NUREG/CR-3686
UCRL-15597 Vol. 1

WIPS—Computer Code for Whip and Impact Analysis of Piping Systems

Part A—User's Manual

G. H. Powell, J. P. Hollings, D. G. Row, P. Chen, F-C. Hu, M. Mahasuverachai, B. Mosaddad, P. Nicklin, S. Nour-Omid, C. Oughourlian, and A. Riahi;
University of California, Berkeley, CA

Prepared for
U.S. Nuclear Regulatory Commission

Lawrence Livermore National Laboratory

NOTICE

Availability of Reference Materials Cited in NRC Publications

Most documents cited in NRC publications will be available from one of the following sources:

1. The NRC Public Document Room, 1717 H Street, N.W.
   Washington, DC 20555
2. The NRC/GPO Sales Program, U.S. Nuclear Regulatory Commission,
   Washington, DC 20555
3. The National Technical Information Service, Springfield, VA 22161

Although the listing that follows represents the majority of documents cited in NRC publications, it is not intended to be exhaustive.

Referenced documents available for inspection and copying for a fee from the NRC Public Document Room include NRC correspondence and internal NRC memoranda; NRC Office of Inspection and Enforcement bulletins, circulars, information notices, inspection and investigation notices; Licensee Event Reports; vendor reports and correspondence; Commission papers; and applicant and licensee documents and correspondence.

The following documents in the NUREG series are available for purchase from the NRC/GPO Sales Program: formal NRC staff and contractor reports, NRC-sponsored conference proceedings, and NRC booklets and brochures. Also available are Regulatory Guides, NRC regulations in the *Code of Federal Regulations*, and *Nuclear Regulatory Commission Issuances*.

Documents available from the National Technical Information Service include NUREG series reports and technical reports prepared by other federal agencies and reports prepared by the Atomic Energy Commission, forerunner agency to the Nuclear Regulatory Commission.

Documents available from public and special technical libraries include all open literature items, such as books, journal and periodical articles, and transactions. *Federal Register* notices, federal and state legislation, and congressional reports can usually be obtained from these libraries.

Documents such as theses, dissertations, foreign reports and translations, and non-NRC conference proceedings are available for purchase from the organization sponsoring the publication cited.

Single copies of NRC draft reports are available free, to the extent of supply, upon written request to the Division of Technical Information and Document Control, U.S. Nuclear Regulatory Commission, Washington, DC 20555.

Copies of industry codes and standards used in a substantive manner in the NRC regulatory process are maintained at the NRC Library, 7920 Norfolk Avenue, Bethesda, Maryland, and are available there for reference use by the public. Codes and standards are usually copyrighted and may be purchased from the originating organization or, if they are American National Standards, from the American National Standards Institute, 1430 Broadway, New York, NY 10018.

GPO Printed copy price: $6.50

NUREG/CR-3686
UCRL-15597 Vol. 1
Intramural #3371609

WIPS—Computer Code for Whip and Impact Analysis of Piping Systems

Part A – User's Manual

Manuscript Completed: March 1983
Date Published: June 1984

Prepared by
G. H. Powell, J. P. Hollings, D. G. Row, P. Chen, F-C. Hu, M. Mahasuverachai, B. Mosaddad, P. Nicklin, S. Nour-Omid, C. Oughourlian, and A. Riahi;
University of California, Berkeley, CA

Lawrence Livermore National Laboratory
7000 East Avenue
Livermore, CA 94550

Prepared for
Division of Engineering Technology
Office of Nuclear Regulatory Research
U.S. Nuclear Regulatory Commission
Washington, D.C. 20555
NRC FIN No. A0136

WIPS - COMPUTER CODE FOR WHIP AND IMPACT ANALYSIS OF PIPING SYSTEMS

PART A

USER'S MANUAL

by
G. H. Powell, J. P. Hollings, D. G. Row, P. Chen, F-C. Hu, M. Mahasuverachai, B. Mosaddad, P. Nicklin, S. Nour-Omid, C. Oughourlian, and A. Riahi

University of California,
Berkeley, California

Prepared for
Lawrence Livermore National Laboratory
Livermore, California
under LLL Subcontract No. 3371609

ACKNOWLEDGEMENTS

The authors wish to acknowledge the assistance and patience of Program Managers P. Albrecht, M. Vagins, and G. Weidenhamer of the Nuclear Regulatory Commission and P. Smith, C-K. Chou, and T-Y. Chuang of the Lawrence Livermore National Laboratory.

Special acknowledgement is due to R. Chun, Lawrence Livermore National Laboratory, for his invaluable assistance in running examples on the CRAY computer and for his extremely thorough review of the manuscript.

L. Calvin extended herself above and beyond the call of duty to prepare the manuscript. The good figures were drafted by G. Feazell.

WIPS - COMPUTER CODE FOR WHIP AND IMPACT ANALYSIS OF PIPING SYSTEMS

PART A

USER'S MANUAL

ABSTRACT

WIPS consists of a series of sixteen computational modules, executed under the control of the WIPS-EXEC executive program. Twelve of the modules are used for specification of piping system properties and analysis control information. These modules constitute the WIPS-INPT package. They are executed interactively, in an appropriate sequence, in order to produce an input data file for the WIPS-ANAL structural analysis module. WIPS-ANAL is typically executed in batch mode, and produces certain output files. The remaining three modules can then be executed interactively to produce tabular and graphical results from the WIPS-ANAL output.

Sections A1 through A10 of this manual present instructions and examples on the use of the WIPS modules. In addition, it is possible to prepare an input data file for WIPS-ANAL without using the WIPS-INPT package, and to run WIPS-ANAL as a stand-alone code. Instructions for direct preparation of WIPS-ANAL input data are presented in Section A11.

TABLE OF CONTENTS

A1. FEATURES AND LIMITATIONS

SUMMARY

This section describes the general features and limitations of the WIPS code.

CONTENTS

A1.1 GENERAL FEATURES

WIPS is a special purpose computer code for analysis of the nonlinear dynamic response of piping systems subjected to strong jet forces. The code can be used to determine: (1) the overall motions of a piping system; (2) the detailed deformations of individual piping components; and/or (3) the impact forces and local deformations at points of pipe-to-pipe or pipe-to-wall impact.

