1 of 3

INTEGRATED BASELINE SYSTEM (IBS)
Version 1.03

MODELS GUIDE

January 1993

**Prepared for
the Federal Emergency Management 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 Federal Emergency Management 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), operated by the Federal Emergency Management Agency (FEMA)^(a), is a system of computerized tools for emergency planning and analysis. 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.

IBS is a dynamic system. Its capabilities are in a state of continuing expansion and enhancement. However, if you have any questions related to the operation of IBS:

WRITE TO: Federal Emergency Management Agency
NP-SE-AS (Attn: Ralph Sinnot)
500 C St. S. W.
Washington, D.C. 20472

OR TELEPHONE: (202) 646-3824

Audience

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

(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.

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. Model Library — Expert System Model (TIMM)

A description of the TIMM program.

10. Integrated Modeling (EESF)

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

A. Technological Hazards Support Module

A description of the Technological Hazards Support Module.

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.

Contents

Preface	iii
About This Guide	v

Section 1. Overview of Computer Modeling and Support Modules

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 IEMIS Application Shell	1.6
1.3.3 Managing the IEMIS User Environment	1.8
1.3.3.1 Using the Manage Job Environment Menu	1.9
1.3.3.2 Modify Current Model Environment	1.10
1.3.3.3 Choose Graphics Control Files	1.12
1.3.3.4 Print or Display Text Files	1.14
1.3.3.5 Access Topography Journal	1.14
1.3.3.6 Show Case Flow Diagram	1.15
1.3.4 Model Library Menu	1.16
1.3.5 Technological Hazards Support Module	1.17
1.3.6 Natural Hazards Support Module	1.18
1.4 IEMIS User Environment	1.19
1.4.1 Environment Files	1.19
1.4.2 Changing the User Environment	1.19
1.4.3 Using Case/Site Functions	1.20
1.5 Using Mapping Functions within Model Subsystems	1.21

Section 2. Regional Evacuation Analysis

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
2.3.1 Selecting Graphic or Text Mode	2.7
2.3.2 Selecting Evacuation Model Input or Population Updating	2.7
2.4 Updating Demographic Data	2.8
2.5 Using the Regional Evacuation Main Menu	2.11
2.6 Generate a First-Cut Evacuation Case - Graphic Mode Menu Only	2.13
2.7 Update Current Site Network (Model Input Data) - Text Mode Menu Only ..	2.14
2.8 Display Site Map from Site File - Graphic Mode Menu Only	2.14
2.9 Execute Evacuation Model - Batch	2.15
2.10 Execute Evacuation Model - Interactive	2.16
2.11 Display Evacuation Results - Graphic Mode Menu Only	2.17
2.12 Print/Display Reports	2.18
2.13 Change Site Name, Set Code, Case Number	2.18
2.14 Case Site Directory and Optional Delete	2.19

Section 3. D2 Chemical Hazard Prediction Model

D2 Chemical Hazard Prediction Model	3.1
---	-----

Section 4. Model Library — MESORAD Model

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
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	4.14
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	4.21
4.6.7 Change Cartesian Grid	4.22
4.6.8 Change Polar Grid	4.23
4.6.9 Display Weather Directory	4.24
4.6.10 Change Output Options	4.25
4.6.11 Generate Polar Grid	4.26
4.6.12 Generate Cartesian Grid	4.26
4.7 Display Base Map - Graphic Menu Only	4.27
4.8 Print/Display Reports	4.28
4.9 Perform File Maintenance	4.31
4.10 Execute Mesorad - Interactive	4.32
4.11 Execute Mesorad - Batch	4.32
4.12 Change Case/Site	4.33
4.13 If You Have Problems	4.33
4.14 Status and Diagnostics Messages	4.34
4.15 Mesorad/FEMA	4.35
4.15.1 Ingestion Pathway Doses	4.35
4.15.2 Mesorad/FEMA Team Reports	4.41

Section 5. Generate Evacuation Network

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	5.7
5.3.3 Selecting Topographies to Be Used During Network Generation ...	5.9
5.3.4 Using the Network/Case Generation Menu	5.9
5.3.5 Defining an Evacuation Boundary	5.11
5.3.6 Defining Host Locations	5.12
5.3.7 Qualifying Roadway Selection	5.15
5.3.8 Generating the Network	5.17

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	5.23
5.4.3 Evacuation Cases and Case Files Created	5.26

Section 6. Dynamic Evacuation Model

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
6.3.1 Selecting Graphic or Text Mode	6.6
6.3.2 Selecting Evacuation Model Input or the IDYNEV Function Menu ..	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 Choosing a Time Period and Subnetwork	6.12
6.4.4 Selecting Which and How Input Data Will Be Modified	6.13
6.4.5 Title (Type 00)	6.17
6.4.6 Identification (Type 01)	6.17
6.4.7 Run Control (Type 02)	6.17
6.4.8 Time Period Specification (Type 03)	6.18
6.4.9 Time Step Data (Type 04) and Output Options (Type 05)	6.18
6.4.10 Origin Control (Type 06)	6.19
6.4.11 Link Characteristics (Type 11)	6.20
6.4.12 Link Turn Movements (Type 21)	6.21
6.4.13 Sign or Signal Control Data (Types 35 & 36)	6.21
6.4.15 Entry Link Volumes (Type 50)	6.23
6.4.16 Source/Sink Volumes (Type 51)	6.23
6.4.17 Load Factors (Type 52)	6.24
6.4.18 Traffic Assignment Parameters (Type 175)	6.24
6.4.19 Origin Trip Production Volumes (Type 176)	6.25
6.4.20 Internal Centroids (Type 177)	6.25
6.4.21 Destination Capacities (Type 178)	6.25
6.5 Graphics Mode of Modifying Input to the Evacuation Model	6.26
6.6 Using the IDYNEV Function Selection Menu	6.31
6.6.1 Execute Evacuation - Interactive	6.32
6.6.2 Execute Evacuation - Batch	6.33
6.6.3 Display Results On Map - Graphics Mode Only	6.33
6.6.4 Display Base Map	6.38
6.6.5 Print/Display Reports	6.38
6.6.6 Change Case/Site	6.38
6.6.7 Perform File Maintenance	6.39
6.6.8 Exit	6.40
6.7 If You Have Problems	6.40
6.8 Status and Diagnostic Messages	6.40

Section 7. Model Library — Outdoor Sound Propagation Model

7.1 Description of the Outdoor Sound Propagation Model	7.2
7.2 What You Need to Know to Use the OSPM Model	7.3
7.3 Using the OSPM Function Selection Menu	7.6

Contents

7.4 Change Case/Site	7.9
7.5 Display Base Map - Graphic Menu Only	7.10
7.6 Contour Results - Graphic Menu Only	7.11
7.7 Report Decibel (dB) - Graphic Menu Only	7.13
7.8 Edit Elevation File - Graphic Menu Only	7.14
7.9 Update Impedance - Graphic Menu Only	7.15
7.10 Place - Graphic Menu Only	7.20
7.10.1 Change Default Siren	7.20
7.10.2 Place Siren	7.21
7.10.3 Delete Siren	7.22
7.11 Update Siren - Graphic Menu Only	7.24
7.12 List Site Specs	7.25
7.13 Execute OSPM - Interactive	7.26
7.14 Execute OSPM - Batch	7.27
7.15 Perform File Maintenance	7.27
7.16 Delete Contours - Graphic Menu Only	7.27
7.17 Create a New Siren Type	7.28

Section 8. Model Library — CHEMS Models

8.1 Description of the CHEMS Models	8.2
8.1.1 Explosion Model	8.2
8.1.2 Heat Radiation Model	8.2
8.2 What You Need to Know to Use the CHEMS Models	8.4
8.3 Using the CHEMS Models Menu	8.11
8.4 Update Case Data	8.13
8.5 Select Spill Location	8.25
8.6 Change Case/Site	8.26
8.7 Base Map Display - Graphic Menu Only	8.27
8.8 Execute Model Inter or Execute Model Batch	8.28
8.9 Print/Display Reports	8.28
8.10 Perform File Maintenance	8.30

Section 9. Model Library - Expert System Model (TIMM)

Model Library - Expert System Model (TIMM)	9.1
--	-----

Section 10. Model Library - Integrated Modeling (EESF)

10.1 Description of EESF	10.2
10.1.1 Evacuation Model (IDYNEV)	10.3
10.1.2 Meteorology/Dose Model (Mesorad/FEMA)	10.3
10.1.3 Display of Map-Related Modeling Results	10.4
10.2 What You Need to Know to Use EESF	10.5
10.3 Obtaining On-Line HELP with Data Input	10.5
10.4 Starting an EESF Work Session	10.6
10.4.1 Selecting Graphic or Text Mode	10.8
10.4.2 Selecting Evacuation Model Input or the EESF Function Menu ..	10.8
10.5 Obtaining and Using the Top-Level EESF Menu	10.9
10.6 Display Base Map - Graphics Menu Only	10.10
10.7 Modify Meteorology/Dose Model Input Data	10.11

10.8 Model Evacuation - Batch	10.12
10.9 Model Evacuation - Interactive	10.13
10.10 Modify Evacuation Start Times for Dose Calculations	10.14
10.11 Execute Mesorad - Batch	10.15
10.12 Execute Mesorad - Interactive	10.16
10.13 Print Reports	10.17
10.14 Display Reports (on terminal)	10.19
10.15 Display Map-Related Graphics - Graphics Menu Only	10.21
10.16 Case File Directory and Optional Delete	10.27
10.17 Change Site Name or Case Numbers	10.28
10.18 Concurrent Execution of Mesorad and IDYNEV	10.29
10.18.1 Initial Sequence of Operations	10.29
10.18.2 Using the Concurrent Execution Options Menu	10.30
10.18.3 Overall Sequence of Operations	10.33
10.19 If You Have Problems	10.35
10.20 Status and Diagnostic Messages	10.36
10.21 References	10.38

Appendix A. Technological Hazards Support Module

Technological Hazards Support Module	A.1
--	-----

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. 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 IEMIS 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. IEMIS 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)
 - Technological Hazards Support Module (TH)
 - Natural Hazards Support Module (NH)
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 IEMIS application shell, use the Manage Job Environment option.
- From the IBS application shell, select the Setup option and use the Manage Job Environment option.

General information about the user environment is provided in the following sections. For further information on specific files controlling the user environment, see 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 user environment menus and forms common to both the IBS and IEMIS applications are described in the IBS User Guide. Those unique to IEMIS are described further in this section. 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 evacuation 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 bases 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.
- **graphics logical names**: IBS logical names that are assigned to the graphics control files.

- **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 IEMIS Application Shell

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

IDYNEV
MESORAD
OSPM
CHEMS

Also accessible through IEMIS are the following application shells:

Exercise Evaluation Simulation Facility (EESF)
Regional Evacuation Analysis (REA)
Natural Hazards (NH)
Technological Hazards (TH)

To start IEMIS, 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 *iemis* and press the Return key. The IEMIS top-level menu will display as shown in Figure 1.2.

```
-----
                        Integrated Emergency Management
                        Information System
                        Version 3.1

Selection
Designator      Selection

      A      Manage Job Environment
      B      Presentation Graphics

      C      Create New Site
      D      Mapping Functions
      E      Model Library

      F      Technological Hazards Support Module
      G      Natural Hazards Support Module
      H      National Preparedness Support Module
      I      Emergency Operations Center Module

      J      Training
      X      Sign Off

Please Enter Selection Designator. _
-----
```

Figure 1.2. IEMIS Top-Level Menu

The functions relevant to IEMIS modeling and special support functions are the following:

- A Manage Job Environment. This function enables you to set up particular configurations of IEMIS hardware, graphics software, map data, and modeling data for use in using IEMIS models and other programs. Operation of options that are the same in the IBS application and the IEMIS application are explained in the *IBS User Guide*. Options available only through IEMIS are explained in the following sections.
- C Create New Site. This function is an interactive program for assembling the geographic and model input data that define an IEMIS site, a geographic locale for IEMIS modeling activities. For further information, see the *IBS Data Management Guide*.
- E Model Library. This option displays a menu of IEMIS modeling functions, as shown in this section.
- F Technological Hazards Support Module. This option displays a menu of functions related to technological hazards.
- G Natural Hazards Support Module. This option displays a menu of functions related to natural hazards.

At publication, the following menu options were not fully implemented.

H National Preparedness Support Module

I Emergency Operations Center Module

Option **J** Training may not be available at your installation.

1.3.3 Managing the IEMIS User Environment

IEMIS Menu: Select **A** (Manage Job Environment)

This section describes options unique to the IEMIS application for managing your user environment— how IEMIS is set up to work with your choice of models and data. By changing the user environment, you can select which data to use: for example, which site to use, which cases are current for the IEMIS models, and which geographic information is accessible for map displays. A graphics workstation is not required. This section covers the following topics:

1.3.3.1 Using the Manage Job Environment Menu

1.3.3.2 Modify Current Model Environment

1.3.3.3 Choose Graphics Control Files

1.3.3.4 Print or Display Text Files

1.3.3.5 Access Topography Journal

1.3.3.6 Show Case Flow Diagram

The following topics are common to both IEMIS and IBS programs and are described in detail in **Section 12 Setup**, of the *IBS User Guide*.

