structures library

UG-0022

DRYDOCK ANALYSIS SYSTEM (DAS) USER'S GUIDE

BETA VERSION

STATIC ANALYSIS OF DRYDOCK STRUCTURES

by

Frank R. Johnson and T.E. Gillum

Naval Civil Engineering Laboratory
Port Hueneme, California 93043-5003

September 1989

DISCLAIMER

PREFACE

The purpose of this report is to present technical information. The information contained in this report is technical in nature. It does not represent the policy of the Naval Civil Engineering Laboratory, the Naval Facilities Engineering Command, the Assistant Commander of Engineering and Design, or the Director of Engineering System Division. It is one of a series designed to provide technical information and documentation for studies conducted in support of the Naval Facilities Engineering Command Graphics Engineering and Mapping System, Engineering Microcomputer Graphics, Computer-Aided Design, Cost Engineering System, and Guide Specifications System as they relate to the Naval Facilities System and life cycle facilities management.

CONTENTS

INTRODUCTION

DAS (Drydock Analysis System) is a collection of FORTRAN programs used to statically analyze drydock structures constructed of reinforced concrete. Preprocessor programs allow graphic creation of a geometric model with its boundary and loading conditions. The SAP4 linear finite element analysis program is used to analyze the drydock structure based upon plane strain analysis theory. Postprocessing programs produce displaced shapes and graphic interpretations of stress results. The overall process is activated through a set of menu selections; the programs reuse of the basic geometric model for several different load cases.

CAPABILITIES

DAS creates a symmetrical (about the Z axis) two-dimensional cross section model for linear finite element analysis. Displacements are computed for each node in the Y and Z directions. Stresses are computed for the two-dimensional plane strain finite elements with respect to the X, Y, Z SHEAR, MAXIMUM PRINCIPLE, and MINIMUM PRINCIPLE orientations.

DAS handles distributed loads located on the outside walls, inside walls, and floor of the drydock cross section model. Ship block loads on the floor of the drydock are modeled with concentrated loads. An unlimited number of load cases can be analyzed by producing various load input files. Boundary conditions constrain the structure based upon a spring constant computed from the soil bearing capacity input by the user. DAS does not account for temperature effects but does incorporate the weight of the structure.

The DRYDOCK ANALYSIS SYSTEM (DAS) is a static analysis package of FORTRAN programs for IBM-PC/AT and compatible microcomputers. Models may be created three ways: (1) from answering displayed questions, (2) from a question log file containing responses to the displayed questions, and (3) by preparing the question log file using Appendix A and a screen editor. If an error occurs during the interactive question session, the user can exit DAS to edit the log file using a screen editor to omit questions that have been answered correctly. The log file also permits the user to build a library of drydock models. The model is complete when the geometry, force, and boundary (spring) conditions are defined.

Analysis is performed using the SAP4 linear structural analysis program. After the user defines the model loading conditions and boundary conditions, the software prepares the SAP4 data input file. The analysis step follows. Depending upon the model complexity (number of nodes and elements), this step might require more than an hour to complete. At least 10 MB of hard disk space are required for this step.

In general, the DRYDOCK programs divide the display into a MODEL DISPLAY section and a MENU section. The MENU section, the lower quarter of the screen, displays the menus and prompts available at the various steps of the DAS process and is the entry area for requested data input. The user will normally select a menu number and will only press the entry key after positioning the cursor to a desired position in the MODEL DISPLAY. The arrow keys on the key pad allow the user to position the cursor. The <INS> key toggles between fast and slow, which is actually a sequence of large and small steps. Large steps (fast) will not normally place the cursor accurately to select a given element or node.

Input data are normally entered in the MENU section of the screen. Things that are generally integer in nature, such as the number of regions or number of elements in a region, are entered without a decimal point. Those that normally might have fractional parts associated, such as forces and material strengths, must be entered with a decimal point. Editing the prompt line command is accomplished by backspacing and retyping the command, yet, once the enter (return) key is pressed changes may not be made (i.e., there is not a command line buffer). In the case of MODEL GENERATION a text editor may be used to make corrections to the model file before proceeding with MESH GENERATION. For forces, strengths, etc., standard recourse is to return to the previous menu and restart that procedure.

The geometric model and loading condition files can be saved for future certification requirements. DAS, using the log file, will store this information in a file that can be used at a later date. In this manner a collection of drydock cross sections can be made to save time and effort for future analysis requirements. The same comments apply to the boundary condition and load condition information.

SOLUTION METHODS

The drydock unit thickness cross section is analyzed using the finite element method. Plane strain quadrilateral elements are used. The following assumptions are implied with this analysis method:

- Long prismatical body
- Loading perpendicular to the longitudinal axis
- Loading does not vary with the length
- All cross sections are in same state or condition
- No axial displacement of ends, midsection, or any section

The maximum normal stress failure theory is used for interpretation of stress results. Tensile (positive) and compressive (negative) thresholds are defined with respect to concrete cracking, steel yielding, and concrete crushing. The positive and negative principle stress values are evaluated with respect to these threshold values.

PROBLEM DATA PREPARATION INSTRUCTIONS

The drydock analysis system (DAS) is invoked by typing the following command:

DAS <model name> <load case>

"Model name" is the name of the drydock model data file that will be prepared as the user responds to the prompts described here after. Alternatively, the user may desire to prepare this file or edit a previously prepared file using a screen editor. Appendix A provides the format for the model data file.

"Load case" is the name of the drydock load data file that must be prepared using a screen editor before starting the DAS procedure. The format for this file is described in Appendix B.

The batch utility screen shown below is used to tie together the four DAS programs. DAS requires two separate system configurations for operation. The batch utility automatically configures and reboots the system for the specified option selected. The pre- and post-processing programs (options 1 and 4) require the Graphic Kernel System (GKS) configuration, while the SAP4 finite element analysis (option 3) requires the full 640K byte memory configuration. The user must also ensure that there are at least 10 MB of hard disk space available for the analysis program during execution. The following menu will be displayed in response to the DAS command:

DRYDOCK ANALYSIS SYSTEM

BATCH UTILITY
Version 1.0

1. Model Generation

2. Format input data file for Static Analysis

3. Static Analysis

4. Result Processing

5. Exit and Return the system to standard configuration

6. Quit

ENTER:

The six choices are:

"Model Generation" (enter 1) will lead the user through a conversational session to create the drydock analysis model. The model includes the drydock cross section geometry, the loading conditions, and the boundary conditions. There is also a feature to review the model and to prepare a plot for the certification report. The review feature is especially helpful if the user chooses to prepare the model data file via a screen editor, or to utilize a file previously created using the conversational model generation feature.