Figure A1.1 illustrates problems which might be solved using WIPS. Figure A1.1a shows the pipe centerlines for a small piping system, in which a guillotine break is postulated in the branch pipe at C.

Figure A1.1b shows a mathematical model which might be used to analyze the overall response of the piping system. The piping components are idealized using one-dimensional beam type elements. The model allows for interconnection between the two sides of the break through pipe run EOPJ, and includes a pipe whip restraint at M.

Figure A1.1c shows a simpler model, ignoring interconnection through EOPJ, and assuming a virtual anchor at I. With this type of model, it is assumed that the whip restraint controls the motion of the pipe, so that significant deformations occur only in run CMNI.

Figure A1.1d shows a model in which the elbow at N is divided into finite elements in order to obtain detailed information on the elbow behavior.

Figure A1.1e shows a similar model, but with the elbow at M divided into finite elements, and with a rigid wall located some distance from the elbow. With this model, an analysis can be made of the impact forces and local pipe deformations following impact with the wall.

Figure A1.1f shows a model in which straight lengths of pipe in run CMNI and a run of an adjacent piping system are both divided into finite elements. This type of model can be used to determine impact forces, local deformations, and overall deformations in both pipe runs following pipe-to-pipe impact. This impact analysis capability is one of the most important features of the WIPS code.

A1.2 SPECIFIC FEATURES

A1.2.1 USER FEATURES

From a user's viewpoint, WIPS has the following major features:

(1) Input data is accepted interactively in free form, typically through a CRT terminal. The code prompts the user and provides help messages if needed. The data is subjected to extensive error checking.

(2) Geometric data can first be specified for an entire piping system and saved on permanent file. Any part of the system can then be extracted for analysis, with a minimum of additional input data. Any number of different analyses can be carried out on parts of a single system. The system geometric data can be modified at any time to change the geometry, enlarge the system, or correct errors.

(3) Piping components can be modeled by beam-type elements (i.e. essentially 1D elements) or by meshes of 3D shell elements. The shell element meshes can be generated with a small amount of input data.

(4) Strain rate effects can be included when the elements yield. Large displacement effects can be considered for both beam and shell elements.

(5) If piping components are modeled using meshes of shell elements, analyses of pipe-to-pipe impact may be carried out. The impact analysis takes into account large local deformations at the impact location and allows for sliding of the pipes relative to each other during impact. Analyses of pipe impact on either rigid or deformable walls or slabs can be carried out. Deformable walls and slabs are modeled using shell elements.

(6) The dynamic response is calculated by step-by-step integration through time. The solution algorithm automatically varies the time step, using large steps when possible and small steps when necessary.

(7) The analysis may be stopped and restarted if desired.

(8) The analysis results are saved on permanent file and may be post-processed in a variety of ways. Both time-history graphs and drawings of deflected shapes may be obtained.

A1.2.2 TECHNICAL FEATURES

WIPS has the following major technical features:

(1) The code is structured as a central analysis module, linked by a data base to input preprocessors and results post-processors. The data base consists of a series of files, each with a well-defined structure, which are generated by the various code modules. New pre- and post-processor modules can be developed relatively easily.

(2) The analysis module is based on the displacement method, with multi-level substructure capability. Piping components which are modeled as meshes of shell elements are treated as substructures. The module consists of a series of base subroutines, which control the solution logic and manage the data, linked to a library of nonlinear finite elements. The logic for the finite element computations has been largely standardized, so that new elements can be added relatively easily.

(3) The analysis module makes use of a sophisticated data base manager developed specifically for structural analysis applications. The pre- and post-processing modules make use of subroutine packages for free-form input, management of simple data bases, and file management. The programming language is FORTRAN 77. The pre- and post-processing modules execute in single precision, and the analysis module in double precision.

(4) The analysis module incorporates a substantially automatic, implicit step-by-step algorithm for dynamic response calculation. Equilibrium errors at the end of any step are controlled by an event-to-event strategy, in which the stiffness matrix is modified each time a significant nonlinearity occurs. Time step selection is based on an estimate of the mid-step equilibrium error. If this error exceeds a user-specified upper tolerance, the time step is reduced. If the error is less than a lower tolerance, the time step is increased.

A1.2.3 LIMITATIONS

WIPS has the following major limitations;

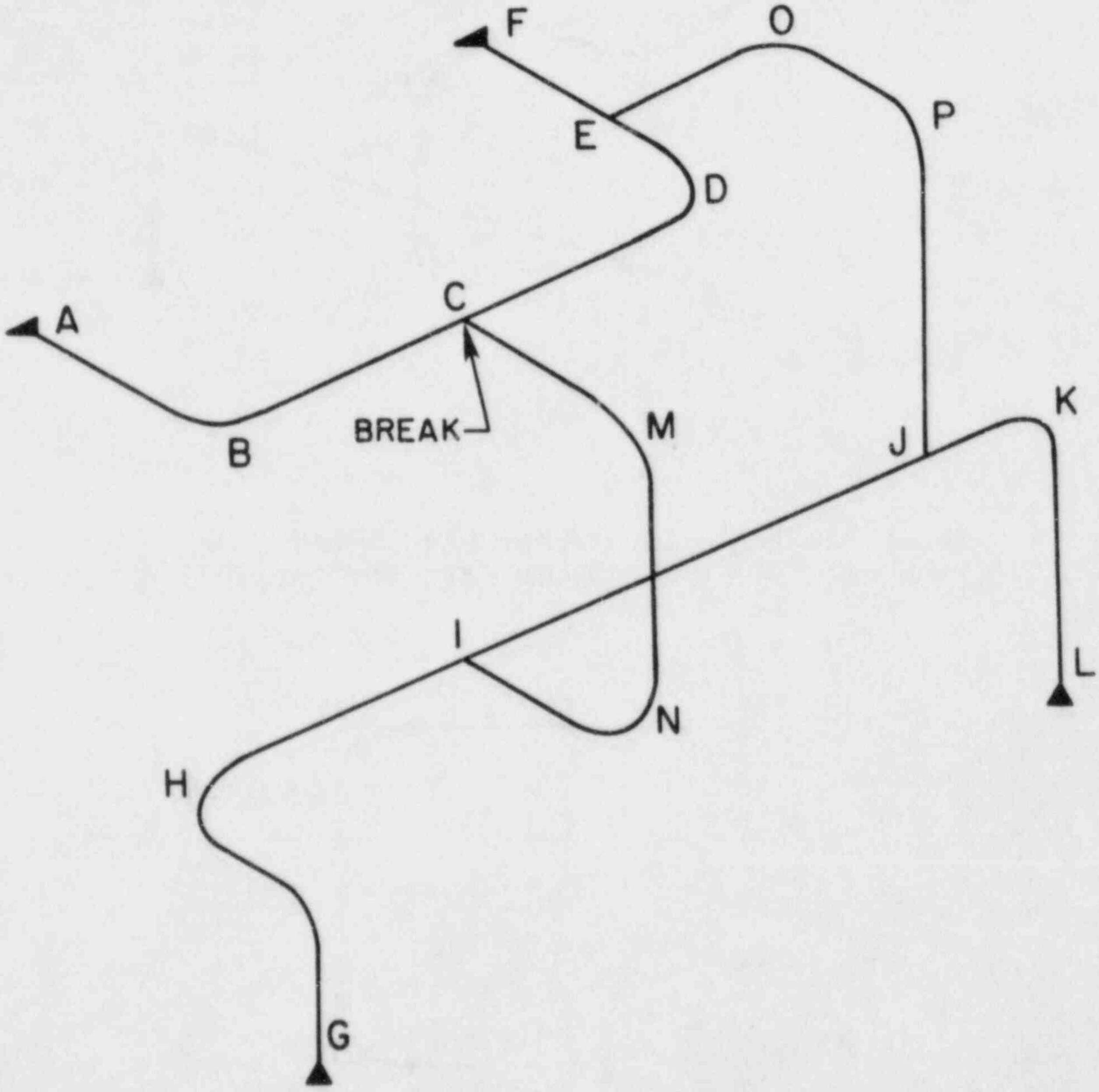
(1) The material model used in the analysis of steel components accounts for nonlinear strain hardening and strain rate effects in a quite general way, but nevertheless has limited capabilities. In particular, the model assumes unlimited ductility and has no direct capability to predict when fracture may occur.