- Modify Hardware Parameters
- Maintain Site Directories
- Review Graphics Control Files
- Update Site Master File
- Copy Site or Case Files

1.3.3.1 Using the Manage Job Environment Menu

When you select A (Manage Job Environment) from the IEMIS menu, your screen will display the menu of job environment options (Figure 1.2).

MANAGE JOB ENVIRONMENT

Selection Designator	Selection
A	Modify Hardware Parameters
B	Modify Current Model Environment
C	Maintain Site Directories
D	Review Graphics Control Files
E	Choose Graphics Control Files
F	Update Site Master File
G	Copy Site or Case File
H	Print or Display Text Files
I	Access Topography Journal
J	Show Case Flow Diagram
X	Exit

Please Enter Selection Designator. _

Figure 1.2. Menu for Manage Job Environment

To select a function from the menu, type the selection designator (letter) and press <RETURN>. The menu options are described in sequence in the following sections.

13.3.2 Modify Current Model Environment

Manage Job Environment Menu: Select B (Modify Current Model Environment)

This option enables you to switch between sites, sets of data, and other characteristics of the current **model environment** of a site: the current cases, elevation set, population set, topographies, icons, and attributes.

When you select this option, IEMIS will display the data input form for Modify Current Model Environment (Figure 1.3).

```

-----
MODIFY CURRENT MODEL ENVIRONMENT

Model Name      Case Number   Source
Curr/Base      Node      Elevation Set   Population Set
-----
MESORAD         ___/___
IDYNEV          ___/___
HEAT            ___/___
OSPM            ___/___

Current Elevation Set _____ mm/dd/yy hh:mm mm/dd/yy/hh:mm
Current Population Set _____ mm/dd/yy/hh:mm mm/dd/yy/hh:mm

Created          Modified

Current Site: _____

Associated Topography Control File: _____

Associated Icon Control File: _____

Associated Attribute Master File: _____
-----

```

Figure 1.3. Form for Modify Current Model Environment

The form and its fields are explained here. An asterisk (*) appears next to each input item; other items supply information but do not require input.

- **Model Name:** This field displays the names of the IEMIS models for which the case numbers are listed. The form displays data for only four models at a time: the up and down arrow keys scroll the models through the four-line area on the form.
- * **Case Numbers--(Current Case/Base Case):** The form displays current and base case numbers for each IEMIS model. Enter a three-digit number (for example, 001) to change the current case.

- **Source Node:** This field displays the name of the node on which the model case was assembled.
- **Elevation Set:** This field displays the name of the elevation data set that was used with the model listed in the first column (not currently implemented).
- **Population Set:** This field displays the name of the population data set that was used with the model listed in the first column (not currently implemented).
- * **Current Elevation Set:** The form initially displays the current elevation set. Enter the name of an elevation data set to change the current set.

Note: Currently this is the only way to switch between elevation data sets. Changing the site does NOT automatically change the elevation data set. Enter the elevation data set name here only after you have selected the desired site, exited this form and reselected the job environment option.

- * **Current Population Set:** Enter the name of a population data set that contains population data for the current site.

For example, an urban site might have different population sets for day time and evening, or for weekdays and weekends.

Note: This will not effect the population topography within the IBS program.

- **Dates Created and Modified:** These fields display the dates on which the current elevation and population sets were first created and last modified.
- * **Current Site:** The form normally displays the full site name but requires only the first four characters of the site name as input. Case numbers on the form are based on your choice of the current site. If you change the current site, the case numbers will change to the highest case numbers found for the new current site. (IEMIS will remind you with a message on the screen.) The sites accessible to you are listed in the **site master file** (SITE.DAT) in your top-level directory.

To change sites, enter the four-character site code for the site. (The site code is the first four characters of the site name).

- * **Associated Topography Control File:** To change the topography control file in use, enter the name of a different topography control file.
- * **Associated Icon Control File:** To change the icon control file in use, enter the name of a different icon control file.

- * **Associated Attribute Master File:** To change the attribute master file in current use for the site, enter the name of a different attribute master file.

13.3.3 Choose Graphics Control Files

Manage Job Environment Menu: Select E (Choose Graphics Control Files)

This option enables advanced users of IEMIS to tell IEMIS which **graphics control files** to use. The graphics control files dictate what items (topographies, icons, areas, etc.) can be displayed by the IEMIS map display software. Novice users of IEMIS will generally not need to use this capability because IEMIS specifies standard system control files.

When you select this option, IEMIS will display the data input form shown in Figure 1.4. By entering the names of special control files that you have created, you can set up IEMIS' map display software to display features that are special to your needs. For example, if you were tracking the transport of hazardous materials, you might relabel some topographic features to suit the application: selected **ROADS AND TRAILS** data might become **LAND ROUTES**, while some **STREAMS** data might become **WATER ROUTES**. To take full advantage of these options, you must have an in-depth knowledge of how the map display software works. Refer to the *IBS Data Management Guide* and the *IBS Utilities Guide* for complete information.

CHOOSE GRAPHICS CONTROL FILES

Topography Control File Name:

Icon Control File Name:

Cover Map Control File:

Attribute Dictionary:

Setup Control File:

The Setup control file may override other values.

Figure 1.4. Form for Choose Graphics Control Files

The fields of the form are explained here. An asterisk (*) appears next to each input item. When you enter a control file name, enter the complete file specification: *disk:[directories]filename*.

- * **Topography Control File Name:** Enter the name of a different topography control file--the file that lists the available topographic features and the database directories where different types of topographic data reside.
- * **Icon Control File Name:** Enter the name of a different icon control file--the file that lists the available icons and the data files that are needed to plot each icon.
- * **Cover Map Control File Name:** Enter the name of a different cover map control file--the file that lists the geographic database areas available for display.
- * **Attribute Dictionary:** Enter the name of a different attribute dictionary (attribute master file)--the file that contains a complete list of attributes that describe graphic items.
- * **Setup Control File:** Enter the name of a different setup control file--the DCL command (.COM) file that assigns the IEMIS logical names to specific file names and directory names of a geographic database.

While you are still viewing this form, you also have the following special option:

To assign all graphics control files and then execute the setup control file according to the form: press **GOLD-Z**. You need not make any changes to the form.

— What Happens When You Alter a Control File Name?

When you exit this form, IEMIS will implement any changes you have made. For example, if you have changed the name of the icon control file, IEMIS will refer to that file for instructions about displaying icons in any graphic displays. IEMIS does this by reassigning a system logical name so that it points to the specific file you chose. Whenever a logical name is reassigned, you will see a message explaining that the previous value of the logical name has been superseded. Assignments will occur in the sequence in which the items are listed on the form.

If you have entered a new name for the setup control file, IEMIS will execute the commands contained in the new setup control file when you exit the form. The setup control file normally contains the logical name assignments for the other control files. Assignments in the setup control file will override any assignments duplicated in the form. It is important to remember this when using the **GOLD-Z** option.

1.3.3.4 Print or Display Text Files

Manage Job Environment Menu: Select H (Print or Display Text Files)

This option enables you to print or display the contents of a text file. This can be helpful when you are planning to use special control files (see Section 1.3.3.3, **Choose Graphics Control Files**).

When you select this option, IEMIS will display the following prompt:

Enter name of file to review:

Enter the file specification of any file. IEMIS will display the following menu. Select an option for printing or displaying the contents of the file. Select an option by pressing the appropriate key:

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

Note: Some IEMIS installations have modified the underlying program that performs this process to support system-specific devices. Consult your local IEMIS manager if you have questions.

1.3.3.5 Access Topography Journal

Manage Job Environment Menu: Select I (Access Topography Journal)

This option enables you to edit the **topography journal file**. The topography journal file is an ordinary text file whose purpose is for you to record how you have used topographic data with different IEMIS models and cases.

When you select this option, IEMIS will display the following prompt:

Press any key to begin editing the topography journal file.
Press X to return to the previous menu.

Pressing any key will start the text editor and enable you to edit the topography journal file. Initially, the journal file is set up with column headings for topography name, case number, and model name:

Topography Name	Cases	Models

You can edit this setup to be whatever you wish: the intention is to provide a way of keeping track of the topographic data you have used.

1.3.3.6 Show Case Flow Diagram

Manage Job Environment Menu: Select J (Show Case Flow Diagram)

This option displays a diagram that shows what data are needed to execute an IEMIS model for a particular case and site. If the model requires results from other IEMIS models as input, the diagram shows the name(s) of these models and the current case numbers for each of them. These current cases must have output before the model of interest can execute.

When you select J from the Manage Job Environment menu, IEMIS will display a case flow diagram and data input form like the one in Figure 1.5.

CASE FLOW DIAGRAM		
INPUT	MODEL	OUTPUT
<div style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> model site </div> <div style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> model site </div> <div style="border: 1px solid black; padding: 5px;"> model site </div>	<div style="border: 1px solid black; padding: 10px; text-align: center; margin: 10px auto; width: 150px;"> _____ (model name) </div>	<div style="margin-top: 100px;"> (site) (case) </div>

MODEL OF INTEREST? _____
 TIME SPAN OF INPUT: _____
 TIME SPAN OF EXECUTIONS: _____
 MODEL SPATIAL RANGES: _____

Figure 1.5. Example of Case Flow Diagram Form

The fields of the form are explained here. An asterisk (*) appears next to each input item. Other fields are for display only.

- * Model of Interest?: Enter the name of the model for which you desire a case flow diagram (for example: IDYNEV, Mesorad, etc.).

The following fields display information about the chosen model:

- Time Span of Input: This field displays the time span over which there is input data for the model.
- Time Span of Executions: This field displays the time span over which the model is to perform a simulation based on its current case input. This is only meaningful for models that simulate situations over time.
- Model Spatial Ranges: This field displays the longitude and latitude ranges of the model of interest (as specified for the current case data).

1.3.4 Model Library Menu

IEMIS Menu: Select E (Model Library)

When you select E on the IEMIS top-level menu, IEMIS displays the Model Library menu shown in Figure 1.6. Each option represents one emergency-related computer model in IEMIS or a supporting function.

Each of these options is described in separate sections in this guide.

MODEL LIBRARY

Selection Designator	Selection
A	Mesorad Model
B	Generate Evacuation Network
C	Dynamic Evacuation Model (IDYNEV)
D	Outdoor Sound Propagation Model (OSPM)
E	CHEMS Model
F	Expert System Model (TIMM)
G	Integrated Modeling (EESF)
X	Exit

Please Enter Selection Designator: _

Figure 1.6. Model Library Menu

1.3.5 Technological Hazards Support Module

IEMIS Menu: Select F (Technological Hazards Support Module)

When you select F on the IEMIS main menu, IEMIS displays the menu shown in Figure 1.7. This menu groups models and support functions that are of interest for emergency planning and evaluations related to technological hazards.^(a)

```
-----
                                TECHNOLOGICAL HAZARDS SUPPORT MODULE

Selection
Designator      Selection

      A      REP Program Database
      B      Hazardous Material Database
      C      Mesorad Model
      D      Dynamic Evacuation Model
      E      Outdoor Sound Propagation Model
      F      CHEMS Model
      X      Exit

Please Enter Selection Designator. _
-----
```

Figure 1.7. Technological Hazards Support Menu

(a) At publication, options A, and B were not yet implemented on the system.

1.3.6 Natural Hazards Support Module

IEMIS Menu: Select G (Natural Hazards Support Module)

When you select G on the IEMIS main menu, IEMIS displays the menu shown in Figure 1.7. This menu^(a) groups models and support functions that are of interest for emergency planning and evaluations related to natural hazards.

NATURAL HAZARDS SUPPORT MODULE

Selection Designator	Selection
A	<i>Flood Insurance Database</i>
B	<i>Earthquake Database</i>
C	<i>Slosh Model</i>
D	<i>Earthquake Intensity Prediction Model</i>
E	<i>Regional Evacuation Analysis</i>
F	<i>Weather</i>
X	<i>Exit</i>

Please Enter Selection Designator.

Figure 1.8. Natural Hazards Support Menu

(a) At publication, options A, B, C, and D were not yet implemented on the system.

1.4 IEMIS User Environment

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

1.4.1 Environment Files

IEMIS maintains two special files in the top-level directory of each user, **SITE.DAT** and **JOB_ENV.DAT**. A third file, a mapping edit file, can also be placed there by model mapping functions.

- **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.
- **CURRENT_EDIT_FILE.DMS** will be read and updated by certain models when the **EDIT_FUNCTIONS** are selected during Base Map Display. This file can contain any sort of graphic vector data that you might wish to create during a model session. These data are separate from graphic model product sets that might be created.