"Format input data file for Static Analysis" (enter 2) will convert the model data base (consisting of several files) to a single file with the format required by the analysis program. The user is required to validate the default material parameters that will be used in this analysis. The results of this program may be reviewed using a screen editor and the SAP4.DAT file. Users familiar with the SAP4 program can edit this file, in accordance with Reference 1, to include conditions not considered in the model generation phase.

"Static Analysis" (enter 3) will invoke the finite element analysis program (SAP4). The model generation and file formatting steps must be complete before this step. The user does not prepare data for, or interact with this program. The user may elect to pursue the SAP4 printed output by using a screen editor or list program capable of scrolling and panning. The SAP4 output file, SAP4.OUT, is a lengthy file consisting of 132 character line lengths. This file can be printed in compressed mode (17 characters per inch) or normal mode (10 characters per inch), depending on the paper width (8.5 x 11 or 14 x 11, respectively).

"Results Processing" (enter 4) will graphically present the finite element analysis results. These results include the model deflected shape, the stress distribution based upon maximum and minimum stress quantities, as well as permissible threshold stress quantities and stress gradients for selected locations on the cross section.

"Exit and Return the system to standard configuration" (enter 5) will return the computer configuration to that state before the DAS command was executed.

"Quit" (enter 6) will exit DAS without returning the system to the original configuration. This will permit the user to perform file edits and general housekeeping chores without having to change the DAS environment.

Model Generation

The model generation feature includes generating the cross section geometry and the associated finite element mesh, generating load functions and assigning appropriate concentrated loads to the respective nodes, and defining the boundary conditions by placing springs at appropriate nodes. Selecting "Model Generation" from the DAS menu will produce the following menu:

DOCKPRE

1. MESH	2. FORCE	3. SPRING
4. SUMMARY	5. PLOTTER	6. EXIT

"MESH" (enter 1) will generate the two-dimensional finite element mesh.

"FORCE" (enter 2) will generate the load functions using a predefined file containing the load function values.

"SPRING" (enter 3) will generate the boundary conditions at specified nodes along the outer edge of the mesh. The spring stiffness is computed based upon the bearing capacity of the surrounding soil.

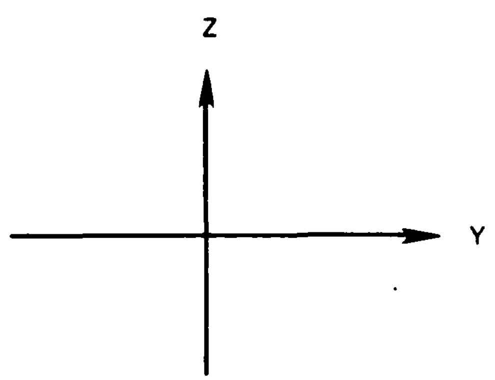
"SUMMARY" (enter 4) will produce a summary display of the mesh, loading functions, and springs.

"PLOTTER" (enter 5) will reproduce the summary display on a plotter.

"EXIT" (enter 6) will return control to the main DAS menu.

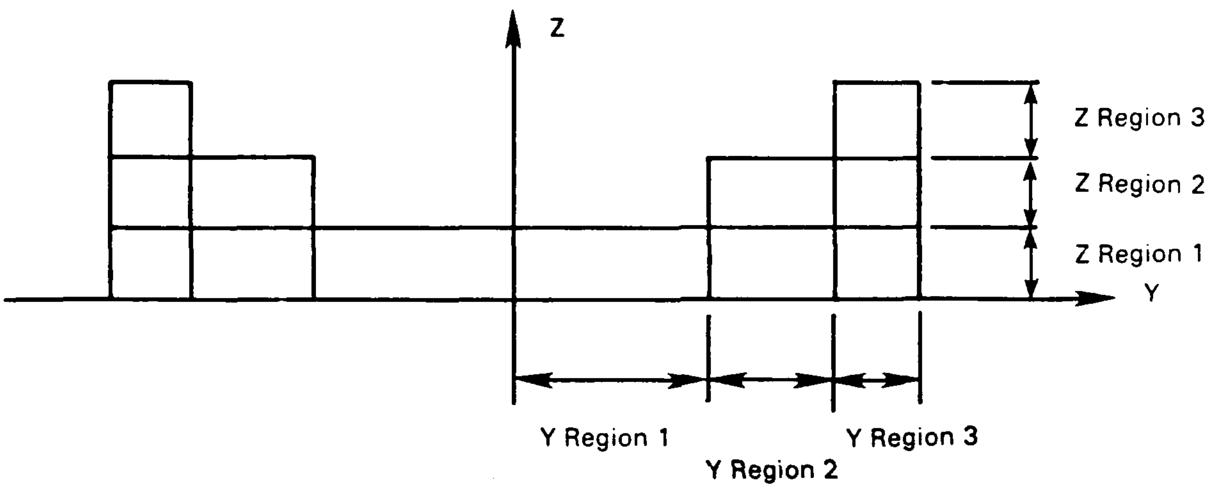
MESH. The MESH menu option will now be explained. An understanding of the following conventions is required before one can correctly respond to the model generation prompts:

The geometric model cross-section is entered in the Y-Z plane;

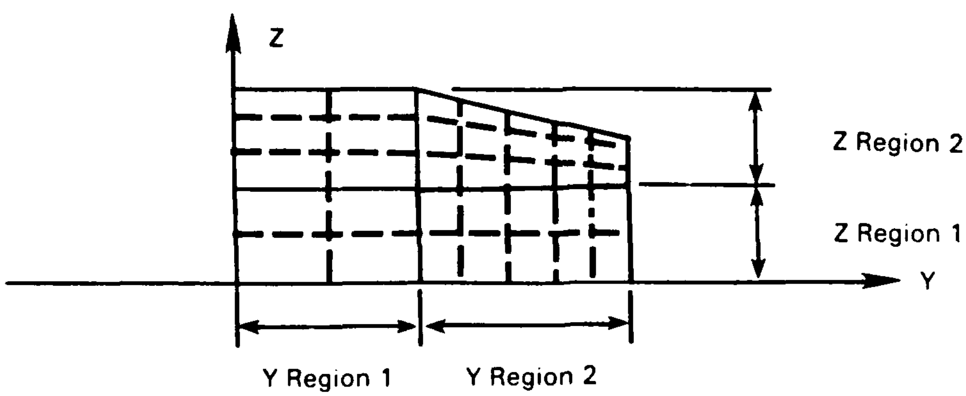


DAS uses a view port or window defined by the minimum and maximum values in the Y and Z directions to establish the overall window boundaries. The values entered should be slightly larger than the extreme values of the drydock cross section width and height, therefore allowing a clear space around the model cross section. These distances should be entered in length units of feet.