(2) The element library enables pipe components and restraints to be modeled in a number of different ways. However, the elements all have limitations and may not provide all of the features required by a particular analyst. In particular, although both elbows and straight pipe segments may be modeled with finite element meshes using shell elements, there is currently no provision for such detailed modeling of branch connections.

(3) For analysis of walls and slabs, there is currently no provision for a concrete material model, and the slab can be idealized using only thin shell elements. Hence, analyses of detailed damage to concrete walls cannot be carried out.

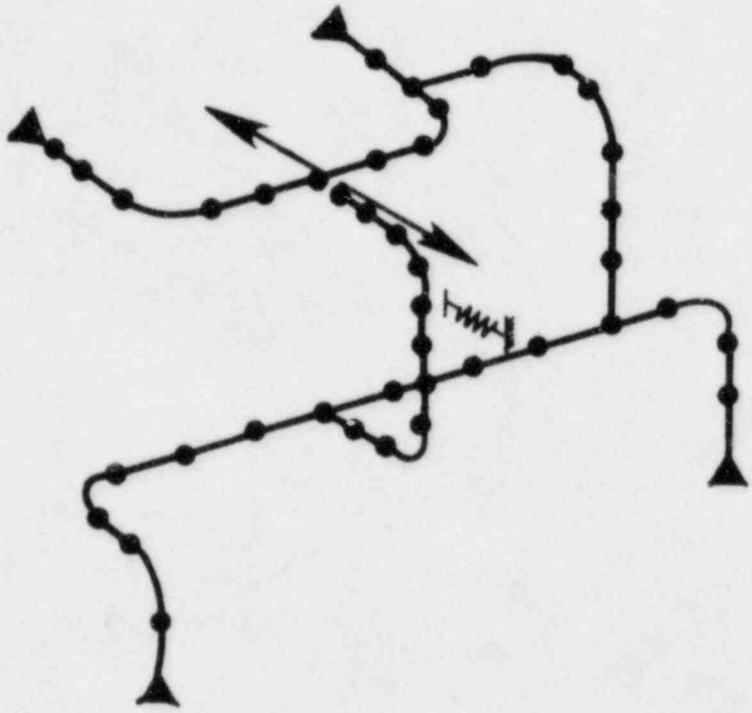
(4) Contact analysis can currently be carried out readily only for straight pipe, elbow and slab geometries (other geometries can be considered, but the amount of data preparation effort increases greatly). The contact analysis algorithm may also have difficulty if the impacting pipes become very severely distorted (i.e. nearly flattened).

(5) Analyses can be carried out only for predetermined jet force magnitudes. There is no provision to consider fluid-structure interaction.