1.4.2 Changing the User Environment

You can change the user environment by using the "Manage Job Environment" function (A on the IEMIS top-level menu). This function provides you with a consistent means of telling IEMIS what data that you wish to use and how you wish 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

The model subsystems all 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 1.9. 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 four-character site code 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 IEMIS 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>), IEMIS 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).

```
-----  
                      MESORAD CASE/SITE SPECIFICATION  
Site Name: MAIN  
Base Site Node: FEMA02  
Mesorad Model Case Number:  0 Based on:  0  
-----
```

Figure 1.9. Example Change Case/Site Form

1.5 Using Mapping Functions within Model Subsystems

The IEMIS models are embedded in modeling subsystems that enable you to create input data sets (cases), execute the models, and obtain model results. Because IEMIS 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. Generally when you select a model subsystem from an IEMIS menu, you will see the following prompt:

DO YOU WISH Graphic or Text Menus? (enter G or T):

- If you enter G, the succeeding operational menus will include options for graphically modifying or displaying case data.
- If you enter T, the succeeding menus will NOT include any graphic options. Thus a subset of IEMIS modeling tasks can be accomplished on a nongraphic terminal.

Displaying Site Maps. The graphic menus generally contain a DISPLAY BASE MAP OPTION. By selecting DISPLAY BASE MAP on the main graphics menu of any IEMIS 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 IEMIS 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 IEMIS workstation (one of various Tektronix terminals), you can print copies of the site map and model graphics using the Tektronix color copier:

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

Section 2

Regional Evacuation Analysis

IEMIS Menu: Select G (Natural Hazards Support Module)

Natural Hazards Support Module: Select E (Regional Evacuation Analysis)

Note: Items A, B, C, and D on the Natural Hazards Support Module menu are not implemented. Item F: Weather Data is maintained by FEMA, but is no longer distributed by PNL. Contact FEMA for further information.

This section describes Regional Evacuation, a capability for applying IEMIS' 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 Generate a First-Cut Evacuation Case
- 2.7 Update the Current Site Network (Model Input Data)
- 2.8 Display Site Map From Site File
- 2.9 Execute Evacuation Model - Batch
- 2.10 Execute Evacuation Model - Interactive
- 2.11 Display Evacuation Results
- 2.12 Print/Display Reports
- 2.13 Change 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 EESF 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 IEMIS. 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.

Regional Evacuation Analysis

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

Natural Hazards Support Module: Select **E** (Regional Evacuation Analysis)

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 Library Menu of IEMIS.

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*).

In its opening sequence of prompts, Regional Evacuation determines whether you will be modifying IDYNEV input data, updating the evacuation model demographics (population data), or performing other Regional Evacuation functions. The general flow of operations is depicted in Figure 2.1. The initial prompts are explained in the following parts.

Regional Evacuation Analysis

Figure 2.1. General Flow of Regional Evacuation Operations in IEMIS

2.3.1 Selecting Graphic or Text Mode

After you select Regional Evacuation Model from the Natural Hazards Support Module menu, you will see the following prompt:

DO YOU WISH Graphic or Text Menus? (enter G or T):

- If you enter **G**, the succeeding menus (for selecting Regional Evacuation functions or for modifying IDYNEV input data) will include options for graphically displaying (and modifying) site and modeling information.
- If you enter **T**, the succeeding menus will **NOT** include any graphic options. Thus a subset of Regional Evacuation tasks could be accomplished on a nongraphic terminal.

2.3.2 Selecting Evacuation Model Input or Population Updating

After you select graphic or text menus, you will see the following prompt:

Do you wish to update evac model input? (enter Y or N):

- If you enter **Y** (yes), then you will begin a data entry sequence for modifying evacuation model input (explained fully in Section 6.4). When you finish modifying the IDYNEV input data, you will return to the Natural Hazards Support menu.

The data input process for the evacuation model involves using the **PREDYNGR user interface**, a program that enables you to create or modify the data to be used as input for the evacuation model. The input data consist largely of items that describe the condition of the IDYNEV roadway network. See Sections 6.4 and 6.5 for background, terminology, and procedures for modifying evacuation model input.

- If you enter **N** (no), then:
 - If you are in text mode, IEMIS presents a menu of Regional Evacuation functions. Section 2.5 explains the options of the Regional Evacuation main menu.
 - If you are in graphic mode, IEMIS displays a prompt:

Do you wish to update evac model demographics? (enter Y or N):

A **Y** (yes) answer will send you to Updating Demographic Data (Section 2.4), while a **N** (no) answer will send you to the Regional Evacuation main menu (Section 2.5).

2.4 Updating Demographic Data

To update the demographic (population) data for the current site, answer Y (yes) to the initial prompt, "Do you wish to update evac model demographics?"

Updating population data consists of changing the population values for population points within the site area or changing the associations between population points and centroids (entry points) on the traffic network. When you change the population values, you are NOT changing the values stored in the site population file: rather you are changing a special set of population values that were extracted from that file.

After you change demographic data, IEMIS gives you the option of including the updated data in new case input for the evacuation model. If you choose to include the data, IEMIS enables you to update the evacuation traffic network by automatically going through the "Generate Evacuation Network" sequence described in Section 5. You will be able to specify which cases (traffic assignment and simulation) will use the updated population data.

Select Topographic Features. When you first choose to "update evac model demographics," you will see a graphic menu of map display functions, which are fully described in the *IBS Utilities Guide*, MPDISPLAY. Use the SELECT TOPOGRAPHIES option to select map features to be included on the site map. When you exit the map display menu, the topographies that you selected for the current site will appear on the map display screen (if they occur in the site data files). The display will automatically be sized to an area that is 1.1 times the current evacuation model area, around the site center. A graphic menu of options at the right side of the screen then appears as shown in Figure 2.2.

```
EXIT
HELP
CONTINUE
FIND NODE
POPULATION DATA
DISPLAY BASE MAP
ZOOM IN
ZOOM OUT
RESTORE BASE PLOT
REFRESH SCREEN
COORDINATE PICK
MAP ZOOM
```

Figure 2.2. Graphic Menu for Updating Model Demographics (Population)

Many of the display manipulation options on this menu are available in the Map Display subsystem (*IBS Utilities Guide*, MPDISPLAY). The functions described here are those especially relevant to examining and updating demographic data: FIND NODE and POPULATION DATA.

The following descriptions assume that you have picked the menu item using the graphics input device (joydisk or bit pad and mouse) at your terminal.

- **FIND NODE:** A text prompt will ask you to input a node number at your terminal keyboard. Type the identification number of a node on the display, and press <RETURN>. The display will zoom in with the indicated node at the center of the display.
- **POPULATION DATA:** Picking "POPULATION DATA" causes the display of hexagonal icons that show the locations of population points. Each population point has an associated population. The system then displays a submenu for examining or modifying population data:

```

EXIT
HELP
CONTINUE

EXAMINE CENTROID
MODIFY CENTROID
EXAMINE POPULATION POINTS
MODIFY POPULATION POINTS
  
```

The following descriptions assume that you have picked the corresponding menu options.

- **Examine Centroid:** This option enables you to view the population data associated with a specific centroid. (Centroids appear on the traffic network as small arrowheads that indicate where a specific portion of the population will enter a link of the network.)

Pick a centroid for which you want to view population data; then pick "CONTINUE." All population points assigned to that centroid will blink. The system will list the population points (and populations) that are assigned to enter the network at that centroid. Press <RETURN> to continue.

- **Modify Centroid:** This option enables you to change the population points that are assigned to the centroid. All such modifications are made to a special file of population point/centroid assignments in the population directory of the current site.

1. Pick a centroid for which you want to change the population point assignments; then pick "CONTINUE."

All population points assigned to that centroid will blink. The system will list the current population assignments to the centroid.

2. Press <RETURN> to continue.

The system will prompt you for corrected population point assignments.

3. Type a population point number; then tap <RETURN>.

4. Repeat step 3 until you have entered the numbers of all population points to be assigned to the centroid.

5. Press <CTRL Z> to end entry.

The system will list the population points and the associated populations assigned to the centroid.

- **Examine Population Point:** Pick a population point; then pick "CONTINUE."

The system will display the current population associated with that point. You may have to press the <DIALOG> key to read the display.

- **Modify Population Point:** This option enables you to change the population count associated with a specific population point (for the current site -- NOT for the maps database).

1. Pick a population point; then pick "CONTINUE."

The system will display the current population at that point and prompt you to enter a new population.

2. Type a new population value and tap <RETURN>.

- **Exit:** When you pick this option, the system prompts, "Do you wish the population changes to become permanent?". If you answer N (no), then IEMIS ignores any changes you made and exits to the previous menu. If you answer Y (yes), then IEMIS will create new (updated) population files, first giving you the option of applying the updated data to the current evacuation case:

Do you wish to apply updated population data to your current evacuation case?

If you answer Y (yes), IEMIS will take you through the evacuation network generation function for updating the traffic network and for choosing which cases will use the updated population data. (See Section 5, **Generate Evacuation Network**.)

- **EXIT:** This option exits the graphic menu for updating demographic data.

2.5 Using the Regional Evacuation Main Menu

If you do not choose to update model input or demographic data, you will see one of the menus shown in Figures 2.3 and 2.4, depending on whether you selected graphics 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).

To select a function from either menu, type the action code letter and press <RETURN>. The Regional Evacuation functions are described in Sections 2.6 through 2.14.

REGIONAL EVACUATION ANALYSIS		
<u>Action Code</u>	<u>Action to be Taken</u>	<u>Explained in Section</u>
A	Generate First-Cut Evacuation Case	2.6
B	Display Site Map From Site File	2.8
C	Execute Evacuation Model - Batch	2.9
D	Execute Evacuation Model - Interactive	2.10
E	Display Evacuation Results	2.11
F	Print/Display Reports	2.12
G	Change Site Name, Set Code, Case Number	2.13
H	Case Site Directory and Optional Delete	2.14
X	Exit	
Current Site Name _____		
Current Population Set Code _____		
Based on Set Code _____		
Evacuation Case Number _____ Based on Case Number _____		
Enter Action Code corresponding to desired action.		

Figure 2.3. Main Menu for Regional Evacuation - Graphic Mode

Regional Evacuation Analysis

REGIONAL EVACUATION ANALYSIS

Action Code	Action to be Taken
A	Update Current Site Network
B	Execute Evacuation Model - Batch
C	Execute Evacuation Model - Interactive
D	Print/Display Reports
E	Change Site Name, Set Code, Case Number
F	Case Site Directory and Optional Delete
X	Exit

Current Site Name _____
Current Population Set Code _____
Based on Set Code _____
Evacuation Case Number _____ Based on Case Number _____
Enter Action Code corresponding to desired action.

Figure 2.4. Main Menu for Regional Evacuation - Text Mode

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

Regional Evac Graphic Menu: Select A (Generate a First-Cut Evacuation Case)

Selecting "Generate First-Cut Evacuation Case" on the Regional Evacuation menu enables you to automatically create a first-cut evacuation case for the current site and evacuation case number. The name of the current site and the number of the evacuation case appear at the bottom of the Regional Evacuation menu.

Table 2.1 summarizes the process of generating a first-cut evacuation case. Notice that this process requires that site files contain Roads and Trails 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**.

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

<u>What You Do</u>	<u>What IEMIS Does</u>
1. Select A - Generate Evacuation Network on the Regional Evacuation menu.	Presents the First Cut Case/Site Specification form.
2. Specify on the form: <ul style="list-style-type: none"> - the site name - the base site node (system) - traffic assignment case # - traffic simulation case # - population set name - flood (SLOSH) output overlay. 	Asks 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 and the number of people per vehicle.	Establishes People Per Vehicle.
4. Load the default parameter settings (or use an existing network parameter file).	Presents the Map Display menu for displaying map features and information (<i>IBS Utilities Guide</i> , MPDISPLAY).
5. Select the topographies to be displayed, including POPULATION and ROADS & TRAILS (or others of similar content).	Retrieves and displays the topography data from the site files. Displays the graphic menu for network and case generation.

Regional Evacuation Analysis

6. **DEFINE EVACUATION BOUNDARY** by graphically drawing a line around the evacuation area. Creates a DMS file (*CnnnEVAC.DMS*) 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 *CnnnEVAC.DMS* 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 **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).
10. Pick **GENERATE EVAC CASES** on the menu. 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.

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

Regional Evacuation Text Menu: Select A (Update Current Site 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.

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

Regional Evacuation Graphic Menu: Select B (Display Site Map from Site File)

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.

TECHNICAL 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.

2.9 Execute Evacuation Model - Batch

Regional Evacuation Menu: Select B (*text mode menu*) or C (*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:

Enter "Y" to execute this case: _

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.

Regional Evacuation Analysis

- 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 separate from your current interactions with IEMIS. 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.

2.10 Execute Evacuation Model - Interactive

Regional Evacuation Menu: Select C (*text mode menu*) or D (*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:

Enter "Y" to execute this case: _

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 2.9) 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.

2.11 Display Evacuation Results - Graphic Mode Menu Only

Regional Evacuation Graphic Menu: Select E (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.

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.

Figure 2.5. Graphics Menu for Display Evacuation Results

2.12 Print/Display Reports

Regional Evacuation Menu: Select **D** (*text mode menu*) or **F** (*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

2.13 Change Site Name, Set Code, Case Number

Regional Evacuation Menu: Select **E** (*text mode menu*) or **G** (*graphic mode menu*)

Select this option on the Regional Evacuation menu to change entries displayed in the case identification area below the menu. The system will present the data input form shown in Figure 2.7. You can then change the site name, population set code, and evacuation case number to access different data.