The model cross section is divided into a subset of quadrilaterals, henceforth called REGIONS, each of which is subdivided into a set of quadrilateral ELEMENTS. The right half of the symmetric model cross section is input by the user with the left half generated by DAS.



The user defines the number of Y REGIONS to the right of the Z-axis (positive Y direction), then for each Y REGION he defines the number of Z REGIONS. REGION boundaries are defined by corner coordinates beginning with the Z REGION corners for Y = 0 boundary line, progressing from bottom to top in sequence (positive Z-direction). Subsequent prompts will request the Y,Z coordinates of the right hand side of each Z REGION of the current Y REGION, completing the current Y REGION, then continuing to the next Y REGION until the model is complete. Thus, REGIONS 1,1; 1,2; 2,1; 2,2 might look like:



The finite element mesh is constructed by subdividing the REGIONs into sets of ELEMENTS. Adjacent regions may have a different number of ELEMENTS. The number of Y ELEMENTS in each Y REGION and the number of Z ELEMENTS in the corresponding Z REGION are defined. Provision is made for regions without elements and areas within a region without elements.

During the model generation step, DAS records in a log file all of the responses a user makes to the conversational prompts. The log file is named with the "model name" on the DAS command. If the DAS command is entered with a <model name> that does not exist from a previous run, the program assumes a new model will be entered. The user may exit the model generation at anytime during the session by pressing CONTROL-C (^C). To restart the model generation from the point of exit simply issue the DAS command with the "model name" of that previous session. Of course, it may be desirable to correct or append the "model name" file using the screen editor, following the format in Appendix A, before proceeding. Reselecting "MESH" after the DAS "Model Generation" option has been chosen will cause the "model name" file to control the recreation of the model until the end of the file is encountered, the control will then revert to the conversational mode.

The model generation prompts for creating the finite element mesh are:

PROMPT> INPUT PLOT WINDOW BOUNDARY YMIN, YMAX (FT):

Enter the view port minimum and maximum Y coordinates (FT) separated with a comma. The origin (0,0) is located at the lower center of the drydock cross section. Adding 20 feet to the drydock width (10 feet on the right and left sides) will produce reasonable window boundaries.

PROMPT> INPUT PLOT WINDOW BOUNDARY ZMIN, ZMAX (FT):

Enter the view port minimum and maximum Z coordinates (FT) separated with a comma. The origin (0,0) is located at the lower center of the drydock cross section. Adding 20 feet to the drydock height (10 feet on the top and -10 to the bottom) will produce reasonable window boundaries.

PROMPT> INPUT THE DRYDOCK MODEL TITLE:

Enter the drydock model title with maximum of 20 characters. This title will appear on the drydock plots.

Regions.

PROMPT> INPUT THE NUMBER OF Y-REGIONS TO THE RIGHT OF Y = 0:

Enter the total number of y-regions to the right of the center line of the drydock (Y = 0). Each region will later be subdivided into elements.

PROMPT> INPUT THE NUMBER OF Z-REGIONS IN Y-REGION n:

Enter the total number of z-regions for the current y-region 'n', where n is a counter to identify the current y-region.

PROMPT> INPUT n VALUES OF Z (FT) FOR THE KEY POINTS ON Y = 0:

This is the left boundary of the first y-region (e.g, Y = 0), thus, in the coordinate pair (Y,Z) Y is always 0. The Z values (FT) are recorded for each Z coordinate on the Y = 0 line. There will be the number of z-regions plus one value, and n identifies the current value.

PROMPT> INPUT (Y,Z) VALUES (FT) FOR THE RIGHT BOUNDARY OF Y-REGION n PAIR m:

This defines the right boundary for each of the y-regions identified by the counter "n". The coordinate pair (Y,Z), identified by the counter "m", defines the corners of the current z-region in the "n"th y-region. Regions on both sides of the Y boundary must be compatible. In this manner all the vertices of the regions will be defined. Note that the number of coordinate pairs are one more than the number of corresponding z-regions.

Entirely Vacant Regions.

PROMPT> HOW MANY Z-REGIONS OF Y-REGION n ARE ENTIRELY VACANT?

Enter the number of z-regions in y-region "n" that do not have finite elements.

PROMPT> REGIONS ARE NUMBERED 1,2,3... FROM THE BOTTOM. LIST THE Z REGION NUMBERS HAVING VACANT ELEMENTS:

For each vacant z-region, enter the z-region identification number. The prompt will appear again until the specified number of vacant z-regions have been identified. The z-regions are numerically identified starting from the bottom, which is the y-axis. This feature will cause the entire selected region to be void of finite elements.

Elements.

PROMPT> INPUT THE NUMBER OF Y-ELEMENTS IN Y-REGION n:

For the "n"th y-region, define the number of elements along the y direction region boundary (y-elements). Compatibility with adjacent regions must be assured by the user.

PROMPT> INPUT THE NUMBER OF Z-ELEMENTS IN Z-REGION n:

For the "n"th z-region, define the number of elements along the z direction region boundary (z-elements). Compatibility with adjacent regions must be assured by the user.

Vacant Elements.

PROMPT> ARE THERE VACANT ELEMENTS IN Y-REGION n ? (Y/N)

This is a vacant element option, as opposed to the previous vacant region option, permitting the selective omission of one or more elements from the "n"th y-region that otherwise has elements. Entering y for yes will cause the following prompts to be displayed:

PROMPT> HOW MANY Z-REGIONS HAVE VACANT ELEMENTS?