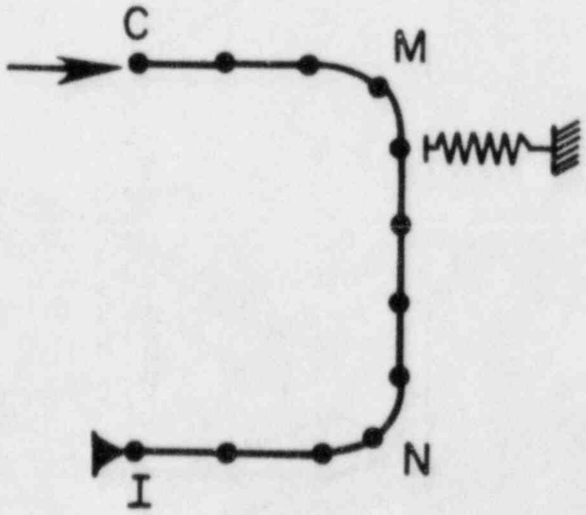


(a) EXAMPLE SYSTEM

FIG. A1.1 - WIPS ANALYSIS SCOPE

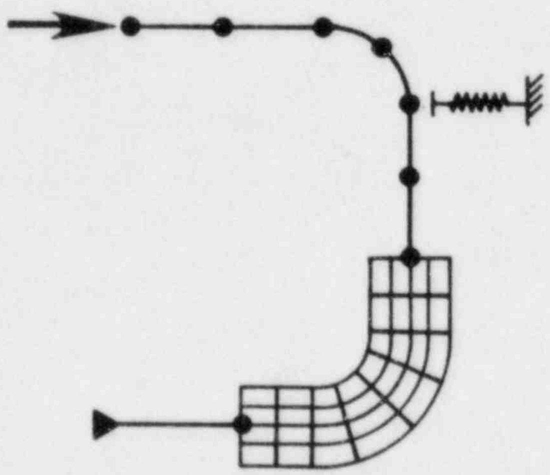


(b) OVERALL RESPONSE OF COMPLETE SYSTEM

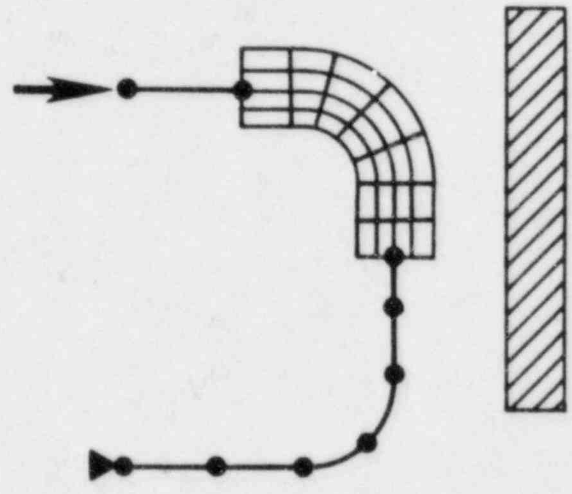


(c) RESPONSE OF SINGLE PIPE RUN

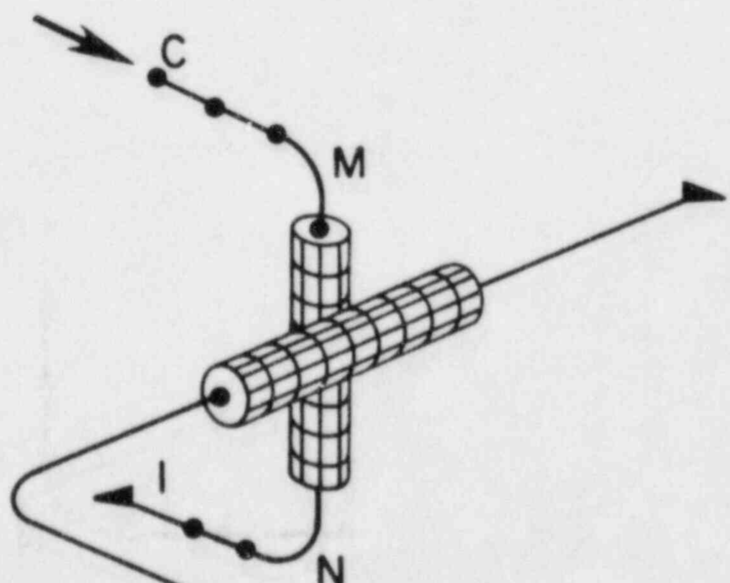
FIG. A1.1 - WIPS ANALYSIS SCOPE (CONT'D)



(d) DETAILED MODELLING OF ELBOW



(e) DETAILED MODELLING OF IMPACT ON RIGID WALL



(f) DETAILED MODELLING OF PIPE-TO-PIPE IMPACT

FIG. A1.1 - WIPS ANALYSIS SCOPE (CONT'D)

A2. COMPUTATIONAL MODULES AND INPUT DATA SEQUENCE

SUMMARY

The chapters in this section provide an overview of the structure of the WIPS code and the steps to be followed in using the code. Detailed user instructions are presented in Sections A3 through A10.

CONTENTS

A2.1 WIPS-EXEC

A2.1.1 GENERAL

WIPS consists of a number of separate computational modules, which share a common data base and are linked through an executive program.

A WIPS user controls the computation through the WIPS-EXEC executive program. WIPS-EXEC maintains control over the data base, and grants access to the computational modules which make up the WIPS system. After the execution in any module is complete, or if a fatal error is detected, control is returned to WIPS-EXEC.

WIPS accepts piping system data and analysis control commands interactively. A problem can be set up and run in a single computer session, or can be defined over a period of time in several separate sessions. Typically, data on the system geometry, pipe component properties, jet force time-histories, etc. will first be input and stored in the data base. When sufficient data is available, commands will be input to construct a specific mathematical model and to analyze this model. If desired, several different mathematical models can be constructed from one set of system data, to consider different break locations or to explore the effects of changing the modeling assumptions.

A2.1.2 WIPS MODULES

The following computational modules may be called from WIPS-EXEC.

- GEOM: accept data on spatial layout of piping system, calculate node coordinates, and plot pipe centerlines.
- MATL: accept data on material properties.
- PIPE: accept data on properties of *pipe* type elements.
- BEAM: accept data on properties of *beam* type elements.
- UBAR: accept data on properties of U-bar *(ubar)* elements.
- GAPF: accept data on properties of gap-friction *(gapf)* elements.
- ELBO: generate finite element meshes for elbow substructures.
- STRP: generate finite element meshes for straight pipe substructures.
- SLAB: generate finite element meshes for flat slab substructures.
- FREC: accept jet force time histories.
- MODL: create a *mathematical model* to be analyzed.
- DATA: create an input data file for the WIPS-ANAL module.
- ANAL: perform step-by-step nonlinear dynamic analysis.
- RSLT: process results, in either tabular or graphical form.
- MERG: read results from a formatted results file and write to an unformatted file.
- PURG: purge unwanted results from a results file.

A2.1.3 DATA FILES

A2.1.3.1 File Types

For any given pipe whip problem, data files of various types are produced by the WIPS computational modules and stored in the data base. Files of the following types may be produced.

- GEOM: geometry data as input.

COOR: nodal coordinates.
MATL: material properties.
PIPE: properties of straight or curved *pipe* type elements.
BEAM: properties of *beam* type elements.
UBAR: properties of U-bar *(ubar)* elements.
GAPF: properties of gap-friction *(gapf)* elements.
ELBO: properties of elbow substructures.
STRP: properties of straight pipe substructures.
SLAB: properties of flat slab substructures.
FREC: jet force time-histories.
MODL: data defining analysis model.
DATA: input data for WIPS-ANAL analysis module.
RSLT: time-history results from WIPS-ANAL.
ECHO: echo of WIPS-ANAL input data.
SLOG: log of WIPS-ANAL solution steps.
PAUS: optional pause file for restart.
PAUZ: optional pause file for restart.

