next breakpoint is reached.

9.18.3 Overall Sequence of Operations

After you have verified the model checklists and obtained the Concurrent Execution Options menu, you are ready to start a concurrent modeling session. The following list is a typical sequence of operations.

1. On the Concurrent Execution Options menu, select A - Restart. (Or exit and return to the EESF top-level menu.)

EESF displays the form to Describe Current Execution (Figure 9.16).

Restart indicates that model execution will start from the beginning. (If you have already executed and viewed model results, you can select B - Continue, which indicates that model execution will continue from its present position.)

2. Enter a number for the Next breakpoint #.

EESF returns to the Concurrent Execution Options Menu.

Note: EESF creates some necessary data files at this point. If you exit now from the Concurrent Execution Options menu (without executing to the selected Next breakpoint), it will cause problems when you subsequently use concurrent execution or try to graphically display case output. Avoid exiting at this point.

3. Optionally select **C - Modify Met/Dose Parameters** and modify weather data for the met/dose model (if desired).
4. Select **D - Execute with modified parameters**.

EESF displays ***** MSG: INITIALIZING MESORAD** and creates some necessary data files. EESF then displays the form for entering an evacuation offset time and sheltering factors to be used in dose calculations. (This form is explained in Section 9.10, **Modify Evacuation Start Times for Dose Calculations**.)

5. Enter an evacuation offset time or sheltering factors, as desired.

EESF performs the requested simulation and then presents the Graphical Mapping Options menu for specifying which output results are to be displayed. (Use of this menu is explained in Section 9.15, **Display Map-Related Graphics**.)

6. Specify the display of traffic data and some met/dose model results on the EESF Graphical Mapping Options menu. (Or press **<GOLD> <Y>** to exit and return to Step 1.)

EESF displays several messages as it prepares for displaying the model results:

If you are displaying results for the first time in this concurrent execution session, EESF displays the Map Display menu so you can choose topographic features for the site map.

7. Pick **SELECT TOPOGRAPHY** and then choose the desired map features from the resulting topography selection menu. Then exit the selection menu and exit from the Map Display menu. (This step occurs only once during a session, so be sure to choose all desired topographies.)

EESF displays the selected topographies, zooming in on the modeling area, and displays the EESF Graphics Display menu (described in Section 9.15, **Display Map-Related Graphics**).

You can then select model output step functions and reporting function from the menu.

8. Use the EESF Graphics Display menu to step through the current model results again. (Or exit and return to the Graphical Mapping Options menu and Step 6.)

9.19 If You Have Problems

IBS is designed to provide as much help as possible to the user in the form of data range checking, and status and diagnostic messages to make system use independent of computer systems specialists. If you have questions, try the following:

1. To obtain on-line help with data forms and fields, press the HELP (<F6>) key on your terminal keypad, as described in Section 9.3.
2. Contact the System Manager for assistance with problems related to the operation of EESF.
3. If problems seem to be related to hardware, refer to your equipment service contract for the name of your vendor representative.

9.20 Status and Diagnostic Messages

The system provides a number of status messages that keep you informed of system activity. These messages generally do not require any reciprocal action on your part. They will be generated automatically under certain circumstances, such as the start of a process that requires substantial computing time and does not involve user interaction.

9.21 References

Met/Dose Model Documentation

National Council on Radiation Protection and Measurements (NCRP). 1984. Radiological Assessment: Predicting the Transport, Bioaccumulation, and Uptake by Man of Radionuclides Released to the Environment. NCRP Report No. 76, Bethesda, Maryland.

Ramsdell, J. V., G. F. Athey, and C. S. Glantz. September 1984. MESOI Version 2.0: An Interactive Mesoscale Lagrangian Puff Dispersion Model with Deposition and Decay. NUREG/CR-3344, PNL-4753, prepared by the Pacific Northwest Laboratory for the U. S. Nuclear Regulatory Commission, Washington, D.C.

Ramsdell, J. V., G. F. Athey, T. J. Bander, and R. I. Sherpelz. The MESORAD Dose Assessment Model: Volume 2: Computer Code. NUREG/CR-4000, PNL-5219, prepared by the Pacific Northwest Laboratory for the U. S. Nuclear Regulatory Commission, Washington, D.C.