The name, code, and number that appear when you first enter the Regional Evacuation 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 case numbers, refer to the case site directory (Section 2.14, *Case Site Directory and Optional Delete*).

```

-----
Current Site Name _____
Current Population Set Code _____
          Based on Set Code _____
Evacuation Case Number _____ Based on Case Number _____
-----

```

Figure 2.7. Site/Case Control Information

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.

Regional Evacuation Analysis

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

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.

- * Site Name: Enter a 4-character site code or a site name string of up to 40 characters. If you change the current site name, the system will automatically update the rest of the entries in the form. If you do change the site name, change 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.)
 - * Population Set Code: Enter a 1- to 20-character code that identifies the site-dependent population set to be used in making a first-cut approximation of the evacuation case.
 - * Based on Set Code: The code of the population set used in creating a new current population set.
 - * Evacuation Case Number: 000 to 999.
 - * Based on Case Number: 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 Site Directory and Optional Delete

Regional Evacuation Menu: Select F (*text mode menu*) or H (*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.

When you exit from the menu in Figure 2.8, a menu for Population File Functions appears, as shown in Figure 2.9. Use these functions to review and optionally delete population case files. Population files contain the information that associates the centroids on the traffic network with specific population points and populations.

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 Site Directory and Optional Delete

POPULATION FILE FUNCTIONS

Function Code	Function Description
A	List Population Case Table of Contents
B	Specify Population Case Numbers for Deletion
X	Exit POPULATION FILE FUNCTIONS

Enter Function Code Corresponding to the Function You Wish
to Execute

Figure 2.9. Menu for Population File Functions

Section 3

D2 Chemical Hazard Prediction Model

IBS Menu: Select EMERGENCY ACTIVITIES--HAZARD ANALYSIS--EXECUTE D2

D2 Run Capabilities Menu: Select B (Modify D2 Case Standard Input)

D2 Run Capabilities Menu: Select C (Modify D2 Case Supplementary Input)

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

3.3 References

3.1 Description of D2

The **EXECUTE D2** option on the **HAZARD ANALYSIS** menu displays the following text-screen menu of "D2 RUN Capabilities." The 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). Also included is a key capability for creating *scenarios*: scenarios are unique combinations of situational data that link the D2 case output with an evacuation model case, meteorological conditions, and *population pattern* for a specific Implementing Procedure.

3.2 Modifying D2 Model Input Data

The D2 model inputs are easy to change; however, the knowledge of *what* those changes should be is left up to you (Whitacre, Griner, Myirski, and Sloop 1987). The model results are meaningful only if its inputs are meaningful. 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 as well.

D2 RUN Capabilities

Selection Designator	Selection
A	Create New D2 Case
B	Modify D2 Case Standard Input
C	Modify D2 Case Supplementary Input
D	Modify D2 Case Input With Latest MET Data
E	Run the Current D2 Case
F	Display Current D2 Case
G	Perform File Maintenance
H	Create Scenario for Current D2 Case
X	EXIT

Enter Option: _

The options of the "D2 RUN Capabilities" menu are described briefly in the following paragraphs, as in the IBS User Guide. The discussion of input options is expanded here.

A - CREATE NEW D2 CASE enables you to create a new case based on an existing case. The existing case is then referred to as the *base case*. You select the case to be used as a base case by either 1) entering a D2 case number directly or 2) searching the agent, munition, and meteorological data for the cases--as explained for **CHANGE D2 TRACK CASE** on the IBS main menu.

B - MODIFY D2 CASE STANDARD INPUT presents the following input screen for entering D2 input data:

D2 STANDARD INPUT				
IBS CASE: _____		ONPOST CASE: _____		CLASS CODE: _____
LOCATION: _____		LONGITUDE _____		LATITUDE: _____
DATE __/__/__		TIME: __:__:__		
SOURCE TERM:				
MUNITION: _____		AGENT: _____		
RELEASE TYPE: _____				
SPILL OR AIRBORNE: _____				
NUMBER OF MUNITIONS: _____				
NUM	QUANTITY	RELEASE DURATION		
____	_____	____	_____	_____
____	_____	____	_____	_____
____	_____	____	_____	_____
____	_____	____	_____	_____
____	_____	____	_____	_____
____	_____	____	_____	_____
MET DATA:				
STABILITY TYPE: _____		TEMPERATURE: _____		
WIND SPEED: _____		WIND DIRECTION: _____		
DESCRIPTION:				

1 C (Celsius)
2 F (Fahrenheit)
3 K (Kelvin)
When you are entering values
in a field, you may select
here from a list of units.

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:

- **IBS Case:** the IBS D2 case number.
- **Onpost Case:** the D2 case number from the onpost system.
- **Class(ification) Code:**
 - COMM = Community Emergency
 - POST = Post Only Emergency
 - LIMI = Limited Area Emergency
 - NONS = Non-Surety Emergency
- **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
- **Date:** the date in the format DD/MMM/YY where MMM is the first three characters of the month. The date is used in determining the season.
- **Time:** the time in the format hh:mm:ss where hh is 00 to 23 (24-hour clock).
- **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 115-mm Rocket, M55
 - 525 525-lb Bomb, MK116
 - 139 Bomblet, M139
 - M23 Land Mine, M23
 - 4.2 4.2-in. Cartridge, M2A1
 - NON Nonmunition

- **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" 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 need to fill in the appropriate values in the supplementary input form.

D2 Chemical Hazard Prediction Model

- **Description:** an 80-character description of the D2 case.

C - MODIFY D2 CASE SUPPLEMENTARY INPUT enables you to change some basic D2 input parameters that are listed as shown in the following (expanded) screen example. These parameters are not expected to be changed frequently.

D2 SUPPLEMENTARY INPUT

```
IBS CASE: 57  ONPOST CASE:       OUTPUT TYPE: Q  TWO MIN
CORRECT: Y
SPECIFY DOSAGES: N                SURFACE: GRA        WOODS TYPE: DW

ALPHA                : 1.40
SYR                  :           
BETA                  :           
SZR                  :           
ATMOSPHERIC PRESSURE (mm Hg) :           
TIME OF EVAP (min)      :           
AREA OF WETTED SURFACE(sq m) :           
LENGTH OF SURFACE DOWNWIND(m) :           
MOLECULAR WEIGHT        :           
MOLECULAR VOLUME        :           
VAPOR PRESSURE (mm Hg)   :           
BOILING POINT(deg K)     :           
TIME AFTER FUNCTIONING   :           
HEIGHT OF STACK (m)      :           
DIAMETER OF STACK (m)    :           
TEMPERATURE OF STACK (deg C) :           
VELOCITY OF EFFLUENT (m/sec) :           
RELATIVE DENSITY OF EFFLUENT :           
FROST PROFILE EXPONENT   :           
HEAT RELEASED (cal)      :           
CLOUD RADIUS (m)         :           
CLOUD COVER (1/10)       :           
CLOUD HEIGHT (ft)        :           
SUN ELEVATION ANGLE (deg) :           
BREATHING RATE (liters/min) :           
REFLECTION COEFFICIENT    :           
SKIN FACTOR FOR SUBJECT CLOTHING:           
SAMPLING HEIGHT          :           
SETTLING VELOC OF CLOUD CENTROID:           
ROUGHNESS LENGTH         :           
SOURCE HEIGHT            :           
SIGMA X                 :           
SIGMA Y                 :           
SIGMA Z                 :           
HEIGHT OF MIXING LAYER    :           
DOSAGE OR CONCENTRATION(1) :           
DOSAGE OR CONCENTRATION(2) :           
DOSAGE OR CONCENTRATION(3) :           
```

When you reach the end of the screen, the parameters list "scrolls" forward to show further parameters.

- **IBS Case:** the IBS D2 case number.
- **Onpost Case:** the D2 case number from the onpost system.
- **Output Type:**
 - 0 Dosage (default)
 - 1 Concentration (mg/m3)
 - 2 Concentration (ppm)
 - 3 Fumigation Concentration
- **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)
- **Specify Dosages:** Y (yes) or N (no)--If you wish your three dosages of interest to be: 1 percent lethality, no deaths, and no effects, this field should be set to N. This means you WILL NOT need to input the three dosage intervals into this form. If you wish to specify your own three dosage intervals, this field should be set to Y. This means you WILL need to specify the three dosage intervals. These dosage intervals are the last three fields in the scrolling set of real number inputs on the screen.
- **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

D - MODIFY D2 CASE INPUT WITH LATEST MET DATA enables you to alter the meteorological input to the model. When you select this option, the IBS reads a file report of current weather conditions and extracts information to be used as input for the current D2 case. The weather conditions file (named WEATHER_OBS.DAT) is created by the onpost system and is periodically transmitted to the IBS for updating of the current weather conditions.

E - RUN THE CURRENT D2 CASE executes the D2 model for the current case inputs. Based on the case results, the IBS creates output files that are identified with the current IBS D2 case number. The IBS also creates one graphic output file that contains polygons depicting the D2 track contours. Note that the amount and type of information contained in this graphic output file depends on the D2 input. If possible, the model produces information on dose "slice" points along the track. Those points appear as asterisks along the track center line. Some chemical agents do NOT result in the generation and appearance of these points. In that case, you will see this message after D2 execution:

SLICE FILE NOT CREATED. DOSAGE ATTRIBUTES WILL NOT BE GENERATED.

F - DISPLAY CURRENT D2 CASE displays the current D2 track contours. If a track already appears in the graphics area, it is erased before the new current track is displayed.

3.3 References

Whitacre, C. G., Griner III, J. H., Myirski, M. M., and Sloop, D. W. 1987.
Personal Computer Program for Chemical Hazard Prediction (D2PC). CRDEC-
TR-87021, U.S. Army Munitions Chemical Command, Aberdeen Proving Ground,
MD.

Section 4

Model Library — MESORAD Model

IEMIS Menu: Select E (Model Library)

Model Library Menu: Select A (Mesorad Model)

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 10.21.

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 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, IEMIS 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 A (Mesorad Model)

When you select Mesorad from the Model Library menu, you will first see a prompt asking you if you would like graphic or text menus:

DO YOU WISH Graphic or Text Menus? (enter G or T):

Depending on your selection of Graphic (G) or Text (T) 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/SITE	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).

```

-----
MESORAD MODEL FUNCTIONS

Selection
Designator      Selection

  A      Print/Display Reports
  B      Update Case Data
  C      Perform File Maintenance
  D      Execute Mesorad - Interactive
  E      Execute Mesorad - Batch
  F      Change Case/Site

  X      Exit

Please Enter Selection Designator: _
-----

```

Figure 4.2. Mesorad Top-Level Menu - Text Mode

To start a Mesorad function on the text menu, type the corresponding letter 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.
- * 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 4.6.10).

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 Options Menu

— 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) 30

START DATE/TIME END DATE / TIME TIME INCREMENT
 DAYS HOURS MINUTES

01-JAN-88 13:15 01-JAN-88 20:00 00 00 15

TITLE	MINIMUM	NUMBER OF CONTOURS	USER DETAILED 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>.

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.

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**: (Not activated at this time).
- **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 B (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
- Display Weather Dir(ectory)
- Change Source Details
- Change Output Options.
- Change Weather Data

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).

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.

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 Meteorological/Dose Model - Text Mode

EXIT	
HELP	
CONTINUE	
CHANGE CONTROL PARAMS	<i>Explained in Section</i> <u>4.6.1</u>
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 Meteorological/Dose Model - Graphic Mode

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 10.21).
- * **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.

Model Library — MESORAD Model

METEOROLOGICAL/DOSE MODEL CONTROL PARAMETERS

Run Title : MAINE YANKEE
10 mile Base Case

Scenario Start Date : 01 OCT 86
Scenario Start Time : 12:00
Time Step Size : 1:00
Scenario Duration : 006 hour 00 minutes

Number of Sources : 1 (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 Meteorological/Dose Model Control Parameters

- * 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 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.)

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.

```

-----
                        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    .1000E+00      Ru-106   .1000E+00      Te-132   .1000E+00
I-131    .1000E+00      I-132    .1000E+00      I-133    .1000E+00
I-134    .1000E+00      I-135    .1000E+00      Xe-133    .1000E+00
Xe-135    .1000E+00      Cs-134    .1000E+00      Cs-137    .1000E+00
Ce-114    .1000E+00      U-234     .0000E+00      Zr-95     .0000E+00

- 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

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.

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.