A2.1.3.2 File Identification

Each data file is identified by an eight-character identifier of the form TTTTPPSS, where TTTT = file type, PP = problem number, and SS = sequence number. For example, file GEOM0204 is a GEOM file cataloged under problem number 2 and is the 4th GEOM file to be created under this problem.

For the MATL, PIPE, BEAM, UBAR, GAPF, ELBO, STRP, SLAB and FREC files, only one file is created for any problem, and a sequence number of zero is used (e.g. MATL0200).

Within any problem, files with zero sequence numbers are identified by file type only, and files with sequence numbers by file type and sequence number.

Each file is automatically cataloged by WIPS-EXEC, with its type, problem number, sequence number, date and time of creation, and an optional comment provided by the user. The file catalog can be listed at any time, and obsolete files can be deleted.

A2.1.4 WIPSLOG AND WIPSCAT FILES

During any WIPS session, all data displayed on the CRT terminal is automatically written to a formatted file named WIPSLOG, to provide a log of the session. A hard copy of the session log can be obtained, if desired, by routing WIPSLOG to a lineprinter. At the beginning of each new session, any existing WIPSLOG data may be deleted or, alternatively, it may be retained. If it is retained, new data is added at the end of the file so that WIPSLOG contains a log of two or more sessions. Note that WIPSLOG is not automatically routed to a lineprinter. This must be done using computer operating system commands, not under WIPS. WIPSLOG may also be displayed on the terminal, if desired, using operating system commands.

WIPS-EXEC also produces a file named WIPSCAT. This file contains the catalog of the files created for the piping system, with their sequence numbers, dates of creation, etc. WIPSCAT is unformatted and cannot be printed or displayed.

A2.2 COMPUTATIONAL MODULES

A2.2.1 GEOM: SPECIFICATION OF SYSTEM GEOMETRY

WIPS-GEOM accepts data on the spatial layout of the piping system. Data files of two types may be produced, namely GEOM and COOR.

GEOM files contain "raw" data on the pipe system layout, before the data is processed to produce X, Y, Z spatial coordinates. GEOM files can be added to and/or changed as desired to add further data or to change incorrect data. Several different GEOM files can be set up for a single problem if desired.

A COOR file can be produced on command from any GEOM file. COOR files contain X, Y, Z coordinates of all points in the piping system, plus other geometrical data.

A2.2.2 PIPE COMPONENT AND SUPPORT PROPERTIES

Tables of stiffness and strength properties for a number of different pipe component and support elements may be specified using a different WIPS module for each element type. For example, WIPS-PIPE accepts properties for straight or curved pipe elements, and WIPS-UBAR accepts properties for pipe whip restraints of U-bar type. Data files of corresponding type are produced by each module. WIPS-MATL accepts data on material properties for use by other modules. Previously created files can be recalled as desired, and new properties added. The new file then replaces the previous one.

A2.2.3 SUBSTRUCTURE PROPERTIES

For detailed analysis of pipe components, lengths of pipe may be subdivided into finite element meshes and modeled using shell elements. Each such length of pipe constitutes a substructure for analysis. Substructures made out of shell elements can also be defined to represent walls and slabs for analysis of pipe impact.

The dimensions and properties of each different substructure type are accepted using a different WIPS module for each type and stored in files with corresponding names. Modules are currently available for straight pipe (WIPS-STRP), pipe elbows (WIPS-ELBO), and walls or slabs (WIPS-SLAB). It is anticipated that modules for other substructure types will be added in future versions of WIPS.

STRP and ELBO substructures are specified in a local coordinate system. Substructures may then be positioned with arbitrary locations and orientations when an analysis model of the piping system is defined.

A2.2.4 FREC: SPECIFICATION OF FORCE RECORDS

WIPS-FREC accepts data on jet force time-histories, as force-time pairs, and produces a file of FREC type. Only one FREC file is produced. This file can be recalled at any time and new records can be added. The new file then replaces the previous file.

A2.2.5 MODL: SPECIFICATION OF ANALYSIS MODEL

WIPS-MODL accepts data defining a mathematical model for analysis. An analysis model can consist of all or part of the piping system. Typically, an analysis model will begin at a postulated pipe break and include sufficient adjacent piping to ensure that an accurate analysis is obtained. The model may include adjacent walls or slabs for impact analysis.

WIPS-MODL makes use of a COOR file from WIPS-GEOM, files of pipe component and support properties, and files of substructure properties. The analyst defines the parts of the complete system to be considered in the analysis model, the pipe component and support properties (referring to previously defined property tables), boundary conditions, and other data, in a series of commands. The commands are processed to produce a MODL file, which constitutes part of the input data for the WIPS-ANAL analysis module. Any number of different analysis models can be set up for a single piping system, to consider different parts of the system or different finite element models for a single part.

A2.2.6 DATA: PREPARATION OF INPUT DATA FILE

The MODL file contains information on the geometry and physical properties of the analysis model but does not contain information on the loading, the duration of the dynamic response, etc. The WIPS-DATA module adds this information to an existing MODL file to produce a DATA file containing complete input data for the WIPS-ANAL module. A single MODL file can be used to produce several different DATA files, as, for example, when the effects of several different jet force histories are to be considered for a single model.

A2.2.7 OTHER FILES SET UP BY WIPS-DATA

WIPS-DATA also sets up and initializes other files used by WIPS-ANAL. These are as follows:

(1) ECHO, for echoing of the input data by WIPS-ANAL. Because a WIPS user will not normally see the WIPS-ANAL input data, the ECHO file will usually be ignored. The file is intended for use in debugging, if WIPS-DATA should prepare a DATA file which contains errors and is not accepted by WIPS-ANAL.

(2) SLOG, for saving a log of the solution sequence followed by WIPS-ANAL during the step-by-step dynamic analysis. Each time WIPS-ANAL performs a computational step, a descriptive line is written on the SLOG file. A typical user will ignore the SLOG file. It is intended mainly for debugging, in the event that numerical difficulties develop in the step-by-step solution.

(3) RSLT, for saving histories of the structure displacements and finite element responses by WIPS-ANAL during the step-by-step analysis. The RSLT file can be processed by the WIPS-RSLT module to produce tabulated and plotted results.