Sherpelz, R. I, T. J. Bander, G. F. Athey, and J. V. Ramsdell. MESORAD Dose Assessment Model: Volume 1: Technical Basis. NUREG/CR-4000, PNL-5219, prepared by the Pacific Northwest Laboratory for the U. S. Nuclear Regulatory Commission, Washington, D.C.

Evacuation Model Documentation

Federal Emergency Management Agency (FEMA). 1984. Exercise Evaluation and Simulation Facility. Evacuation Events Model. Part I-PREDYN User's Guide. FEMA-REP-6, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1984. Exercise Evaluation and Simulation Facility. Evacuation Events Model. Part II-Users Manual. FEMA-REP-7, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1986. Description of an Integrated Trip Assignment and Distribution Model (TRAD) for the IDYNEV System. TR-187, prepared by KLD Associates, Inc. for the Federal Emergency Management Agency, Washington, D.C.

Federal Emergency Management Agency (FEMA). 1988. PREDYN User's Guide. TR-131A, prepared by KLD Associates, Inc. for the Federal Emergency Management Agency, Washington, D.C.

Lieberman, E., B. Andrews, M. Davila, and M. Yedlin. 1983. Macroscopic Simulation for Urban Traffic Management: The TRAFLO Model. FHWA-RD-80, Federal Highway Administration, Washington, D.C.

Lieberman, E. 1984. Formulations for the DYNEV and IDYNEV Traffic Simulation Models Used in EESF. TR-154, prepared by KLD Associates for the Federal Emergency Management Agency, Washington, D.C.

Pacific Northwest Laboratory. 1988. Updated User's Guide for the IDYNEV System Including the TRAD Model. TR-188, prepared by KLD Associates, Inc. for the Pacific Northwest Laboratory, Richland, WA.

Protective Action Guides

Department of Energy (DOE). 1984. Guidance on Offsite Emergency Radiation Measurement Systems. Phase 2 - The Milk Pathway. WINCO-1009, prepared by Idaho National Engineering Laboratory for the Department of Energy, Washington, D.C.

A

application shell

- IBS 1.3, 6
- IBSSH 1.3
- IEMIS 1.6

application shells

- models 1.3
- other 1.6

C

case

- base 1.5
- creating a new 1.11
- current 1.5
- definition of 1.5
- switching 1.11
- system base 1.5

case site directory

- optional delete 2.14

chemical source inputs

- barge 8.18
- bullet tank 8.20
- fixed storage tank 8.20
- pipeline 8.20
- rail tanker 8.19
- road tanker 8.19
- ship 8.19

CHEMS Base Map, Display 8.24

CHEMS Case Data

- chemical spilled 8.12
- environmental conditions 8.14
- exit 8.21
- mode of operation 8.16
- model selection/model dependent inputs 8.17
- output specifications 8.21
- source inputs 8.18
- tank specification form 8.12
- time of spill/current time 8.13
- update 8.11

CHEMS Case/Site 8.23

- case number, CHEMS model 8.23
- change 8.23

- create a new case 8.24

- site name 8.24

CHEMS File Maintenance, Perform 8.26

CHEMS Models 8.1

- description of 8.2
- dispersion model 8.2
- explosion model 8.2
- heat radiation model 8.2
- menu 8.10
- model batch, execute 8.25
- model interactive, execute 8.25
- using 8.3

CHEMS Reports, Print/Display 8.25

CHEMS Spill Location, Select 8.23

Concentration of Radionuclides
in Leafy Vegetables\ 4.34**D**

D2

- Description of 3.2
- Modifying Model Input Data 3.3

D2 Chemical Hazard Prediction Model 3.1

D2 model 3.2

- D2 RUN capabilities 3.2
- inputs 3.3
- modifying supplementary input 3.7
- reference 3.3
- track 3.3

dispersion model

- cryogenic liquid, release 8.8
- high vapor pressure liquid, release 8.8
- liquified compressed gas, release 8.8
- overview 8.8