```

-----
                                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

4.6.4 Change Checkpoints

Use the form in Figure 4.12 to change the name and locations of up to 50 checkpoints for dose and concentration level. For each country, the checkpoint locations must be coordinated with the cartesian coordinates (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.
- * 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.

```

-----
                                CHECKPOINTS
-----
Checkpoint      X              Y
CHKPT01      0.3000E+01 (km)  0.3000E+01 (km)

Y (Y,N)      Level Type      Level 1      Level 2
Y (Y,N)      Concentration    0.1000E+01    0.1000E+01
Y (Y,N)      Thyroid          0.0000E+00    0.0000E+00
Y (Y,N)      Whole Body       0.0000E+00    0.0000E+00

- 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

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, but the met/dose model will use only those with a "Y" entered in the last data field. 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 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.

Model Library — MESORAD Model

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

STATION DEFINITIONS

Station No.	Location		Elev	Name
	X	Y		
<u>0001</u>	<u>0.0000</u>	<u>0.0000</u>	<u>11</u>	<u>STA1</u>

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

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).
- * 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>.

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

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

Figure 4.14. Form for Changing Met/Dose Model Terrain

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).

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

Topography

West	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Center
Row	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>

Center	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	Col	East
16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31
<u>200</u>	<u>210</u>	<u>210</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>200</u>	<u>220</u>	<u>220</u>	<u>230</u>	<u>230</u>	<u>210</u>	<u>210</u>	<u>200</u>	<u>200</u>	<u>200</u>

- 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

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

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.

```
-----
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 - Puff Summary - TSTWF - TIMER	- Dose Report - Field Team Summary Release Report - PAG Dose Summary Report - 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.

Model Library — MESORAD Model

- 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:

Key Press Function

- 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

Model Library — MESORAD Model

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

2

of

3

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 meteorological/dose (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             _____
```

Enter "Y" to execute this case: _

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 meteorological/dose (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/Site

Mesorad Text Menu: Select F (Change Case/Site)

Selecting CHANGE CASE/SITE from the Mesorad top-level menu will display the Mesorad case/site specification form (example shown below in Figure 4.25). The "Site Name" in the form refers to the current site. The "Base Site Node" displays the name of the current computing system node. The "Mesorad Model Case Number" is the "current case" number. The "Based on" case 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).

If you change Site Name, you should change 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.)

```

-----
                        MESORAD CASE/SITE SPECIFICATION
Site Name:MAIN
Base Site Node:FEMA02
Mesorad Model Case Number:  0  Based on:  0
-----

```

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 checking, 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.
- AN UNANTICIPATED ERROR HAS BEEN ENCOUNTERED WITH THE EVACUATION BASE CASE - PLEASE REPORT TO YOUR SYSTEM MANAGER.
- The link-node diagram will be excluded from the background map.
- The background map will not be drawn.
- If you have also selected options that ARE currently implemented, these functions will be performed at this time.

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 x 365 (d)

— Concentration of Radionuclides in Pasture

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.

Model Library — MESORAD Model

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

Food Product	Quantity Consumed per Year (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 i th 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 percent 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 percent 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 **E** (Model Library)

Model Library Menu: Select **B** (Generate Evacuation Network)

This section describes Generate Evacuation Network, a capability for automatically generating initial evacuation cases and a traffic network for the evacuation model, IDYNEV. 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 EVacuation) model. When applied to a site that has been extracted from the National 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 National 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 & TRAILS" 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 and Trails 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

<u>What You Do</u>	<u>What the System Does</u>
1. Select B - Generate Evacuation Network on the Model Library menu.	Presents the First-Cut Case/Site Specification form.
2. Specify on the form: <ul style="list-style-type: none"> - the site name - the base site node (system) - traffic assignment case # - traffic simulation case # - population set name - flood (SLOSH) output overlay. 	Asks 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. 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>).
4. Select the topographies to be displayed, including POPULATION and ROADS & TRAILS (or others of similar content).	Retrieves and displays the topography data from the site files. Displays the graphic menu for network and case generation.
5. 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.
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.

Generate Evacuation Network

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

<u>What You Do</u>	<u>What the System Does</u>
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 percent population that will evacuate and the number of people per vehicle.	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.

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

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

5.3.1 Specifying Site Name, 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: Enter the name of the site for which you wish to generate a first-cut traffic network.
- * Base Site Node (display only): the name of the node on which the site database is stored.
- * First Cut TA Case: Enter a case number for an IDYNEV **traffic assignment** (TA) case to be created.
- * First Cut Sim Case: Enter a case number for an IDYNEV **simulation case** to be created.

First Cut Case/Site Specification Form

Site Name: _____
 Base Site Node: _____
 First Cut TA Case: 1
 First Cut Sim Case: 2
 Based on Population Set Name: BASIC POPULATION
 Flood Output Overlay (optional): _____

Figure 5.1 Menu for Generate Evacuation Network

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

The name of the default site population set is generally "Basic Population" -- 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 & TRAILS 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.

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 about what network parameters to use.

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

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

- > Enter a percentage value. For example, to indicate a 90 percent 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, you will see the following prompt:

DO YOU WISH TO LOAD DEFAULT NETWORK PARAMETER SETTINGS (Y OR N):

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

- YES -- Load default network parameter settings. "Yes" 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 CnnnEVACPARAM.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.
- NO -- Do NOT load default network parameter settings. "No" 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 "No."

Generate Evacuation Network

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.

Be sure to pick ROADS & TRAILS 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
QUALIFY ROADWAY SELECTION
GENERATE NETWORK
DEFINE EVAC BOUNDARY
DEFINE HOST LOCATIONS
GENERATE EVAC CASES
ENABLE ROADWAY NETWORK
DISABLE ROADWAY NETWORK

ZOOM IN
ZOOM OUT
DISABLE ROADWAY NETWORK
```

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*).
- **QUALIFY ROADWAY 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.
- **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, IEMIS generates network information in preparation for case generation. (See Section 5.3.8.)
- **DEFINE EVACUATION BOUNDARY:** Allows you to draw a boundary line that defines the evacuation area. (See Section 5.3.5.)
- **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 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) of the evacuation roadway

network. The network is based on the site ROADS & TRAILS information (or other similar topographic information), the evacuation boundary, and any special qualifications that you have made with QUALIFY ROADWAY 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

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

Generate Evacuation Network

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, 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.

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 to support the population from the evacuation area.

```

EXIT

HELP
CONTINUE

DEFINE HOST LOCATION

DELETE ONE HOST

DELETE ALL HOSTS
---

COORDINATE PICK

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

Figure 5.4. Graphics Menu for Defining Host Locations

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.

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
    
```

Generate Evacuation Network

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. 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 & TRAILS 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 CnnnEVACPARAM.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 "No" when the system asks if you wish to load default 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.

Generate Evacuation Network

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	Y	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 & TRAILS" 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 & TRAILS 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.)
- * **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 & TRAILS topography data are available. If you try to generate a network without first selecting the ROADS & TRAILS 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, you may answer Y and the system will allow you to continue.
- 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 ROADS & TRAILS 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 =          7
POPULATION ASSIGNED TO          7 CENTROIDS
TOTAL POPULATION ASSIGNED =      8013
TOTAL POPULATION POINTS TO EVACUATE =          5
TOTAL POPULATION POINTS EVALUATED =      48
<RETURN> to continue
```

When you press <RETURN> to continue, the system estimates turns that vehicles could take to exit the evacuation area and associates each host location with exit nodes on the evacuation boundary:

```
TURNS ESTIMATED
HOST/EXIT ASSOCIATIONS COMPLETED
```

Generate Evacuation Network

When the GENERATE NETWORK menu item returns to its normal color, the system is ready to generate initial evacuation cases with GENERATE EVAC CASES, explained in the following section.

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.

- 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.
```

- 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 system 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].)
- 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. 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.**

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 *CnnnINP.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, *CnnnLNK.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. 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 must include a ROADS & TRAILS 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 & TRAILS 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 & TRAILS 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:

<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 & TRAILS** data, as well as further explanation of attributes, in a master list of attribute codes in the *IBS Data Management Guide*.

The network building process makes the following assumptions based on the attributes associated with items in the **ROADS & TRAILS** 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)

- 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.

Generate Evacuation 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>Free-Flow Speed</u>	<u>Description</u>	<u>Minor Code(s)</u>
60	Interstate Highways	52
60	U.S. Highways	54
60	Limited access routes	350
60	Toll roads	360
55	Primary (Class 1) roads	60, 70, 80 90
55	Interstate connector	175
55	Toll road connectors	365
45	Secondary state/county roads	58
45	Class 2 roads	100, 110, 120, 130
45	Clover leaf or interchange	250
35	Class 3 road or street	140
35	Class 4 road or street	150
25	Cul-de-sac	40
25	Traffic circle	240

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

<u>Minor Code(s)</u>	<u>Description</u>
51, 53, 55, 57	All roads proposed or under construction
160, 170	Trails (Class 5)
181	Foot bridges
190	Ferry crossings
200	Parking area perimeters
210	Arbitrary extensions of line (join or closure)
330	Roads 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, and roadway information in the national

database is not defined in sufficient detail to include that data. Therefore the following simple method is used to assign possible "through" or "turn" movements for links at each network node.

- Links are defined in the order of the data read from the road-defining topography file. For example, consider a link X whose second node location (as read from the ROADS & TRAILS data) is identical to the first node location of a link Y. Link X is considered to be an upstream link of link Y, and link Y is considered to be a downstream link of link X.
- Turns are assigned for each downstream link in the following manner:
 - a. The first node found is assumed to be a through movement.
 - b. The second node found is assumed to be a right turn.
 - c. The third node found is assumed to be a left turn.
 - d. The fourth node found is assumed to be a diagonal turn.

This sequence is an arbitrary way of generating the necessary data.

- If a link comes back from a node chosen as a through movement, that link is designated as opposing a left turn.
- If no downstream nodes are found for a link, the upstream node is designated as a turnaround (such as a dead end, cul-de-sac, or turning circle).

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 must include a population topography data file 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.

Generate Evacuation Network

- 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:** All (100 percent) 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 system assumes a 12-hour loading period during which those who do evacuate will begin their trips. The percentage distribution of evacuees is assumed to follow this schedule:

Evacuation Hour	Percentage of Vehicle Trips (Evacuations) Begun
1	45.5
2	12.2
3	12.5
4	10.6
5	5.7
6	4.2
7	3.9
8	0.9
9	1.3

10	1.3
11	0.3
12	1.6

The rates in this table are based on timings reported for hurricane evacuations from Miami, New Orleans, and Mobile.^(a)

Although these rates were derived from reported departure times for hurricane evacuations, they are open to debate. Clearly, if 45.5 percent of a large metropolitan area begin their evacuations in the first hour, the resulting traffic jams would prevent most from leaving at all.

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, 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

Evacuation Hour	%	Discharge Required	Discharge Allowed	Vehicles Queued for Discharge
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 12 hours. Trip

(a) Rogers, G. O. October 1985. Decisions to Evacuate in Hurricanes: A Social-Psychological Model of Evacuation Decisions. Prepared for the Federal Emergency Management Agency by the University Center for Social and Urban Research, University of Pittsburgh.

Generate Evacuation Network

Generation continues loading over six more time periods (TP 13 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** (*Cnnn*INP.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** (*Cnnn*INP.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** (*Cnnn*EVACPARAM.DAT): a binary file that contains general default values for necessary roadway data that do not occur in the ROADS & TRAILS 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.*

IEMIS Menu: Select E (Model Library)

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

This section describes the Interactive Dynamic 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. Major results of the traffic simulation can be viewed graphically. Other results serve as input to other models, such as the 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. 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 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 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** 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 time 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.

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 Model Library 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 Dynamic Evacuation Model (IDYNEV) from the Model Library menu of IEMIS or Evacuation, from the Emergency Activities menu of IBS.

6.3.1 Starting an IDYNEV Work Session from IEMIS

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

- Modification of IDYNEV input data (through PREDYNGR)
- All other functions (model execution, selection of output reports, display of model results, site and case control, and file maintenance).

In its opening sequence of prompts, the IDYNEV portion of the system determines whether you will be modifying input data or performing other IDYNEV functions. The general flow of IDYNEV operations is depicted in Figure 6.1. The initial prompts are explained in the following sections.

Figure 6.1. General Flow of IDYNEV Operations in IEMIS

6.3.1 Selecting Graphic or Text Mode

After you select IDYNEV from the Model library menu, you will see the following prompt:

DO YOU WISH Graphic or Text Menus? (enter G or T):

- If you enter G, the succeeding menus (for selecting IDYNEV functions or for modifying input data) will include options for graphically displaying (and modifying) site and evacuation network information. (Section 6.5 explains a graphic means of updating some IDYNEV input data. Sections 6.6.3 explains how to graphically display model results.)
- If you enter T, the succeeding menus will NOT include any graphic options. Thus, a subset of IDYNEV tasks could be accomplished on a nongraphic terminal.

6.3.2 Selecting Evacuation Model Input or the IDYNEV Function Menu

After you select graphic or text menus, you will see the following prompt:

Do you wish to update evac model input? (enter Y or N):

Although not phrased as such, this prompt gives you a choice between modifying the input data set for the current case or selecting from a menu of other IDYNEV functions:

- If you enter Y (yes), you will begin a data entry sequence for modifying IDYNEV input (Section 6.4).
- If you enter N (no), 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.

6.3.3 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 respond Y to the prompt, "Do you wish to update evac model input?", your screen will move to the **PREDYNGR user interface**, which enables you to create or modify the input data to be used as input 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 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 an internal point from which vehicles enter the traffic network. An origin represents a population center.
- **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 -- PREprocessor for input to the DYNamic evacuation model with a GRaphics mode of data input.
- **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 Periods and Time Intervals**: 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 answer "Y" (YES) after the "Do you wish to update evac model input?" prompt, or select option A (Change Input Data) from the Evaluate Evac Model menu, PREDYNGR will greet you with the message and instruction shown in Figure 6.2. PREDYNGR gives you access to a set of **data forms** that specify all the data that serve as input to the evacuation model, IDYNEV.