Enter the number of z-regions in the "n" y-region having vacant elements.

PROMPT> REGIONS ARE NUMBERED 1,2,3... FROM THE BOTTOM.
A NEGATIVE REGION NUMBER MAKES THE ENTIRE REGION VACANT.
LIST Z-REGION NUMBERS HAVING VACANT ELEMENTS:

For each z-region having vacant elements, enter the z-region identification number. The prompt will appear again until the specified number of vacant z-regions have been identified. The z-regions are numerically identified starting from the bottom, which is the y-axis. This feature will cause the entire selected region to be void of finite elements if a negative region identification is used.

PROMPT> ARE ANY ELEMENTS OF Y-REGION n, Z-REGION m, AND
ELEMENT COLUMN k VACANT? (Y/N)

For the region identified by the y-region "n" and the z-region "m" counters, decide if the "k"th element column has any vacant elements. The "k" counter precedes from left to right. A "yes" response will create the following prompt:

PROMPT> HOW MANY VACANT ELEMENTS?
IF ALL ELEMENTS ARE VACANT ENTER A 0:

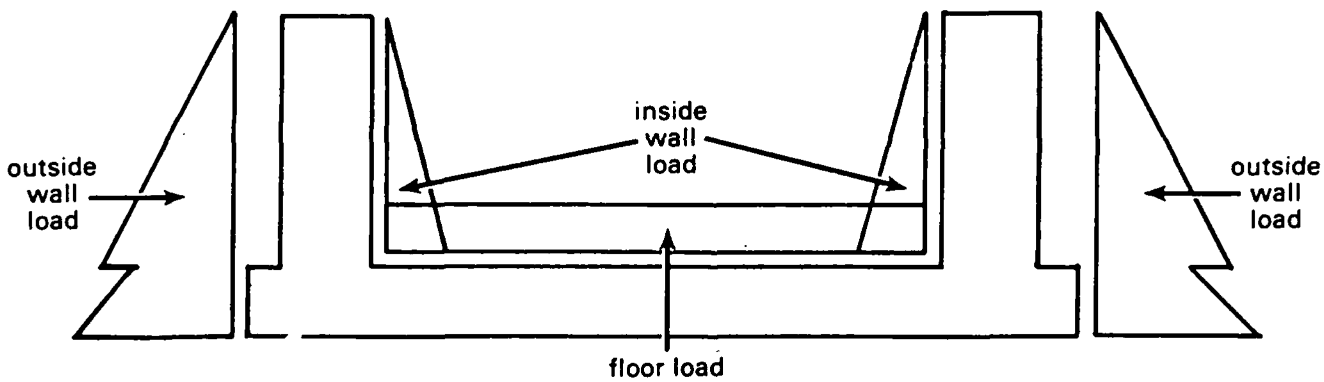
Enter the number of vacant elements. A zero entry will vacate all elements of the column.

PROMPT> ELEMENTS ARE NUMBERED 1,2,3... FROM THE BOTTOM OF
THE CURRENT Z-REGION.
LIST THE VACANT ELEMENTS:

Enter the vacant element identification number, defined with respect to the bottom of the current z-region. The prompt will repeat until the specified number of vacant elements in this column have been identified.

FORCE. The FORCE menu option is discussed in the following section. Before selecting this option the user must prepare a file containing the load function information. The format of this file is discussed in Appendix B. A future edition of DAS will have a program that will prepare the necessary load function data.

DAS allows the user to define numerous load cases for a specific geometric model through the use of a load input data file. The user constructs various load input files for each load case to be analyzed before entering the DAS system. The load input filename is entered on the command line following the corresponding model name when entering DAS. The load input file consists of three basic loadings: (1) outside wall distributed loading, (2) inside wall and/or inside floor distributed loading, and (3) inside floor block concentrated loading. All length distances are entered in feet, while force units are pounds per square foot (psf). Examples of typical DAS load input files are given in Appendix B along with a sketch of the associated load case.



The load function data is converted to equivalent concentrated nodal loads applied to the node locations on the model cross section corresponding to the locations specified in the load function data file. The equivalent concentrated load is computed from the load function ordinate with respect to the location of the node and the tributary area associated with the node where the concentrated load will be applied. The following are DAS prompts that are used to associate the load function with the drydock cross section:

PROMPT>	ENTER	1 OUTSIDE NODE 2 INSIDE NODE 3 DONE:
Enter:	1	To select the top and bottom (order dependent) nodes corresponding to the outside wall load case. The right wall locations will also be used for the left wall. The load functions for the right and left side may be different.

2 To select the top and bottom nodes corresponding to the inside wall load case along with the nodes defining the floor hydrostatic and blocking loads.

3 To exit and return to the model generation menu.

PROMPT> ON THE RIGHT HAND SIDE, USE THE CURSOR TO LOCATE INITIAL POINT OF THE LOAD APPLICATION REFERENCE LINE

PROMPT> ON THE RIGHT HAND SIDE, USE THE CURSOR TO LOCATE THE ENDING POINT OF THE LOAD APPLICATION REFERENCE LINE

The previous two prompts define the line of nodes to which the load will be applied. Only the right hand side of the model is addressed, the left side being symmetrical. However, the load function need not be symmetrical.

PROMPT> ENTER 1 DISPLAY FORCE VALUE
2 RETURN TO MAIN MENU:

Enter: 1 To display the approximate force value of the selected point on the load function curve.

2 To return to model generation menu.

If "DISPLAY FORCE VALUE" is selected the following will appear:

PROMPT> USE THE CURSOR TO LOCATE THE DESIRED LOAD LOCATION.

SPRING. The SPRING menu option is used to define the boundary conditions for the drydock cross section mode. Springs are used to represent the stiffness of the supporting soil. The spring constant or stiffness is computed from the bearing capacity of the soil with respect to the tributary area associated with the node to which the spring is attached. The user may select single springs, a spring applied to a single specified node, or a series of springs applied to a line of contiguous nodes.

PROMPT> INPUT SOIL BEARING CAPACITY (PSI):

Enter the soil bearing capacity in psi units.