Dose Commitment

- radionuclide concentration, usage factor,
and dose factor 4.38

DOSE DETAILS 4.3

Dynamic Evacuation Model 6.1

E

EESF 9.1

- base map, display 9.8
- case file directory 9.24
- case number, change 9.25
- description of 9.2
- evacuation model [IDYNEV] 9.3
- evacuation start times for dose calculations, modify 9.11
- evacuation summary report, example 9.11
- execute MESORAD, batch 9.12
- execute MESORAD, interactive 9.13
- map-related graphics, display 9.18, 19
- map-related modeling results, display 9.4
- MESORAD AND IDYNEV\ 9.26
- messages, status and diagnostic 9.33
- meteorology/dose model [MESORAD/FEMA] 9.3
- meteorology/dose model, modify 9.9
- model evacuation, batch 9.10
- model evacuation, interactive 9.10
- on-line HELP, obtaining 9.5
- optional delete 9.24
- problems, if you have 9.33
- reports on terminal, display 9.16
- reports, print 9.14
- site name, change 9.25
- top-level menu 9.6
- using 9.4
- work session, starting 9.5

EESF [Exercise Evaluation and Simulation Facility] 9.1

- using 9.4
- computer memory and resources 9.5
- functions 9.4
- graphic mapping options 9.5
- meaningful use of 9.5
- top-level menu
 - graphic mode 9.7
 - obtaining 9.6
 - text mode 9.8
 - using 9.6
- work session
 - graphic or text mode, selecting 9.6

elevation set 1.5

emergency situations
simulations of 1.5

environmental conditions

- atmospheric conditions 8.14
- ground conditions 8.15
- water conditions 8.15

evacuation case generation
limitations of 5.2

evacuation key terms

- centroids 6.7
- channelization 6.7
- links 6.7
- nodes 6.7
- origin 6.7
- PREDYNGR 6.8
- sources and sinks 6.8
- subnetwork 6.8
- time interval 6.8
- time period 6.8
- traffic assignment 6.8
- traffic network 6.8
- trip distribution 6.8

evacuation model

- IDYNEV 2.3
- Messages, Status and Diagnostic 6.39
- stopping 2.11
- summary report 2.11
- traffic network 2.2

evacuation network, generating an

- requirements 5.2
- what you need to know 5.1

evacuation network generation 5.1

- automatic evacuation case generation 5.20
- case files created 5.27
- case numbers 5.4
- description of 5.1
- evacuation boundary, defining an 5.9
- evacuation cases 5.27
- flooding 5.4
- generating the network 5.16
- host locations, defining 5.11
- initial evacuation cases, generating 5.17
- network building, description of 5.20
- network parameter file, specifying 5.6

- network/case generation menu 5.8
- population set 5.4
- qualifying roadway selection 5.14
- roadway network, displaying the 5.19
- topography selection 5.7
- trip generation, associating populations 5.24

- evacuation start times for dose calculations
 - apply shelter factors 9.12
 - vehicle shelter factor 9.12

- Evacuation Model Input
 - key terms 6.7
 - modifying data 6.7

- Evacuation Traffic Network and Cases
 - generation options 5.3
 - POPULATION file 5.2
 - ROADS & TRAILS file 5.2

- explosion model
 - blast damage criteria 8.5
 - hazard distance calculation 8.5
 - mass of vapor exploding 8.4
 - overview 8.3
 - TNT blast overpressures 8.4
 - TNT equivalent mass 8.4

- F**
 - flood output overlay
 - SLOSH output 5.5

- G**
 - graphics control files 1.5
 - graphics logical names 1.5

- Graphics Mode
 - graphics selection menu 6.26
 - input, modifying 6.28
 - map, manipulating 6.27
 - modify input menu 6.26
 - modifying input 6.25

- H**
 - heat radiation model

- overview 8.6

I

- IBS model 1.6

- IDYNEV Function Selection 6.29
 - base map, graphical display 6.36
 - case/site, change 6.37
 - Execute - Batch 6.32
 - Execute - Interactive 6.31
 - exit 6.38
 - file maintenance, perform 6.38
 - map results, graphical display 6.33
 - problems, if you have 6.38
 - reports, print/display 6.36

- IDYNEV model
 - case data, graphic modification 6.26
 - description of 6.2
 - Interactive DYnamic Network
 - EVacuation 6.1
 - links 6.2
 - nodes 6.2
 - PREDYNGR, interactive input program 6.2
 - steps to assemble input data 6.2
 - Text or Graphic Mode 6.6
 - traffic assignment 6.2
 - traffic simulation 6.2
 - trip distribution 6.2
 - What You Need to Know 6.4

- IDYNEV model
 - Interactive DYnamic EVacuation 5.1
 - simulation 5.1
 - traffic assignment 5.1