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 lists the basic keystrokes used in interactions and data entry.

Welcome,

This is PREDYNGR, the interactive forms and graphics display preprocessor for DYNEV/TRAD. Unlike PREDYN, PREDYNGR uses Data entry-based forms as do other IEMIS subsystems.

Key	Function
-----	----------

F1	Backup to the field on the preceding line.
F2	Advance to the field on the next line.
F3	Backup horizontally to a preceding field.
F4	Advance horizontally to the next field.
RETURN	Enter the current field and advance to the next.
GOLD (F5)Z	Signal completion of the current form.
GOLD (F5)Y	Signal completion of the current form set.
HELP (F6)	Get help.

When you are ready to proceed, press <RETURN>.

Figure 6.2. Greetings Message Showing Entry Conventions for Changing the Input Data to the Evacuation Model

Dynamic Evacuation Model

When you press <RETURN> at the PREDYNGR greetings message, you will begin a layered series of input interactions as depicted in Figure 6.3. 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.

Figure 6.3 Overview of an IDYNEV Data Input Session

Modifying the Capacities and Free-Flow Speeds of Network Links. After you press <RETURN> at the PREDYNGR greeting, 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 percent = 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:

DO YOU WISH NETWORK CHANGES TO BE MADE PERMANENT? (Y OR N):

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 Model Library menu. Typing Y will display 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?

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 the new data records are being copied to disk. When the copying is complete, you will exit from PREDYNGR.

6.4.3 Choosing 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.

— Specifying a Subnetwork

The software described in Section 4 (Create New Site) 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 (of 1 to 255):

ENTER SUBNETWORK NUMBER (HIGHEST NUMBER IN USE IS 1:

If the "highest number in use" is greater than one, then 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.

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, or R). R

Enter time period.

Figure 6.4. Example of Time Periods Designation Form

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.).

— Choosing a Type of Data Form

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

ENTER FORM TYPE TO BE CREATED/EDITED, MENU, GRAPHICS, OR EXIT.

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

To 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.)

- Choose 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 E for exit. Then press <RETURN>. The system will ask you, "DO YOU WISH NETWORK CHANGES TO BE MADE PERMANENT?" (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 Library menu.)

Limited on-line HELP is available by pressing the HELP <F6> key.

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

— Choosing 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 with the following prompt:

ENTER FUNCTION TYPE (Add, DElete, Modify, Block change, DOne):

This prompt serves as a menu of the functional options that follow.

To specify the function type: Type A (or DE or M or B or DO). Then press <RETURN>.

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

- **Block changes:** 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.
- **Done:** When you select **Done**, the system checks to see if you are finished with this data input session:

ARE YOU FINISHED EDITING?

- If you answer N for "no," then the system will ask you to choose another time period and subnetwork (Section 6.4.3).
- If you answer Y for "yes," then the system will present the ending sequence of prompts, starting with, "**DO YOU WISH NETWORK CHANGES TO BE MADE PERMANENT?**" (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 for 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 for 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 no form 49 is 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 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 1/10000 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 1/10000 deg
 - 2nd four decimal places: an integer number of 1/100,000,000 deg

- * Radius: Enter a radius in miles (nnnn.nnnnnnnn)

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 percent to 9 percent.
- * 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.
- * Receiving nodes: Enter code 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 downstream receiving node.
- * 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 200); 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 of 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 code 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 several intervals. 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 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 percent radial evacuation without voluntary features. **This data is used only in context with an EESF traffic simulation that will be used in conjunction with a subsequent 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 permits 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. These data are 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

After you have answered the prompt

ENTER FORM TYPE TO BE CREATED/EDITED, MENU, GRAPHICS, OR EXIT.

by entering 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. 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 Utilities Guide).

2. Pick the map features that you want to display while modifying the evacuation model input.

Be sure to pick ROADS & TRAILS and POPULATION 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).

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 accessible 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. To keep track of where you are, you can enable the display of a normally invisible dialog screen that shows prompts for the operations:

Figure 6.8. Menu for Graphically Modifying IDYNEV Input Data

To show the dialog screen: Press the <DIALOG> key at the left of the top row of function keys. Examine the dialogue screen carefully; it sometimes continues to display old, possibly helpful dialogue. To redisplay the graphics, press the <DIALOG> key again.

— 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 bit pad and 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 will move the map so that it centers on a location of your choosing. The system will prompt you for a radius of the zoomed area

(in miles) and for longitude and latitude coordinates of the location. Enter these inputs carefully with commas (or spaces) between, and press <RETURN>. If the location that you want centered is somewhere on the displayed map, you can use the Coordinate Pick to determine its coordinates.

— Modifying Evacuation Model Input Data

Consider the following example for adding a link on a traffic network. (If you are using a bit pad and 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.

If you press the <DIALOG> key now, you will see a prompt to pick the downstream node.

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 IEMIS 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 answer N to the prompt, "Do you wish to update evac model input?", you will be turned over to the IDYNEV Function Selection menu. Sections 6.6.1 through 6.6.8 will explain the 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 Sel. 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: **Enter "Y" to execute this case: _**.

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: C101
 Site: MAINE YANKEE
 Selection Statistics WILL be generated.
 CASE IS SIMULATION
 Simulation start time = 0: 0
 Simulation covers 5 time periods with total time span of:
 0:18:30 Using the following time period breakdown:
 300 300 300 9900 0 0 0 0 0 0 0 0 0 0 0 0
 Enter "Y" to execute this case: _

Figure 6.11. Check List for IDYNEV Model Execution (Example)

6.6.2 Execute Evacuation - Batch

IDYNEV Function Sel. 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: Enter Y to execute this case: _

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 & TRAILS** 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 bit pad with 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 #	<-- Site and case
EXIT	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	
SET TIME STEP	
## HOURS, ## MINUTES	<-- Time Period label appears
TEXT DISPLAY ENABLED	when model results are
ICON DISPLAY ENABLED	run or stepped.
REFRESH SCREEN	

Figure 6.12. Graphics Menu for Display Results on Map

The following descriptions of the menu functions assume you have picked the menu item. 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); then pick CONTINUE. 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); press CONTINUE.

This option displays a report on the centroid, including the time period, time period duration, traffic flow rate, upstream and downstream nodes, and link information.

- **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 an area value.

- **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. Cumulative network statistics are available at each time step. Users at field workstations may need to press the **<DIALOG>** key at the top of the keyboard to view text results.

- **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.
- **SET 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).

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 Sel. 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 Sel. 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
Site Name _____
Base Site Node _____
Current Case _____ Based on Case _____
-----

```

Figure 6.14. Form for Change Case/Site (Example)

6.6.7 Perform File Maintenance

IDYNEV Function Sel. 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 IEMIS, 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 IEMIS 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 IEMIS 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.
- I'm sorry, but you seem to have selected at least one option that has not, as yet, been implemented. Be advised that the implementors of IEMIS will install these features shortly.
- If you have also selected options that ARE currently implemented, these functions will be performed at this time.

Note: Some of the above messages refer to IEMIS. It is possible that you could see some of these messages while performing IBS functions that are not part of IEMIS. The messages were defined prior to the IBS implementation and have not been rewritten to consider the expanded functions offered by IBS. The basic meaning of the messages probably is still correct.

Section 7

Model Library - Outdoor Sound Propagation Model

IEMIS Menu: Select **E** (Model Library)

Model Library Menu: Select **D** (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. 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/Site
- 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/Delete Site
- 7.11 Update Siren
- 7.12 List/Update Site Specs
- 7.13 Execute OSPM - Interactive
- 7.14 Execute OSPM - Batch
- 7.15 Perform File Maintenance
- 7.16 Delete Contours
- 7.17 Create a New Siren Type

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 Outdoor Sound Propagation Model (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/site specification form. 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 Model

With the information supplied in this section, you will be able to execute the Outdoor Sound Propagation Model and obtain model results. To use the Outdoor Sound Propagation Model 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 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 = **T**errain **A**nalysis **C**ell, the cell file format use 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 National 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
$ 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 **MYELEV** (consisting of **MYELEV.TAC** and **MYELEV.TAA**).

<u>LATLONXUTM Prompt</u>	<u>Your Response</u>
Enter Input File (.TAC):	MYELEV
Enter Output File (.TAC):	MYELEV 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>

3. Check the site directory to see that the UTM elevation files have been created.

For the example above, these would be **MYELEV_UTM.TAC** and **MYELEV_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 MYELEV (which would consist of MYELEV.TAC and MYELEV.TAA).

<u>CREATETAC Prompt</u>	<u>Your Response</u>
ENTER NAME OF TAC FILE :	MYELEVAT
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 MYELEVAT.TAC and MYELEVAT.TAA.

Using the Elevation and Impedance Data Sets in OSPM. To use the elevation and impedance data sets that you have created, set the Current Elevation Set by using IBS's Modify Current Model Environment function. Just enter the *filename* of the standard elevation data set for the site that you want to use. Note that changing the current site does NOT automatically change the current elevation data set. You must change the site before changing the elevation data set.

7.3 Using the OSPM Function Selection Menu

Model Library Menu: Select D (Outdoor Sound Propagation Model)

When you select the Outdoor Sound Propagation Model from the Model Library menu, you will first see a prompt asking you whether you would like graphic or text menus:

DO YOU WISH Graphic or Text Menus? (enter G or T):

Depending on your selection of Graphic (G) or Text (T) menus, 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/SITE	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

Figure 7.1. OSPM Top-Level Menu - Graphic Mode

```

-----
                                OSPM FUNCTION SELECTION

Selection      Selection
Designator

  A      List Site Specs
  B      Execute OSPM - Interactive
  C      Execute OSPM - Batch
  D      Change Case/Site
  E      Perform File Maintenance
  F      Create a New Siren Type      (Section 7.17)
  X      Exit

Enter Selection Designator: __
-----
  
```

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 letter and press <RETURN>.

The Outdoor Sound Propagation Model functions are described in Sections 7.4 through 7.16, 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 F 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:

Do you want to save the OSPM updates made during this session? (Y/N) __

- 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/Site

OSPM Text Menu: Select D (Change Case/Site)

When you pick CHANGE CASE/SITE from the OSPM menu, the system displays the form shown in Figure 7.3 used for switching to a different site name or changing the OSPM case number. The site name in the form refers to the current site.

```
-----  
OUTDOOR SOUND PROPAGATION MODEL  
Site Name:  MAINE YANKEE  
Base Site Node:  FEM02  
Current Case:  003    Based on Case:  000  
-----
```

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: Enter a four-character site code corresponding to a site that has been created for IBS.
- * Base Site Node: Enter a four-character code corresponding to the current node used for this base site. (*Information not used.*)
- * Current 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.

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.

- * Based on Case: 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.

Changing the Elevation Data Set. 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.) To change the elevation data set, use Modify Current Model Environment function (described in Section 1).

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 4-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. The menu area will display the case number, frequency, and lowest contour value; and in the legend area a map scale. When you are through viewing the contours, pick EXIT to return to the OSPM menu. You can then delete the contours with 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.
3. Pick CONTINUE.

The system will display the sound level report similar to the following one on the dialogue screen for a moment. You can press the <DIALOGUE> key to review the information again.

```
-----  
Sound Levels for location Lon,Lat  -69.586  44.008  
  
5.54      DBA  
3.01      DBC  
0.000E+00 64Hz  
0.000E+00 128Hz  
0.000E+00 512Hz  
0.576E-02 1 KHz  
0.518E-06 2 KHz  
0.000E+00 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.SITES.site]filename_UTM.TAC* in the site directory, where "site" is the IBS site code and "filename" is the name of the current elevation data set--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.)

```
EXIT  (SAVE TAC FILE CHANGES)
QUIT  (DEL TAC FILE CHANGES)
HELP
CANCEL
CONTINUE

MODIFY ELEVATION POINT

MODIFY GROUPED ELEVATION POINTS

RETURN ELEVATION
RETURN SLOPE
ENABLE ELEVATION RESOLUTION
DISABLE ELEVATION RESOLUTION

DISPLAY ELEVATION STATISTICS
DISPLAY ATTRIBUTES

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

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 available from the National Map database, the impedance data are needed only by OSPM and must be generated outside of IEMIS. 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.

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.

3 of 3

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 RESOLUTION:** Shows the resolution of the impedance data by drawing white grid lines around the impedance data cells.

To show the impedance data grid, just pick ENABLE IMPEDANCE RESOLUTION. If the current display 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:

COLUMNS : XXXX
ROWS : XXXX

DO YOU STILL WISH TO PLOT THE CURRENT DENSITY?

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:** Turns off the capability to display 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).

- **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:

<u>#</u>	<u>MAJOR</u>	<u>MINOR</u>	<u>PARAM</u>	<u>DESCRIPTION</u>
1	7	150	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 7.10. 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. (Do NOT attempt to enter anything in the siren number fields: that can change the display.)

The siren names will scroll up to reveal more sirens (if any are present).

2. When you have found the desired siren type, enter it in the appropriate field.
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): 1
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 7.11. Form for Change Default Siren

7.10.2 Place 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/Delete 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 7.12. 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 7.13).

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) Y

Do you want to look for other sirens at this location? (Y/N) N

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/Delete 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/ Delete 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.
- * **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: 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

Do you want to look for other sirens at this location? (Y/N) N
-----

```

Figure 7.14. Form for Updating a Siren (Example)

7.12 List Site Specs

OSPM Text Menu: Select A (List 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	<u>232.0</u>	(deg clockwise from north)
Ground-Level Wind Speed	<u>3.6</u>	(mph)
Reference Higher-Elevation Wind Speed	<u>4.5</u>	(mph)
Ground-Level Temperature	<u>74.5</u>	(degrees F.)
Higher-Elevation Temperature	<u>73.4</u>	(degrees F.)
Ground-Level Elevation	<u>35.0</u>	(feet)
Reference Higher-Elevation	<u>197.2</u>	(feet)
Relative Humidity	<u>64.0</u>	(%)
Barometric Pressure	<u>764.000</u>	(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 **B** (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.
- The system will prompt

Do you wish to save the OSPM updates made during this session? (Y?N)

If you answer Y (yes) the system will display the following message:

Saving Updates. Press <RETURN> to continue.