PROMPT> COUNTER CLOCKWISE SEQUENCE FROM LEFT TO RIGHT
SPRING LOCATION MENU:

SINGLE SPRING LOCATED ON; 1) LEFT SIDE 2) BASE 3) RIGHT SIDE
LINE OF SPRINGS LOCATED ON; 4) LEFT SIDE 5) BASE 6) RIGHT SIDE
7) EXIT:

For single spring locations the following will appear:

PROMPT> USE THE CURSOR TO LOCATE A SINGLE SPRING LOCATION POINT.

Use the cursor to select a node, then hit <CR> to create and display the spring.

For a series of springs located on a line of nodes the following will appear:

PROMPT> USE THE CURSOR TO LOCATE THE END POINTS OF THE SPRING LOCATION REFERENCE LINE. THE END POINT LOCATION MUST BE DEFINED IN A COUNTER CLOCKWISE SEQUENCE FROM THE TOP LEFT.

Use the cursor to select the first node defining the beginning of the reference line, then hit <CR> to enter the node, then select the second node defining the end of the reference line in the same manner.

Enter option 7 to exit or return to the model generation menu.

SUMMARY. The SUMMARY menu option will display the drydock model cross section with its associated loading and boundary conditions.

PLOTTER. The PLOTTER menu option will also provide an option to plot the summary display so the information can be included in a report. The plotter is assumed to be compatible with the Hewlett Packard Graphics Language (HPGL).

PROMPT> ENTER 1 FOR MESH PLOT
2 FOR FORCE AND SPRING PLOT:

Enter:	1	For the drydock cross section finite element mesh to be plotted.
	2	For the drydock cross section finite element mesh, including boundary conditions and loading conditions.

Format Input Data File for Static Analysis

The model data base will be converted to a single file having the format required by the analysis program. The material properties that will be used in the analysis have been incorporated into the steps. However, the user may change these values by responding to the following dialogue:

THE CONCRETE MATERIAL PROPERTIES ARE:

1	Weight density	=	145.000 LB/FT**3
2	Poissons ratio	=	0.240
3	Compressive strength of concrete	=	3500.000 PSI

DO YOU DESIRE TO CHANGE THESE PROPERTIES? Y/N:
WHICH ONE? 1, 2, OR 3:
ENTER NEW VALUE:

The modified material properties and the change interrogative are displayed for validation by the user before the process proceeds.

Results Processing

The results processing feature includes displacement and stress displays. The displacement display includes the deformed mesh superimposed on the undeformed reference mesh. The various stress quantities can be displayed in a variety of ways. A stress distribution display shows the selected stress for each element with respect to a spectrum of eight colors. The maximum and minimum stress extremes can be shown in red and blue respectively, or a maximum and minimum stress threshold can be defined. The stress distribution display is used to select colinear elements of interest; so the stress gradient, the change in stress with respect to distance, can be displayed. Selecting "Results Processing" from the DAS menu will produce the following menu:

DOCKPOST

1 = DISPLACEMENT   2 = STRESS   3 = FAILURE   4 = EXIT

"DISPLACEMENT" (enter 1) will display the deformed geometry.

"STRESS" (enter 2) will display the stress distribution with respect to the maximum and minimum selected stress values. A stress gradient may also be prepared.

"FAILURE" (enter 3) will display the stress distribution with respect to stress thresholds defined from allowable stress values for concrete and steel. A stress gradient may also be prepared.

"EXIT" will return control to the main DAS menu.

DISPLACEMENT. The DISPLACEMENT menu option will now be explained.

PROMPT> ENTER THE DISPLACEMENT MAGNIFICATION FACTOR:

Enter the displacement magnification factor to enhance the displacement values so the deformed structure will appear displaced in the display.

PROMPT> ENTER
1 NEW MAGNIFICATION FACTOR
2 DISPLAY DISPLACEMENT VALUES
3 PLOT
4 RETURN TO DOCKPOST MENU:

Enter:

1 To change the displacement magnification factor.

2 To select and display node coordinate and displacement values.

3 To plot the displacement plots on the HP-plotter.

4 To return to the DOCKPOST menu.

PROMPT> USE THE CURSOR TO LOCATE A NODE ON THE UNDEFORMED VIEW TO DISPLAY ITS COORDINATES (FT) WITH DISPLACEMENTS (IN).

Use the cursor to select a node on the undeformed mesh, then hit <CR>. The program will display the (y,z) coordinates of the undeformed mesh and the y and z displacements associated with the node.

PROMPT> MOVE THE CURSOR TO SELECT ANOTHER NODE OR ENTER <CR> TO EXIT.

Use the cursor to select the node of interest on the undeformed mesh. A <CR> without cursor movement will redisplay the displacement menu.

STRESS. The STRESS menu option will produce stress distribution and gradient displays of selected stress quantities. This option is discussed below.

The user must select the stress quantity of interest. The stress type is selected from the following menu. Figure 1 shows the normal stresses, Y STRESS (S11) and Z STRESS (S22), and the SHEAR STRESS (S12). MAXIMUM and MINIMUM PRINCIPAL STRESSES are in the plane of the element and the X STRESS is the third principal stress acting on the plane of the element. The ANGLE is the angle in degrees from (1) the local y axis at point O, or (2) the n axis at the midpoints, to the axis of the algebraically largest principal stress.

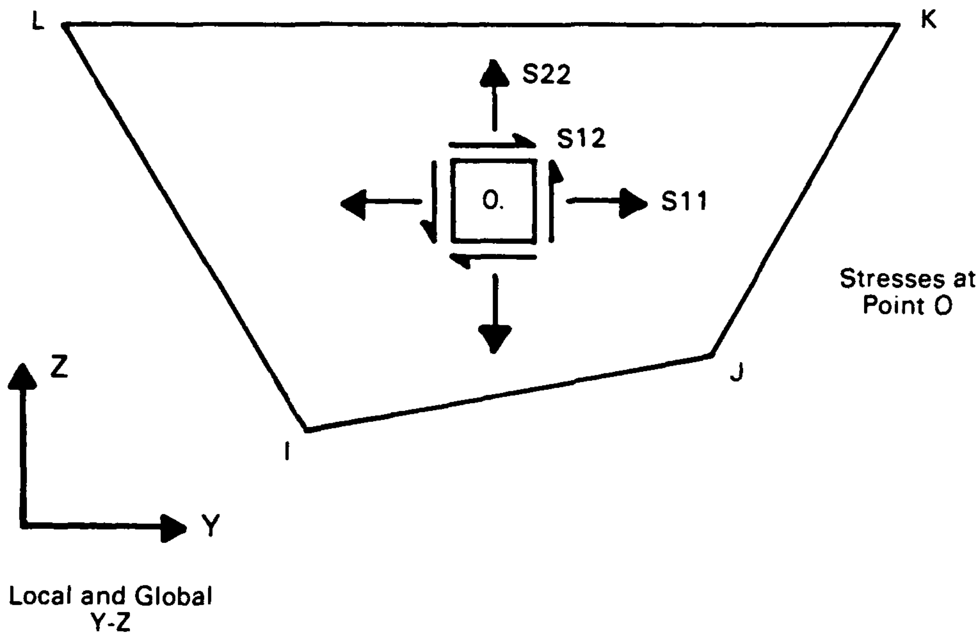


Figure 1. Stress block in local and global coordinates.