(4) Two "pause" files, of PAUS and PAUZ types, for saving the complete current state if a "pause" command is given to WIPS-ANAL. These files are optional and are set up only if it is desired to pause in the analysis and then restart in a later computer run.

A2.2.8 ANAL: STEP-BY-STEP DYNAMIC ANALYSIS

WIPS-ANAL makes use of an existing DATA file, together with associated ECHO, SLOG, and RSLT files. For restart, the PAUS and PAUZ files are also used. WIPS-ANAL reads input data from the DATA file, performs the execution, and writes information on the ECHO, SLOG, RSLT, PAUS, and PAUZ files.

Because the execution times are likely to be long, WIPS-ANAL will typically be executed in background or batch mode. The other modules all run in interactive (foreground) mode.

A2.2.9 PAUSE AND RESTART

If the duration of significant dynamic response for a given model and loading is not known in advance, it can be specified that WIPS-ANAL consider a specified time period, then pause. The analysis may then be restarted to run for one or more additional time periods.

The time period to be considered and the pause requirement are specified in the WIPS-DATA phase. If a pause is specified, WIPS-ANAL saves the state at the end of the analysis on the PAUS and PAUZ files (by copying the WIPS-ANAL data base). If an additional time period is to be considered, WIPS-DATA can be used to create a modified DATA file, instructing WIPS-ANAL to restart from the saved state.

A2.2.10 RSLT: RESULTS POST-PROCESSING

WIPS-RSLT post-processes any RSLT file from WIPS-ANAL to produce final tabulated and/or plotted results. Tables can be produced showing variations with time of any response quantities (displacements or finite element results), time-histories of any response quantities may be plotted, and deflected shapes at any specified times during the response may be drawn.

A2.2.11 MERG: COPY FORMATTED RESULTS DATA TO RSLT FILE

The RSLT file is unformatted for computational efficiency. This creates no problems when all phases of WIPS are executed on a single computer. For large problems, however, it may be desirable to prepare the DATA, ECHO, SLOG and RSLT files on a minicomputer, transfer the DATA, ECHO and SLOG files to a mainframe computer for execution, and return the results to the minicomputer for post-processing. In such cases it may be necessary to transfer the results as a formatted file, because unformatted files produced on different computers will generally not be compatible. Hence, it is not possible to transfer the RSLT file directly. The WIPS-MERG module allows formatted results data to be copied to an unformatted RSLT file in the form required by WIPS-RSLT.

A2.2.12 PURG: PURGE UNWANTED DATA FROM RESULTS FILE

Occasionally WIPS-ANAL will fail to execute properly and unwanted data may be present on a RSLT file. WIPS-PURG allows the unwanted data to be purged.

A2.2.13 UNITS

Data may be specified in a number of different length and force units. At the beginning of execution in any module, the units must be specified, and these units must be used during the execution of that module. If desired, the units may be different in different modules, or during different computer runs using a single module. For example, if both U.S. and metric pipe sizes are to be considered, WIPS-PIPE may be executed using inch units for the U.S. sizes, then executed a second time with centimeter units for the metric sizes.

A2.3. FREE FORM INPUT

A2.3.1 GENERAL

A2.3.1.1 Prompting

Data is always input in response to a prompt from WIPS. A colon (:) always indicates that data is to be input.

If the colon is preceded by a question (i.e.?:), a "yes" or "no" answer is always required. In this case, "Y" or "YES" constitute a "yes" response and "N", "NO", or blank constitute a "no" response.

If the colon is not preceded by a question, a list of required data will be indicated. This list will usually be self-explanatory. In some cases (particularly for WIPS-MODL), however, the user will need to study the User's Guide in advance and have it at hand when entering data.

A2.3.1.2 On-Line Help

If the word "HELP" is typed as the first item following the colon, a message is displayed to clarify what is required.

A2.3.2 FORM OF INPUT LINE

A2.3.2.1 Cases Without Option Identifiers

In general, an input line can consist of any mixture of integer, real, and alpha data items. In most cases, the data items must be entered in a specific sequence. In some cases, however, *option identifiers* may be used, and the affected data can be entered in any sequence.

For an input line without option identifiers, the following rules apply.

(1) Data items must be separated by either a blank or a comma.

(2) Integer items must consist of numbers only, with no decimal point or other characters.

(3) Real items can be in FORTRAN F or E form. For example, 1.234 may be entered as 1.234, .1234E1, 1234.E-03, etc. There must be no blanks. If the number is an integer, the decimal point may be omitted.

(4) Alpha items can consist of any alpha-numeric characters, up to a maximum of 4. Characters in excess of 4 are ignored. If fewer than 4 characters are entered, trailing blanks are added.

(5) If the number of items entered is less than the number requested, the missing items are read as zero or blank.

(6) If two successive commas (,,) are entered, the corresponding item is read as zero or blank. The corresponding items are also read as zero or blank for three or more successive commas (e.g. ",,,," = 3 zero or blank items).

(7) If the first entry in the line is a comma, the first item is read as zero or blank.

(8) An input line must consist of no more than 80 characters.

(9) If desired, an optional comment can be added at the end of the line. This comment is printed in the session log but is otherwise ignored. To use this option, leave 3 blanks after the last data item, then add the comment.

A2.3.2.2 Option Identifiers

In some cases a data line may contain optional items, which may or may not be specified. In this case each optional item must be preceded by an option identifier, and the items may be entered in any convenient sequence. Consider, for example, the following line:

5, 4.6E3, PROP=3, WFAC=1.5, THIS=YES

In this line the first two items are required, and the remainder are optional. The words "PROP", "WFAC", and "THIS" are option identifiers. The "=" symbol relates each identifier to its associated data and must always follow the identifier. Any optional items not specified are undefined. The WIPS modules are written to default undefined items to predetermined values where possible, or else to display an error message.

Option identifiers are currently used only in the WIPS-MODL phase.

A2.3.2.3 Error Messages

If improper data is entered in any line (e.g. decimal point for an integer item), the improper data is flagged, and the data line must be re-entered.

A3. WIPS-EXEC USER GUIDE

SUMMARY

Maintenance of WIPS files and execution of the WIPS modules are controlled through WIPS-EXEC. This section contains a user's guide for WIPS-EXEC. User guides for the other WIPS modules are contained in Sections A4 through A10.