- IDYNEV Work Session
 - Evacuation Model Input 6.6
 - flow of IEMIS operations 6.5
 - Graphic or Text Mode, Selecting 6.6
 - IDYNEV Function Menu 6.6
 - starting an 6.5
 - starting from IEMIS 6.5
 - starting from the IBS Program 6.6

- IDYNEV, [see evacuation model and evacuation key terms] 2.3

- IEMIS

Index

- case/site functions, using the 1.11
 - environment files 1.10
 - starting 1.7
 - top-level menu 1.7
 - user environment 1.10
 - user environment, changing the 1.10
- IEMIS application shell**
- models 1.6
- M**
- map-related graphics**
- current or cumulative statistics 9.19
 - dynamic features 9.19
 - evacuation graphics 9.18
 - grids 9.19
 - menu of graphics display 9.20
 - menu of options 9.18
 - met/dose graphics 9.18
 - specify limits for graph data selection 9.19
 - static features 9.19
- mapping functions**
- using 1.12
- menus**
- graphic mode 1.12
 - text mode 1.12
- MESORAD AND IDYNEV**
- concurrent execution 9.26
 - concurrent execution options menu, using 9.28
 - form for controlling concurrent execution, example 9.29
 - menu for modifying met/dose parameters 9.30
 - menu of concurrent execution 9.28
 - operations, initial sequence of 9.26
 - operations, overall sequence of 9.31
- MESORAD - Batch, Execute 4.31**
- MESORAD/FEMA 4.33**
- Concentration of Radionuclide 4.34
 - in Meat 4.37
 - in Milk 4.37
 - in Pastures 4.36
 - Dose Commitment 4.38
 - Gamma + Beta Exposure Rates 4.40
 - Gamma Exposure Rate 4.40
 - Ingestion Pathway Doses 4.33
 - Radioiodine Air Concentrations 4.39
 - Team Reports 4.39
- MESORAD Graphic Options 4.5**
- Contour Specifications 4.8
 - Graphical Mapping Options 4.5
 - Map-Related Graphics, Displaying 4.9
 - Parameters, Changing Global Graphics 4.7
- MESORAD - Interactive, Execute 4.30**
- MESORAD Model**
- Description of 4.2
 - Dose Model 4.3
 - Map-Related Display 4.2
 - Messages, Status and Diagnostics 4.32
 - Meteorology Model 4.3
 - Model Library 4.1
 - Problems, If You Have 4.32
 - What You Need to Know 4.2
- MESORAD Top-Level Menu 4.4**
- Change Case/Site 4.31
 - Display Base Map 4.27
 - Graphic Options 4.5
 - Perform File Maintenance 4.30
 - Print/Display Reports 4.27
 - Update Case Data** 4.11
 - Update Elevation 4.11
- meteorology/dose model [MESORAD/FEMA]**
- dose model 9.4
 - meteorology model 9.3
- model selection/model dependent**
- explosion 8.17
 - fire radiation 8.17
 - vapor dispersion 8.17
- models**
- access 1.2
 - CHEMS 1.6
 - D2 1.6
 - IDYNEV 1.6
 - MESORAD 1.6
 - OSPM 1.6

N

Natural Hazards Support Module
Regional Evacuation Analysis 2.1

network building
intersection of network links 5.24
topography requirements 5.20

O**OSPM**

base map, graphical display 7.11
case/site, change 7.10
contour results, graphic menu only 7.12
contours, delete 7.31
decibel report 7.14
description of 7.3
Digital Mapping Structure 7.DMS 7.13
elevation data, obtaining and using 7.4
elevation file, edit 7.15
execute OSPM - batch 7.31
execute OSPM - interactive 7.29
file maintenance, perform 7.31
function selection menu 7.8
impedance, update 7.16
new siren type, create 7.32, 34
overview of 7.4
siren, place 7.23
siren, update 7.27
site specs, list 7.28
what you need to know to use 7.4

OSPM, Place Siren
add siren 7.24
default siren, change 7.23
delete siren 7.25

Outdoor Sound Propagation Model [see
OSPM] 7.1

P

population set 1.6

PREDYNGR

continuing input session 6.10
data form, modifying 6.12
data form, selecting 6.12
data input forms 6.9