PROMPT> ENTER STRESS TYPE;

1	Y STRESS	2	Z STRESS
3	X STRESS	4	SHEAR STRESS
5	MAX PRINCIPLE STRESS	6	MIN PRINCIPLE STRESS
7	ANGLE TO PRINCIPLE AXIS	8	RETURN TO DOCKPOST MENU:

Enter 1 to 7 to select a stress orientation for display.

PROMPT> ENTER

1 RETURN TO STRESS MENU

2 STRESS GRADIENT PLOT:

Enter:	1	To redisplay the stress menu.
	2	To create a Stress Gradient Plot.

PROMPT> USE THE CURSOR TO LOCATE THE ELEMENT CENTER POINTS OF THE STRESS REFERENCE LINE.

Use the cursor to select centers of the elements at the left and right ends of the reference line along which the stress gradient is desired. The reference line is a contiguous set of adjacent elements. Additional element center locations may be required to adequately describe the element line of interest.

PROMPT> ENTER

1 DISPLAY THE GRADIENT COORDINATES

2 PLOT THE STRESS GRADIENT

3 RETURN TO STRESS MENU:

Enter:	1	To read the stress gradient coordinates from the display.
	2	To plot the stress gradient display on the HP plotter.
	3	To return to the stress menu.

PROMPT> READ STRESS AND DISTANCE VALUES WITH CURSOR ENTER <CR> TO EXIT.

Use the cursor to select the point of interest on the stress gradient curve, then hit <CR>. The distance and stress values (x,y coordinates) will be displayed.

FAILURE. The FAILURE menu option will now be discussed. The failure criterion employed is based upon the maximum normal stress theory. Tensile failure (shaded red) is assumed to occur when the greatest positive principal stress is larger than the tensile strength for the chosen mode of failure. For the concrete mode of failure, the tensile strength f_{ct} (psi), is defined by ACI-11.2:

$$f_{ct} = 6.7\sqrt{f'_c}$$

where f'_c = compressive strength of concrete (psi)

The steel mode of failure is based upon the yield stress, f_y(psi), of the selected reinforcing steel.

Compression failure in concrete (shaded blue) occurs when the greatest negative principal stress is less than the compressive strength of concrete, f'_c.

PROMPT>	INPUT THE CONCRETE COMPRESSIVE STRENGTH (PSI):	
	Enter the concrete compressive strength.	
PROMPT>	INPUT THE STEEL YIELD STRESS (PSI):	
	Enter the reinforcing steel yield stress.	
PROMPT>	CHOOSE STEEL OR CONCRETE FAILURE MODE (S OR C):	
	Enter the character S for steel or C for concrete failure modes defined above.	
PROMPT>	ENTER	1 RETURN TO DOCKPOST MENU
		2 STRESS GRADIENT PLOT:
Enter	1	To redisplay the DOCKPOST menu.
	2	To create a Stress Gradient Plot.
PROMPT>	USE THE CURSOR TO LOCATE THE ELEMENT CENTER POINTS OF THE STRESS REFERENCE LINE.	
	Use the cursor to select centers of the elements at the left and right ends of the reference line along which the stress gradient is desired. The reference line is a contiguous set of adjacent elements. Additional element center locations may be required to adequately describe the element line of interest.	
PROMPT>	ENTER	1 DISPLAY THE GRADIENT COORDINATES
		2 PLOT THE STRESS GRADIENT
		3 RETURN TO DOCKPOST MENU:
Enter:	1	To read the stress gradient coordinates from the display.
	2	To plot the stress gradient display on the HP plotter.
	3	To return to the stress menu.
PROMPT>	READ STRESS AND DISTANCE VALUES WITH CURSOR ENTER <CR> TO EXIT.	
	Use the cursor to select the point of interest on the stress gradient curve, then hit <CR>. The distance and stress values (x,y coordinates) will be displayed. Hitting <CR> without moving the cursor will cause an exit from this section.	

DAS INSTALLATION

To load and run DAS you will need the following microcomputer components:

- An IBM-PC/AT (or compatible) Microcomputer
- 640 KB RAM
- MS-DOS Version 3.xx or later
- An EGA or better graphics system
- A 360KB DSDD diskette drive (for installation)
- 10 MB of free hard disk storage
- Math Coprocessor

DAS is supplied on nine 360KB DSDD diskettes. The first diskette, DISK1, includes a batch file (START.BAT) that will create the following directory structure and load DAS into them:

ROOT:\DRYDOCK

ROOT:\DRYDOCK\DOCKPRE

ROOT:\DRYDOCK\SAPDECK

ROOT:\DRYDOCK\SAP4

ROOT:\DRYDOCK\DOCKPOST

To install DAS, insert DISK1 in a drive with 360KB DSDD read capability and type:

COPY A:\START.BAT <CR>

START C: <CR> (for a C hard disk)

or

START D: <CR> (for a D hard disk)

Follow the displayed instructions, responding to the questions as appropriate to your microcomputer's environment. When finished, remove START.BAT by typing:

ERASE START.BAT <CR>

During installation the START.BAT procedure will query the user for the existence of a compatible Hewlett Packard pen plotter:

HP PLOTTER? (Y/N)

REFERENCE

1. Naval Civil Engineering Laboratory User Guide, UG- _____: SAP4 User Guide, Version 1.0, Structural Analysis Program for Static and Dynamic Response of Linear Systems," by Frank R. Johnson, Port Hueneme, CA (in preparation).