Section A3.1 describes the procedures for using WIPS-EXEC under the VAX-UNIX operating system. A new section must be added for each different operating system.

CONTENTS

A3.1 WIPS-EXEC USER GUIDE: VAX-UNIX SYSTEM

A3.1.1 BEGIN EXECUTION

A3.1.1.1 Upper and Lower Case

The VAX-UNIX operating system allows either upper or lower case alpha characters. For WIPS execution under VAX-UNIX, use *lower case* for all input data *except for file names* (which are upper case).

A3.1.1.2 Problem Directory

A separate UNIX directory should be established for WIPS problems. Several different problems may be run from a single directory or, if it is preferred, a new directory can be established for each problem.

A3.1.1.3 Initiation of WIPS-EXEC

To begin execution, type "wips". The message "EXEC-WIPS EXECUTIVE" will be displayed.

A3.1.1.4 Problem Number

If this is the first problem in the directory, a problem description will be requested. If this is not the first problem in the directory, problem number 1 is assumed. The problem number may be changed using the "problem" command (see Section A3.1.1.7).

A3.1.1.5 Purging of WIPSLOG

The question "Purge WIPSLOG file?:" is displayed next. If the response is "yes", any data currently in WIPSLOG is deleted. If the response is "no", any data in WIPSLOG is retained, and the data for the current session is added at the end of the file. This allows session logs for several sessions to be accumulated, if desired.

A3.1.1.6 Next Command

The message "NEXT WIPS-EXEC COMMAND" is displayed next (and also each time a WIPS-EXEC command is completed). The available commands are listed in Section A3.1.2.

A3.1.1.7 Problem Number

The default problem is problem number 1. To select a different problem, specify the WIPS-EXEC command "problem". The problem number will then be requested. If there is no existing problem with this number, a problem description will be requested.

A3.1.2 WIPS-EXEC COMMANDS

The WIPS-EXEC commands are as follows.

help: Display WIPS-EXEC commands.

list: Display names, dates, and descriptions of all "active" files in the current problem. A file becomes "inactive" if it is deleted.

delete: Delete one or more files. Following this command, requests for the type and sequence number (if needed) of the file(s) to be deleted will be displayed. To stop, return a blank file type. If a file is deleted, it is removed from the file set and cannot be recovered. However, a reference to the file remains in the WIPSCAT catalog.

listprob:	Display the descriptions of all problems.
listall:	Display names, etc. for all files, both active and inactive, in the current problem.
listcat:	Display names, etc. for all active files in all problems.
catalog:	Catalog a file not produced by WIPS for the current problem. See Section A3.1.3.
problem:	Change to a different problem. See Section A3.1.1.7.
geom:	Execute WIPS-GEOM. See section A4.
matl:	Execute WIPS-MATL. See Section A5.1.
frec:	Execute WIPS-FREC. See Section A5.2.
pipe:	Execute WIPS-PIPE. See Section A6.1.
beam:	Execute WIPS-BEAM. See Section A6.2.
ubar:	Execute WIPS-UBAR. See Section A6.3.
gapf:	Execute WIPS-GAPF. See Section A6.4.
strp:	Execute WIPS-STRP. See Section A7.1.
elbo:	Execute WIPS-ELBO. See Section A7.2.
slab:	Execute WIPS-SLAB. See Section A7.3.
modl:	Execute WIPS-MODL. See Section A8.
data:	Execute WIPS-DATA. See Section A9.1.
anal:	Execute WIPS-ANAL. See Section A9.2.
rslt:	Execute WIPS-RSLT. See Section A10.
quit:	End execution of WIPS.

The commands may be executed in any sequence, subject to the restriction that certain WIPS modules depend on files produced by other modules. If any improper sequence is specified, sooner or later it will be detected and control returned to WIPS-EXEC.

A3.1.3 CATALOG COMMAND

A3.1.3.1 Purpose

Files produced for any problem by WIPS are automatically cataloged in the WIPSCAT file and are available for use by any WIPS module. It is also possible to transfer files from one problem to another using the *catalog* command. For example, it might be desirable to setup a MATL file under one problem and then catalog copies of this file under one or more other problems.

A3.1.3.2 Procedure

The "catalog" command must be used carefully because WIPS-EXEC does not check for consistency. The procedure is as follows.

(1) Enter WIPS-EXEC and give the "catalog" command. The file type is requested. WIPS-EXEC assigns a file name, using the file type, the current problem number, and the next available sequence number for the file type (if needed). WIPS-EXEC then displays the assigned name.

(2) Exit from WIPS and, using operating system commands (e.g. UNIX "copy" command), create a file with the assigned file name (e.g. copy MATL0100 MATL0200; copy GEOM0103 GEOM0302).

(3) To exit from the "catalog" command, return a blank file type.

A4. WIPS-GEOM USER GUIDE

SUMMARY

This section describes the geometrical definition of a piping system and presents instructions and an example for data input to the WIPS-GEOM module.

CONTENTS

A4.1 GEOMETRICAL DEFINITION OF PIPING SYSTEM

A4.1.1 PIPE RUNS

For the purpose of defining a piping system to WIPS, the system is regarded as an assemblage of *pipe runs*, each consisting of a continuous sequence of piping components. For example, the system shown in Fig. A4.1.1 can be regarded as an assemblage of pipe runs, as follows:

Run 1: Point A to Point F

Run 2: Point G to Point L

Run 3: Point C to Point I

Run 4: Point E to Point J

This is not the only possible breakdown of the system into runs, but it is the most natural, and is the one which would probably be used. If, however, the run from, say, Point G to Point L was particularly long, it could be divided into two or more runs (for example, from Point G to Point I and from Point I to Point L). The only restriction in dividing pipe runs in this way is that if an elbow or straight pipe substructure is defined, it must lie entirely within a single run. Hence, runs should not be subdivided at points where substructures may possibly be located in subsequent analyses.

A single pipe run could also be specified to run, for example, through Points A, B, C, M, N, I, H, and G in Fig. A4.1.1. However, this is not a natural way to describe a piping system and is not recommended.

A single pipe run may, if desired, form a closed loop. However, it is generally recommended that loops be divided into two or more runs.