Destination Capacities [Type 178] 6.24
ending input session 6.10
Entry Link Volumes [Type 50] 6.22
Evacuation Data [Type 49] 6.21
free-flow speeds, modify 6.8, 10
function type, selecting 6.13
function types, modified 6.13
graphic mode input, selecting 6.12
Identification [Type 01] 6.16
input requirements 6.15
Internal Centroids [Type 177] 6.24
Link Characteristics [Type 11] 6.19
link capacities, modify 6.8
Link Characteristics [Type 11] 6.19
Link Turn Movements [Type 21] 6.20
Load Factors [Type 52] 6.22
menu of block change options 6.14
menu of form types 6.13
Origin Control [Type 06] 6.18
Origin Trip Production Volumes [Types
176] 6.23
Output Options [Type 05] 6.17
overview of input session 6.9
Run Control [Type 02] 6.16
Sign Control Data [Types 35] 6.20
Signal Control Data [Type 36] 6.20
Source/Sink Volumes [Type 51] 6.22
starting input session 6.8
subnetwork, specifying 6.11
time period form, example 6.11
Time Period Specification [Type 03]
6.17
time period, specifying 6.11
Time Step Data [Type 04] 6.17
Title [Type 00] 6.16
Traffic Assignment Parameters [Type
175] 6.23

PREDYNGR

evacuation model data input program
2.5

printing

black and white hard copy 1.13
color hard copy 1.13

R

Regional Evacuation Analysis
case number, changing 2.13
case site directory 2.14

- caveat 2.2
- current site network, updating 2.8
- evacuation case 2.2
- evacuation model, batch execution 2.10
- evacuation model, interactive execution 2.10
- evacuation progress prediction 2.2
- evacuation results, displaying 2.11
- evacuation results, printing or displaying 2.13
- evacuation time prediction 2.2
- first-cut evacuation case, generating a 2.8
- general description of 2.1, 2
- IDYNEV 2.3
- population file functions 2.15
- program flow 2.5
- set code, changing 2.13
- site map from site file, displaying 2.8
- site name, changing 2.13
- starting a work session 2.5
- traffic network 2.2
- uses of 2.2
- what you need to use 2.3

reports, print

- dose report 9.14
- dose report, example 9.15
- evacuation report 9.14
- field team summary release report 9.14
- ingestion pathway report 9.14
- menu of options 9.15
- met/dose case debug output 9.14
- PAG dose summary report 9.14

S

- Setup option 1.3

site

- definition of 1.6

site maps

- displaying 1.12
- displaying model data on 1.12
- graphic editing 1.12
- printing 1.12

site master file 1.6

sites

- switching between 1.11

SLOSH

- background information 5.5
- use of 5.5

SLOSH [see also flood output overlay] 5.5

T

traffic network, generation of

- first-cut 2.2

U

Update Case Data

- Cartesian Grid, Change 4.21
- Cartesian Grid, Generate 4.27
- Checkpoints, Change 4.18
- Control Parameters, Change 4.13
- Output Options, Change 4.25
- Polar Grid, Change 4.22
- Polar Grid, Generate 4.26
- Source Details, Change 4.15
- Station Definitions, Change 4.19
- Terrain, Change 4.20
- Weather Data, Change 4.16
- Weather Directory, Display 4.23

user environment

- changing 1.4
- current setup 1.3
- general information 1.3
- graphics displays 1.4
- IBS system 1.3
- managing 1.4
- modifying 1.3, 4
- start-up 1.4
- terminology 1.5

DISTRIBUTION

No. of
Copies

No. of
Copies

OFFSITE

ONSITE

- 2 DOE/Office of Scientific and
Technical Information

COL Lawrence E. Skelly
United States Army Nuclear and
Chemical Agency
7500 Backlick Road, Building 5073
Springfield, VA 22150-3198

Mr. J. Thompson
Operations Support Directorate
HQ FEMA
5321 Riggs Road
Gaithersburg, MD 20882

John Bower
Bower Software Services
104 E. 36th Place
Kennewick, WA 99337

DOE Richland Operations Office

R. B. Goranson K8-50

- 33 Pacific Northwest Laboratory

B. M. Bailey (20)
M. J. Burford
T. R. Downing
S. W. Matsumoto (2)
S. L. Savelli
S. N. Stevens
J. R. Williams
C. Winters (2)
Publishing Coordination
Technical Report Files (3)

DATE

FILMED

5/10/94

END