ACKNOWLEDGMENTS

Mr. Bao Kim Nguyen did the programming for the DAS software. His skill and effort is greatly appreciated.

The graphics algorithms utilize the American National Standard Institute (ANSI) Graphical Kernel System (GKS), ANSI X3.124-1985 level 2b, as implemented by Graphics Software Systems' GSS-TOOLKIT KERNEL SYSTEM.

The embedded finite element analysis program was obtained from the Earthquake Engineering Research Center, College of Engineering, University of California, Berkeley, California. The program is described in Report No. EERC 73-11, "SAP IV, A Structural Analysis Program For Static And Dynamic Response of Linear Systems," by K. Bathe, E.L. Wilson, and F.E. Peterson, April 1974. The program was converted to the IBM PC by Mr. J. Holland, assisted by conversion software provided by Haested Methods Inc.

Appendix A

LOG FILE DATA PREPARATION

```
LINE |
TYPE |                                   COLUMN
     |          1         2         3         4         5         6
     | 1234567890123456789012345678901234567890123456789012345678901234567890
-----+-----------------------------------------------------------------------
A    |    -50.000              50.000             -10.000              50.000
B    | DRYDOCK TITLE
C    |     3
D    |     1    2    3
E    |   0.00  10.00
F-1  |  10.00   0.00  10.00  10.00  10.00  20.00
F-2  |  20.00   0.00  20.00  10.00  20.00  20.00  20.00  30.00
F-3  |  40.00   0.00  40.00  10.00  40.00  20.00  40.00  30.00
G-1  |     0
G-2  |     1   -2
G-3  |     0
H    |     3    3    3
I    |     3    3    3
E    |   0.00  10.00
     |
     |
     |    ---- DEPENDENT UPON THE RESPONSE OF THE QUESTION ----
     |
     |
F-1  |  10.00   0.00  10.00  10.00  10.00  20.00
G    | N
F-2  |  20.00   0.00  20.00  10.00  20.00  20.00  20.00  30.00
K    | Y
L    |     1    1
M    | N
N    | Y
O    |     1    3
M    | N
F-3  |  40.00   0.00  40.00  10.00  40.00  20.00  40.00  30.00
G    | N
```

The log file is prepared automatically by DAS during the model generation phase. It is classified to preclude having to reanswer questions that have previously been answered correctly. It also serves as a means to develop a library of drydock models. If one desires, this file can be prepared or edited using a screen editor. This file was created using the following format information:

DESCRIPTION OF INPUT LINES

Line Type A **Format: 4(F10.3,10X)**

Column	Description
1 - 10	YMAX of window (ft)
21 - 30	YMAX of window (ft)
41 - 50	ZMIN of Window (ft)
61 - 70	ZMAX of Window (ft)

Line Type B **Format: A20**

Column	Description
1 - 20	Identifiable Title

Line Type C **Format: I5**

Column	Description
1 - 5	No. of Y-region to the right of Y = 0

Line Type D **Format: 20(1X,I4)**

Column	Description
2 - 5	No. of Z-regions in Y-region = 1
6 - 10	No. of Z-regions in Y-region = 2
12 - 15	....
17 - 20	....
....	....
97 - 100	No. of Z-region in Y-region = total no. of Y-region to the right of Y = 0

Line Type E **Format: 20(F6.2,1X)**

Column	Description
1 - 6	Z-coordinates along the Y-line where Y = 0
8 - 13	....
15 - 20	....
....	....
135 - 140	....

Line Type F **Format: 20(F6.2,1X)**

Column	Description
1 - 6	Z-coordinates along the Y-line where Y not equal to zero (Y/=0) and Y-region = 1 to the total of Y-region to the right of Y = 0
8 - 13	....
15 - 20	....
....	....
135 - 140	....

Line Type G **Format: 21(1X,I4)**

Column	Description
2 - 5	No. of Z-region in Y-region = 1 are entirely vacant
6 - 10	No. of Z-region in Y-region = 2 are entirely vacant
12 - 15	....
17 - 20	....
....	....
97 - 100	No. of Z-region in Y-region = total no. of Y-region to the right of Y = 0, are entirely vacant

Line Type H **Format: 20(1X,I4)**

Column	Description
2 - 5	No. of Y-element in Y-region = 1
6 - 10	No. of Y-element in Y-region = 2
12 - 15	....
17 - 20	....
....	....
97 - 100	No. of Y-element in Y-region = total no. of Y-region to the right of Y = 0

Line Type I **Format: 20(1X,I4)**

Column	Description
2 - 5	No. of Z-element in Z-region = 1
6 - 10	No. of Z-element in Z-region = 2
12 - 15	....
17 - 20	....
....	....
97 - 100	No. of Z-element in Z-region = total no. of Z-region

Line Type G **Format: A5**

Column	Description
1 - 5	There are no vacant elements in Z-region n and Y-region m.

Line Type K Format: A5

Column	Description
1 - 5	There are vacant elements in Z-region n and Y-region m.

Line Type L Format: 20(1X,I4)

Column	Description
2 - 5	No. of Z-region that was vacant element
6 - 10	Z-region n has vacant element
12 - 15	....
17 - 20	....
....	....
97 - 100	....

Line Type M Format: A5

Column	Description
1 - 5	There are no vacant elements in element k of Z-region n and Y-region m

Line Type N Format: A5

Column	Description
1 - 5	There are vacant elements in element k of Z-region n and Y-region m

Line Type O Format: 20(1X,I4)

Column	Description
2 - 5	No. of vacant element
6 - 10	element k is vacant
12 - 15	....
17 - 20	....
....	....
97 - 100	....

Appendix B

LOAD CASE DATA PREPARATION

This appendix describes how to prepare the load data file for DAS. The active and passive soil pressure, hydrostatic pressure, and ship blocking loads are considered. Future extensions will add equavilent static earthquake loads. A program to compute these loads is currently under development.

DESCRIPTION OF INPUT LINES

Line Type A

Column	Description
1 - 80	Load case header for passive or active pressure

Line Type B

Column	Description
1 - 10	Depth (FT) on the left side from top of drydock to the location of the pressure function value.
11 - 20	Pressure function value (PSF) for the left side of the drydock.
21 - 30	Depth (FT) on the right side, from the top of the drydock to the location of the pressure function value.
31 - 40	Pressure function value (PSF) for the right side of the drydock.