The current OSPM case has the following attributes:

Site: MAINE YANKEE

OSPM Case No: C000

BASED ON ELEVATION SET: MAINEL

MIN/MAX Lon: -69.90766393 -69.48453272

MIN/MAX Lat 43.79725911 43.10394023

A total of 8 sirens will be modeled.

Enter "Y" to execute this case: Y

Figure 7.16. 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 7.17. 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 F (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 F. 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.

Model Library - Outdoor Sound Propagation Model

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 7.18. 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 **ROTATING** or **OMNI-DIRECTIONAL**.
- * Reference Sound Pressure Values: For each octave-band frequency, enter a value for sound pressure at 100 ft.

Section 8

Model Library - CHEMS Models

IEMIS Menu: Select E (Model Library)

Model Library Menu: Select E (CHEMS Models)

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**
- **Fire Model**
- **Dispersion Model**

8.2 What You Need to Know to Use the CHEMS Models

- **Overview of the Explosion Model**
- **Limitations of Using the Explosion Model**
- **Overview of the Fire Model**
- **Limitations of Using the Fire Model**
- **Overview of the Dispersion Model**
- **Limitations of Using the Dispersion Model**

8.3 Using the CHEMS Model Menu

8.4 Update Case Data

8.5 Select Spill Location

8.6 Change Case/Site

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 percent 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 percent and 10 percent. 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. IEMIS 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 What You Need to Know to Use 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 IEMIS, 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

$$m_{\text{TNT}} = m_{\text{Vapor}} \cdot \frac{(\text{Heat of Combustion of Chemical})}{(\text{Heat of Combustion of TNT})} \quad (8.1)$$

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 percent and 10 percent of the mass of vapor released in the accident participates in the explosions phenomenon. Based on this, a conservative factor of 10 percent 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 psig). 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 What You Need to Know to Use 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

Model Library Menu: Select E (CHEMS Models)

When you select CHEMS from the Model Library menu, you will first see a prompt asking you if you would like graphic or text menus:

DO YOU WISH Graphic or Text Menus? (enter G or T):

Depending on your selection of Graphic (G) or Text (T) menus, you will see one of the menus of the CHEMS Models subsystem (Figure 8.3 or 8.4).

EXIT	
HELP	
CANCEL	
CONTINUE	<u>Explained in</u> <u>Section</u>
UPDATE CASE DATA	8.4
SELECT SPILL LOCATION	8.5
CHANGE CASE/SITE	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 Site Name, 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. CHEMS 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 A (Modify Current CHEMSHAZ Case Data)

Graphics Menu: Select **Update Case Data**

When you pick this option from the CHEMS Models menu, IEMIS displays a text menu of case data modification selections as shown below.

```

-----
                                CHEMSHAZ MODELS
                                CHANGE MODEL PARAMETERS

Selection      Selection
Designator

      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 Selection A (CHEMICAL SPILLED)

When you select this option, a form is presented providing you five options for specifying a Chemical Identification field:

```

Headings
1: CHRIS Chemical Code
2: Chemical Name
3: CAS #
4: UN / DOT #
5: STCC #
Enter a Selection Number : _
Enter Chemical Identification : _
  
```

CHRIS chemical code is a 3-letter alphabet code used for each of approximately 1200 chemicals by the U.S. Coast Guard in its Chemical Hazard Response Information System (CHRIS). The CAS number is the number assigned by the Chemical Abstracts Service 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. 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. 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

Units Available psi inches of water mm Hg atm bars Pascal	==>	Unit Codes 1 psi 2 inches of water 3 mm Hg 4 atm 5 bars 6 Pascal
--	-----	---

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 Selection B (TIME OF SPILL / CURRENT TIME)

When you select this option, 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 Selection C (ENVIRONMENTAL INPUTS)

When you select this option, a menu is presented as shown below.

```
-----
                                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

8.4.3.1 Selection A (Atmospheric Conditions)

When you select this option, 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.

8.4.3.2 Selection B (Ground Conditions)

When you select this option, 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.

8.4.3.3 Selection C (Water Conditions)

When you select this option, 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 Selection D (MODE OF OPERATION)

When you select this option, a menu is presented as shown in Figure 8.7.

```
-----  
                        OPERATION MODE  
                        Choose Mode of Operation  
                        1. Emergency Mode  
                        2. Contingency Mode  
                        Choice: _  
-----
```

Figure 8.7. 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 description 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 Selection E (MODEL SELECTION / MODEL DEPENDENT INPUTS)

When you select this option, a menu is presented showing the current chemical to be modeled and allowing you to choose one of several models (1 = FIRE RADIATION, 2 = VAPOR DISPERSION, 3 = WATER DISPERSION, and 4 = 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 4 (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 Selection F (SOURCE INPUTS)

When you select this option, a menu entitled CHEMICAL RELEASE ENVIRONMENTS is presented with seven options shown as follows:

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 8.8. 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 psig, 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, the menu presented provides four options for showing the tabular hazard distances. You may select feet, meters, kilometers, or miles.

8.4.8 Selection X (EXIT)

On exit, you will be prompted to change the case title.

- * **Case Title:** 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

<u>Off-Site Target Description</u>	<u>Incident Flux, kW/m²</u>
1) Outdoor areas occupied by 20 or more persons during normal use, such as beaches, playgrounds, outdoor theatres, 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: 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	12.6
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/Site

CHEMS Model Text Menu: Select C (Change Site Name, Case Number)

When you pick this option (CHANGE CASE/SITE) from the CHEMS Model menu, IEMIS displays the case/site specification form (example shown in Figure 8.9). The site name in the form refers to the current site. The case number shown for the model is the "current case."

```
-----
                        CHEMSHAZ MODEL
                        -----

CHEMSHAZ Model Case Number: 000  Based on: 000
Site Name:MAIN
-----
```

Figure 8.9. Form for Changing Site and Case Number

Use the arrow keys, <RETURN>, and <GOLD> <Z> to edit the data in the following fields of the form:

- * CHEMS Model Case Number: Enter one of 1000 possible case numbers from 000 to 999. Always enter all three digits of the case number in the case number field.

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 Based on: field.

- * Based on: 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. Always enter all three digits into the field.
- * Site Name: Enter a four-character site code corresponding to a site that has been created for IBS.

Press <GOLD> <Z> to accept the data and exit the form.

If you change Site Name, you should change elevation and population data sets, as well. You will be presented with a screen of messages reminding you of this and providing information about defaults applied by the system when you change sites.

8.7 Base Map Display - 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.

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 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 8.10, 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 8.10. 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 8.11. 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 8.11. Menu for Performing File Maintenance

Section 9

Model Library - Expert System Model (TIMM)

IEMIS Menu: Select **E** (Model Library)
Model Library Menu: Select **F** (Expert Systems Model)

TIMM^(a) is the IEMIS implementation of The Intelligent Machine Model (TIMM). TIMM is a computer program that allows you to build expert systems for your own applications.

Note: TIMM is currently accessible only on the main system at FEMA headquarters. TIMM is not distributed by PNL. Contact FEMA for further information.

(a) TIMM is a registered trademark of General Research Corporation, Santa Barbara, California.

Section 10

Model Library - Integrated Modeling (EESF)

IEMIS Menu: Select E (Model Library)

Model Library Menu: Select G (Integrated Modeling)

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.

- 10.1 Description of EESF
- 10.2 What You Need to Know to Use EESF
- 10.3 Obtaining On-Line HELP with Data Input
- 10.4 Starting an EESF Work Session
- 10.5 Obtaining and Using the Top-Level EESF Menu
- 10.6 Display Base Map - Graphics Menu Only
- 10.7 Modify Meteorology/Dose Model Input Data
- 10.8 Model Evacuation - Batch
- 10.9 Model Evacuation - Interactive
- 10.10 Modify Evacuation Start Times for Dose Calculations
- 10.11 Execute Mesorad - Batch
- 10.12 Execute Mesorad - Interactive
- 10.13 Print Reports
- 10.14 Display Reports - On Terminal
- 10.15 Display Map-Related Graphics - Graphics Menu Only
- 10.16 Case File Directory and Optional Delete
- 10.17 Change Site Name and Case Numbers
- 10.18 Concurrent Execution of Mesorad and IDYNEV
- 10.19 If You Have Problems
- 10.20 Status and Diagnostic Messages
- 10.21 References

10.1 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.

10.1.1 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.

IEMIS 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 10.4).

10.1.2 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 10.21, **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 10.18 describes the concurrent execution and display of results from the EESF models.

10.1.3 Display of 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.

10.2 What You Need to Know to Use 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 10.3). Section 10.21 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.

10.3 Obtaining On-Line HELP with Data Input

If you have a question about a particular item on a menu or on a data input form, you can press the HELP key <F6> above the numeric keypad.

- o Pressing <HELP> the **first** time produces help information pertaining to the **current input field**.
- o 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 can be presented. Press <RETURN> to page through the available information and get back to the data form or menu.

10.4 Starting an EESF Work Session

Model Library Menu: Select G (Integrated Modeling)

An EESF work session is a continuous period of time during which you use EESF. The session starts when you select EESF from the IEMIS Model Library menu and ends when you return to the Model Library menu.

Overview. EESF incorporates the data input process of the evacuation model (IDYNEV), so (like the evacuation model), the program flow for functions in EESF is generally divided into two separate parts:

- Modification of evacuation input data (*through PREDYNGR--the evacuation model data input program*)
- All other functions (*Mesorad input, Mesorad and IDYNEV model execution, selection of output reports, display of model results, site and case control, and file maintenance*).

In its opening sequence of prompts, EESF determines whether you will be modifying IDYNEV input data or performing other EESF functions. The general flow of EESF operations is depicted in Figure 10.1. The initial prompts are explained in the following sections.

Figure 10.1. General Flow of EESF Operations in IEMIS

10.4.1 Selecting Graphic or Text Mode

After you select Integrated Modeling (EESF) from the Model library menu, you will see the following prompt:

DO YOU WISH Graphic or Text Menus? (enter G or T):

- If you enter G, 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 enter T, the succeeding menus will NOT include any graphic options. Thus, a subset of EESF tasks could be accomplished on a nongraphic terminal.

10.4.2 Selecting Evacuation Model Input or the EESF Function Menu

After you select graphic or text menus, you will see the following prompt:

Do you wish to update evac model input? (enter Y or N):

Although not phrased as such, this prompt gives you a choice between modifying the IDYNEV input data set for the current case or selecting from the main menu(s) of EESF functions:

- If you enter Y (yes), you will begin a data entry sequence for modifying evacuation model input (Section 6.4). When you finish modifying the IDYNEV input data, you will return to the Model Library menu.

The data input process for the evacuation model involves using the **PREDYNGR user interface**, a program that enables you to create or modify the data to be used as input for the evacuation model. The input data consist largely of items that describe the condition of the IDYNEV roadway network. As background for users who are unfamiliar with the evacuation model, Section 6.4.1 supplies a list of key terminology for the model. Section 6.4.2 (**Starting a Session with PREDYNGR**) and subsequent 6.4.X sections describe the model input data and the actual process of modifying the data. Section 6.5 explains a graphics-based method for changing the evacuation model input data.

- If you enter N (no), the system will present a menu of EESF functions, including base map display, Mesorad input modification, execution of both models, selection of output reports, display of model results, site and case control, and file maintenance. Section 10.5 explains the options of the EESF Function Selection menu(s).

10.5 Obtaining and Using the Top-Level EESF Menu

When you answer N to the prompt, Do you wish to update evac model input?, you will be turned over to one of the EESF top-level menus shown in Figures 10.2 and 10.3. Sections 10.5 through 10.18 explain how to use the various functions that may be requested from the EESF top-level menu(s).