Line Type C

Column	Description
1 - 80	Load case header for hydrostatic pressure on the inside of the drydock.

Line Type D

Column	Description
1 - 10	Depth (FT) of the floor from the top of the drydock.
11 - 20	Hydrostatic pressure (PSF) at the floor level.

Line Type E

Column	Description
1 - 80	Load case header for the ship support block load

Line Type F

Column	Description
1 - 10	Distance (FT) from the drydock center line to the center line of the ship. A positive quantity means the ship is to the right of the drydock center line.

Line Type G

Column	Description
1 - 10	Distance (FT) to first ship support block from the ship center line.
11 - 20	Ship support block load (LB/FT)

Line Type H

Column	Description
1 - 3	"END" to terminate the data file

TYPICAL DAS LOAD INPUT DATA FILE

```
LINE  |                     Column
TYPE  |          1         2         3         4         5         6
      |123456789012345678901234567890123456789012345678901234567890
-------------------------------------------------------------------
A     |LOAD CASE 1        OUTSIDE
B-1   |       0.0       0.0       0.0       0.0
B-2   |      -1.0      -1.0       6.0     360.0
B-3   |      36.0    3120.0      36.0    3120.0
B-4   |      36.0     372.0      36.0    3720.0
B-5   |      52.0    5020.0      52.0    5020.0
C     |                   INSIDE
D-1   |       0.0       0.0    2752.0
D-2   |      43.0    2750.0
E     |                   BASE
F     |       6.0
G-1   |       6.0   46100.0
G-2   |       6.0   30900.0
G-3   |      10.0   40900.0
G-4   |      14.0   30900.0
H     |END
```

Appendix C

INPUT AND OUTPUT FOR EACH DAS BATCH UTILITY OPTION

1. Model Generation

Input: Interactively
Previous session file

Output: CORNDT.DAT
ELMNDT.DAT
FOR2NOD.DAT
SPG2NOD

-- Graphic data files --
MESH.PLT
FORCE.PLT
SPRING.PLT

2. Format input data file for Static Analysis

Input: CORNDT.DAT
ELMNDT.DAT
FOR2NOD.DAT
SPG2NOD

Output: SAP4.IN

3. Static Analysis

Input: SAP4.IN

Output: SAP4.OUT
TAPE2. -- displacement
TAPE10. -- stress
TAPE8. -- geometry and element information

4. Result Processing

Input: TAPE2. -- displacement
TAPE10. -- stress
TAPE8. -- geometry and element information

Output: none

Appendix D

CONTENTS OF THE DISTRIBUTION DISKS

The DAS Distribution Package includes:

DISK1 Batch File and System Kernel:

DRYDOCK	
START	BAT
WARMBOOT	COM
KINGS	DAT
LOAD1	DAT
LOAD2	DAT
LOAD3	DAT
TEST1LD2	DAT
CHARDD5	DAT
TEST1LD1	DAT
TEST1LD3	DAT
TEST1X-S	DAT
ASK	EXE
FONTS	H
KERNEL1	SYS
KERNEL2	SYS
META	SYS
GSSCGI	SYS
HPPLOT	SYS
IBMEGA	SYS
ANSI	SYS
FONT106	TBL
FONT101	TBL
FONT102	TBL
FONT103	TBL
FONT104	TBL
FONT105	TBL

Note: The contents of the KERNEL.SYS file 1 and 2 are:

KERNEL1.SYS for the hardware without the HP-plotter connection as:

1: DISPLAY
3: WISS

KERNEL2.SYS for the hardware with the HP-plotter connection as:

1: DISPLAY
3. WISS
4: PLOTTER

Directory of subdirectory \DRYDOCK

C-DRIVE
D-DRIVE

Directory of subdirectory \DRYDOCK\C-DRIVE

DAS.BAT
AUTOEXEC.GKS
AUTOEXEC.SAP
CONFIG.GKS
CONFIG.SAP

Directory of subdirectory \DRYDOCK\D-DRIVE

DAS.BAT
CONFIG.SAP
CONFIG.GKS
AUTOEXEC.GKS
AUTOEXEC.SAP

<u>DISK2</u> MESHGEN DOCKGEN1

DOCKGEN.EXE

<u>DISK3</u> MESHGEN DOCKGEN2

DOCKGEN.OVL

<u>DISK4</u> SAPDECK

SAPDECK.OQL
SAPDECK.EXE
ZSQ.EXE
USQ.EXE

<u>DISK5</u> SAP4.1

SAP4.EXE

<u>DISK6</u> SAP4.2

SAP4COM.OVL

<u>DISK7</u> SAP4.3

SAP4DYN.OVL

<u>DISK8</u> POST PROCESSOR - POST1

DOCKPOST.EXE

<u>DISK9</u> POST PROCESSOR - POST2

DOCKPOST.OVL

DAS

FEEDBACK REPORT

The Naval Civil Engineering Laboratory is fully dedicated to supporting GEMS users. A primary requirement for this task is to establish a priority listing of user requirements. It would be of great value to the development of new software if you, the user, would complete the feedback questions below. Since each individual user may have specific requirements, please reproduce this page as many times as necessary.

Please circle the number that best applies in questions 1 through 4, complete the other questions, fold at tic marks, and mail to NCEL with franked label on reverse side or to address at bottom of page.

1. Was the software beneficial (productive)?

No benefit 0 1 2 3 4 5 6 7 8 9 10 Very beneficial

2. Was it easy to use (user friendly)?

Difficult 0 1 2 3 4 5 6 7 8 9 10 Very easy

3. Does this software make decisions more reliable?

No 0 1 2 3 4 5 6 7 8 9 10 Yes

4. Does it better document the design?

No 0 1 2 3 4 5 6 7 8 9 10 Yes

5. Did it save time?

Yes_____ No_____ Estimated percent saved_____

6. What would make future software more user friendly?

7. What further support would you like to have on the GEMS system?

8. What other comments or remarks would you like to add?

Activity____

Telephone____

Mail address is:

NAVFAC GEMS Support Group
Naval Civil Engineering Laboratory
Code L54
Port Hueneme, CA 93043-5003