The text mode menu in Figure 10.3 contains only the basic IDYNEV and Mesorad functions (except for IDYNEV input data modification). The graphics mode menu in Figure 10.2 includes graphics functions as well: 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	10.6
B	Modify Meteorological/Dose Model Input Data	10.7
C	Model Evacuation - Batch	10.8
D	Model Evacuation - Interactive	10.9
E	Modify Evacuation Start Time for Dose Calculations	10.10
F	Execute Mesorad - Batch	10.11
G	Execute Mesorad - Interactive	10.12
H	Print Reports	10.13
I	Display Reports	10.14
J	Display Map-Related Graphics	10.15
K	Case File Directory and Optional Delete	10.16
L	Change Site Name or Case Numbers	10.17
M	Concurrent Execution of Mesorad And Idynev	10.18
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 10.2. 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 Site Name or 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 10.3. EESF Top-Level Menu - Text Mode

It is normal to begin a session by selecting the site name or case numbers before selecting any of the other listed options. This avoids inadvertent creation of new cases. See Section 10.17, **Change Site Name or 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 10.6 through 10.18.

10.6 Display Base Map - Graphics Menu Only

EESF Top-Level Menu: Select A (*graphic mode menu*)

This option produces the standard IEMIS menu of Map Display functions. For a complete description of these functions, refer to the *IBS Utilities Guide*, MPDISPLAY.

10.7 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 10.4 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, please 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 10.4. Menu for Selecting Forms for Changing Parameters in the Meteorological/Dose Model - Text Mode

10.8 Model Evacuation - Batch

EESF Top-Level Menu: Select **B** (*text mode menu*) or **C** (*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:

Enter Y to execute this case: _

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, IEMIS executes the model separate from your current interactions with IEMIS. 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 IEMIS. 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 10.21) for specific information on the evacuation model.

10.9 Model Evacuation - Interactive

EESF Top-Level Menu: Select C (*text mode menu*) or D (*graphic mode menu*)

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

Enter "Y" to execute this case: _

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, IEMIS executes the model as part of your current process; you will have to wait until the model is finished before you can continue any IEMIS 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 10.8) 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 IEMIS 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, IEMIS 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 10.5 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 10.5. Example of an Evacuation Summary Report

10.10 Modify Evacuation Start Times for Dose Calculations

EESF Top-Level Menu: Select D (*text mode menu*) or E (*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 10.6. 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:  HH  MM
                                                            00  00
Apply shelter factors (Y or N):  _
Vehicle Shelter Factor:         0.00
Sector Shelter Factor:         0.00
-----
```

Figure 10.6. 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.

10.11 Execute Mesorad - Batch

EESF Top-Level Menu: Select E (*text mode menu*) or F (*graphic 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:

Enter Y to execute this case: _

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. Although you cannot interact with the met/dose model program while it is running, you can simultaneously display the graphic results of the model. 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 10.21) for specific information on the met/dose model.

10.12 Execute Mesorad - Interactive

EESF Top-Level Menu: Select F (*text mode menu*) or G (*graphic 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:

Enter Y to execute this case: _

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. Although you cannot interact with the met/dose model program while it is running, you can simultaneously display the graphic results of the model. For large cases, execution in batch mode (Section 10.11) may be more convenient.

Refer to Section 4 and to the Mesorad source documentation (listed in Section 10.21) for specific information on the met/dose model.

10.13 Print Reports

EESF Top-Level Menu: Select G (*text mode menu*) or H (*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 10.7. (Refer also to Section 10.14, **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 10.14).
- * 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.

Figure 10.8 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 10.7. 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 THYROID REM	WHOLE BODY REM	CUMULATIVE 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 10.8. Example Page of EESF Dose Report Printed Output

10.14 Display Reports (On Terminal)

EESF Top-Level Menu: Select H (*text mode menu*) or I (*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 10.7. The report options are identical to those for printing reports. (For descriptions of the reports, refer to Section 10.13, **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 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 the report(s) on a printer
D	Display the report on the terminal screen.
X	Exit without printing or displaying

Note: Some IBS installations have modified the underlying program that performs this process to support local devices. Consult your local System Manager if you have questions.

Figure 10.9 is an example of EESF (Mesorad) displayed output.

Model Library - Integrated Modeling (EESF)

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 10.9. Example Page of EESF Dose Report Displayed Output

10.15 Display Map-Related Graphics - Graphics Menu Only

EESF Top-Level Menu: Select J (*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 during or after the met/dose model runs. That is, you can Model Mesorad - Interactive and then monitor progress with map-related graphics as the model is running. For example, it is possible to watch changes in such items as plume growth and wind indicators as the model completes each simulation time step.

Picking DISPLAY MAP-RELATED GRAPHICS from the EESF graphic-mode menu will display a menu of EESF Graphical Mapping Options (Figure 10.10) 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 10.10. 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 10.10) and then press <GOLD> <Z>.

EESF then calls on the IEMIS 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.

IEMIS then reads the necessary model output files, draws the map background features, and presents the EESF graphics display menu shown in Figure 10.11.

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 10.11.

Site Met/Evac ###/###	<-- Site and Met/Evac case
EXIT	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	
RUN EVAC OUTPUT (EO)	
RESTART EO	
STEP EO FORWARD	
STEP EO BACKWARD	
SET EO TIME STEP	
REFRESH EO NETWORK	
## HOURS, ## MINUTES	<-- IDYNEV Time Period label
TEXT DISPLAY ENABLED	appears when model output
ICON DISPLAY ENABLED	results are run or stepped.
STEP MET/DOSE OUTPUT	
REPORT DOSE	
DISPLAY LEGEND	
MMM/DD HH:MM	<-- Start date and time step
	time display for Mesorad

Figure 10.11. EESF Graphics Display 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 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 main 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 period, time period duration, traffic flow rate, upstream and downstream nodes, and link information.

- **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.
- **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. Cumulative network statistics are available at each time step. Users at field workstations may need to press the **<DIALOG>** key at the top of the keyboard to view text results.

- **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.
- **STEP EO BACKWARD:** This option displays the traffic network results for the previous time step.
- **SET 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.

10.16 Case File Directory and Optional Delete

EESF Top-Level Menu: Select I (*text mode menu*) or K (*graphic mode menu*)

This option will display the menu shown in Figure 10.12. 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.

CASE FILE FUNCTIONS	
Function Code	Function Description
A	Display Directory of Evacuation Cases
B	Delete Case Output for a Specified Evacuation Case
C	Delete Case Input and Output for a Specified Evacuation Case
D	Display Directory of MET/DOSE Cases
E	Delete Case Output for a Specified MET/DOSE Case
F	Delete Case Input and Output for a Specified MET/DOSE Case
X	Exit to Previous Menu

Enter function code corresponding to the function you wish to execute

Figure 10.12. Menu for Case File Directory and Optional Delete

10.17 Change Site Name or Case Numbers

EESF Top-Level Menu: Select J (*text mode menu*) or L (*graphic mode menu*)

This option displays the site/case selection form (example shown in Figure 10.13). 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 10.16, **Case File Directory and Optional Delete**).

The data fields of the form in Figure 10.13 follow.

- * **Site Name:** Enter the 4-character name for the site that you desire.

The convention is to use the first four letters of the actual facility name, whenever possible. If the site you desire does not exist on the system, contact the System Manager. (You may be able to obtain a list of site names by typing SITES at the \$ prompt before starting IEMIS: \$ SITES <RETURN>.) 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.)

- **Base Site Node:** This display-only field shows the name of the computer system node on which the current site resides.
- * **Evacuation Model Case Number** and **Met/dose Model Case Number:** The first of the two case numbers shown for each model is called 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 Based on case).
- * **Based on:** The second or Based on case 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.

EESF

Site Name: MAIN
Base Site Node: FEMAO2
Evacuation Model Case Number: 101 Based on: 000
Met/dose Model Case Number: 817 Based on: 000

Figure 10.13. Form for Change Site Name, Case Numbers (Example)

10.18 Concurrent Execution of Mesorad and IDYNEV

EESF Top-Level Menu: Select **M** (*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 break-points 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).

10.18.1 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--**Enter Y to execute this case.**
- 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--**Enter Y to execute this case.**
- 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).

The system then creates some files that you will need later and runs the evacuation model through the first time period.

- 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 10.10.
- 3. Enter an evacuation offset time and shelter factors, as desired.
- EESF then displays a message--**INITIALIZING MESORAD**--and takes a few minutes to set up the models for starting concurrent execution and display.

After completing this initial sequence, EESF displays a menu of Concurrent Execution Options. These options are explained in the following section.

10.18.2 Using the Concurrent Execution Options Menu

The Concurrent Execution Options menu appears as shown in Figure 10.14. 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 10.18.3.

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 10.14. 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 10.15. 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 10.15.

```

-----
                        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) 01:00

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 10.15. 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 10.16. 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 10.16. 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, IEMIS will:
 - display the EESF Graphical Mapping Options menu for input. This menu is shown and explained in Section 10.15, **Display Map-Related Graphics**. (Exiting from this menu with <GOLD> <Y> will return to the Concurrent Execution Options menu in Figure 10.14.)
 - 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 Graphics Display Menu shown in Figure 10.10 in Section 10.15, **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.

10.18.3 **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 10.15).

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 **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 10.10, **Modify Evacuation Start Times for Dose Calculations**.)

5. Enter an evacuation offset time or sheltering factors, as desired.

EESF presents the **Graphical Mapping Options** menu for specifying which output results are to be displayed. (Use of this menu is explained in Section 10.15, **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:

 IEMIS is now performing the following function:

CREATING A MAP-BASED GRAPHICAL DISPLAY

 Processing evacuation model input file (path name)
 Processing evacuation model output file (path name)

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 10.15, **Display Map-Related Graphics**).

EESF then displays the selected model results overlaid on the site map. One set of results is displayed for each valid breakpoint until the model reach the Next breakpoint, selected in Step 2.

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.)

10.19 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 10.3.
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.

10.20 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.

Under certain circumstances, the following diagnostic messages will appear. The 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".
- 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".
- THE EVACUATION CASE NUMBER THAT YOU SPECIFIED FOR A BASE CASE DOES NOT EXIST.
- PLEASE TRY AGAIN.
- The Met/Dose Model has completed processing.
- The Met/Dose model was initiated at a previous time and is still running.
- I'm sorry, but with the current design of EESF, you'll have to wait until it completes before you can start another run.
- 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.

- **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 EESF 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 EESF 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.**
- **We have a problem here...There doesn't seem to be any met/dose input data available for the current case. EESF needs to access input data in order to display output. Please generate input data for the current case (or specify a different case) and try your request again.**
- **I'm sorry, but there doesn't seem to be met/dose data generated yet for your current met/dose case. Perhaps you meant to select the "Model Meteorology/Dose" option in the EESF Functions menu and just forgot to do so. If this is the case, make the selection and try again. On the other hand, the problem may be that the computer has simply not had time to generate any data yet. If you suspect that this is true, simply wait a few seconds and then try the "Display Map-related Graphics" again. If you continue to have problems, contact the System Manager, give him your User Name and the name of the site you are currently working with, and describe the problem to him.**
- **I'm sorry, but you seem to have selected at least one option that has not, as yet, been implemented. Be advised that the implementors of EESF will install these features shortly.**
- **If you have also selected options that ARE currently implemented, these functions will be performed at this time.**
- **You have specified a time interval for which no output exists. EESF cannot comply with your request; please specify a different starting and/or ending time that is more consistent with the current met/dose case.**
- **EESF has found it necessary to display graphical map-related output for a time interval that is different than that specified. The information will be displayed starting and/or ending at the time that was specified for met/dose modeling.**

10.21 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 20814.

Ramsdell, J. V., Athey, G. F., and Glantz, C. S. 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., Athey, G. F., Bander, T. J., and Sherpelz, R. I. 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., Bander, T. J., Athey, G. F., and Ramsdell, J. V. 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., Andrews, B., Davila, M., and Yedlin, M. 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.

Appendix A

Technological Hazards Support Module

The following selections on the Technological Hazards Support Module menu are not implemented at this time.

- REP Program Database
- Hazardous Material Database

DISTRIBUTION

No. of
Copies

OFFSITE

- 2 DOE/Office of Scientific and
Technical Information
- 5 Federal Emergency Management
Agency
Attn: Ed Corvi
500 C Street SW
Washington, DC 20472
- Federal Emergency Management
Agency
Attn: Ralph Sinnot
500 C St. SW
Washington, D.C. 20472
- Madhu Beriwal
Innovative Emergency Management
2351 Energy Drive, Suite 3009
Baton Rouge, LA 70808
- COL Lawrence E. Skelly
United States Army Nuclear and
Chemical Agency
7500 Backlick Road, Building 5073
Springfield, VA 22150-3198
- John Bower
Bower Software Services
104 E. 36th Place
Kennewick, WA 99337

No. of
Copies

ONSITE

- 1 DOE Richland Field Office
R. B. Goranson
- 26 Pacific Northwest Laboratory
B. M. Bailey (10)
M. J. Burford
T. R. Downing
S. W. Matsumoto (3)
W. D. Millard
S. L. Savelli
E. E. Schrank
J. R. Williams
C. Winters (3)
Publishing Coordination
Technical Report Files (3)

DATE

FILMED

44 / 7 / 94

END