A4.1.2 CONTROL POINTS

A4.1.2.1 Control Points Along Pipe Runs

Each pipe run is defined by a sequence of *control points*. A control point *must* be placed at every discontinuity in the pipe run, as follows:

(1) At the two ends of the run.

(2) At all *branch points* in the run.

(3) At all *tangent points* where straight pipe connects to an elbow or curved pipe.

(4) At all points where the pipe cross section or material properties change.

(5) At all points where elbow or straight pipe substructures begin or end.

(6) At all support points or points with pipe whip restraints.

(7) At all points where substantial lumped masses are located (e.g. valves, flanges).

These control points automatically become node points for the subsequent structural analysis. Control points *may* be placed at other locations in the run, if desired. However, it is not necessary to place a control point at every point which is to be a node for the structural analysis. Node points along straight and curved segments of pipe can be generated automatically without being defined as control points.

A4.1.2.2 Other Control Points

In addition to control points along the pipe runs, a control point must be defined at each *tangent intersection point*, where the two tangents to a curved segment intersect. Further, *reference points*, which do not lie on the pipe run, may be specified if desired. Reference points will typically be used to assist in specifying the geometry of the piping system. Tangent intersection points and reference points are not nodes for the subsequent structural analysis.

A4.1.3 SPECIFICATION OF CONTROL POINT COORDINATES

A4.1.3.1 Coordinate Specification Options

The spatial layout of the pipe run is defined by the control point coordinates along the run. In the WIPS-GEOM module, a WIPS user is required to define the global X,Y,Z coordinates of all control points. Note, however, that it is not necessary to input the X,Y,Z coordinates directly for each control point, because generation options are available to simplify the procedure. Only the spatial layout is defined in the WIPS-GEOM phase. The structural elements and/or substructures along the pipe runs are defined in the WIPS-MODL phase.

The WIPS-GEOM module requests data for each pipe run in turn and for each control point *in turn along the pipe run*. The information requested on a control point is as follows:

(1) The control point name or number (a unique identifier, maximum 4 characters).

(2) The control point type, as follows.

- (a) DF or blank = default point, having no special characteristics.
- (b) TN = tangent point, where a tangent meets a bend or elbow.
- (c) TI = tangent intersection point, where the two tangents defining a bend intersect. A TI point must be specified between each pair of TN points.
- (d) RF = reference point.
- (e) BR = branch point, at a tee or branch connection. In the present version of WIPS, BR points are not given special treatment and may be treated as DF points if desired. It is anticipated that the BR distinction may be needed in future versions.

(3) The coordinate specification option to be used, as follows.

- (a) DI: direct input of X,Y,Z coordinates.
- (b) OF or blank: ΔX, ΔY, ΔZ offsets from another control point. This is the default option (i.e. blank = OF) for all except tangent points.
- (c) TN: automatic calculation of tangent point coordinates. This is the default option for a tangent point (i.e. blank = TN).
- (d) ST: generation along the straight line joining two other control points.
- (e) CU: generation along the curve joining two successive tangent points.

In addition, if a control point is repeated (as, for example, at a branch point where two runs meet), the coordinate specification DU must be used to indicate a "duplicate" point.

(4) The number of additional (unnamed) node points to be inserted between this control point and the preceding point. Node points are equally spaced between control points. Nodes may be inserted in either straight or curved segments.

(5) The coordinate data, as follows, depending on the coordinate specification option.

- (a) DI option: X; Y; Z.
- (b) OF option: name of control point from which offsets are measured (default = preceding control point); and ΔX, ΔY, ΔZ from the offset point.
- (c) TN option: no data required.
- (d) ST option: name of control point at beginning of line (default = preceding point); name of point at end of line (default = following point); and proportion of length to new control point.
- (e) CU option: as for ST, but beginning and end points must be successive TN points. The default is to the immediately preceding and immediately following TN points. This default will almost always apply.

A4.1.3.2 On-Line Help

When WIPS-GEOM requests data, the questions are usually self-explanatory. If not, type "HELP". A description of the required data will be displayed, and the question repeated.

A4.1.3.3 Correction of Errors

If unacceptable data is entered, an error message is printed and the question is repeated.

A4.1.4 GEOM AND COOR FILES

The data for each control point becomes (after some processing) a line of GEOM data. These lines can be displayed on the CRT screen and/or printed in the session log. If errors have been made, they may be corrected by modifying the GEOM data. The modification options available are:

(a) Change the data for a control point.

(b) Delete a control point.

(c) Insert one or more control points.

In addition, any GEOM data may be extended by adding more pipe runs.

When sufficient GEOM data has been entered, it may be processed to produce COOR data. In the COOR data, each line defines the X, Y, Z coordinates of a control point or a generated (unnamed) node. If it is not possible to calculate all coordinates (because of errors or missing information in the GEOM data), a message is displayed, and those points for which the coordinates could not be calculated are marked by an asterisk in the COOR data. The COOR data can be displayed and/or printed in the session log. If all coordinates can be calculated, the COOR data is "complete". If not, corrections must be made to the GEOM data, and new COOR data produced.

At the end of the WIPS-GEOM phase, the GEOM and COOR data can be saved permanently in the WIPS-EXEC data base, as GEOM and COOR files. The COOR data can be saved only if it is complete.

An option to save the GEOM data is also available before the COOR data is produced. It is generally wise to save the GEOM data at that stage, in case a fatal arithmetic error is detected by the operating system during the coordinate calculation. Under the UNIX system, if a fatal error is detected, execution of WIPS-GEOM is terminated and control returned to WIPS-EXEC. The GEOM file is then cataloged. Under other operating systems control may not be returned to WIPS-EXEC. In this case the GEOM file will exist but will not have been cataloged. The WIPS-EXEC "catalog" command must then be used.

A4.1.5 MODIFICATION OF AN EXISTING GEOM FILE

If any GEOM files exist on entry to WIPS-GEOM, the question "MODIFY EXISTING GEOM FILE?:" is asked. If the response is "NO", a new set of GEOM data is begun. If the response is "YES", a GEOM file sequence number is requested and that file becomes the GEOM data. The data may then be modified or extended as desired.

A4.1.6 GEOMETRY PLOTTING

If a "complete" set of COOR data is generated in the current WIPS-GEOM session, all or any part of the piping system may be plotted. Plotting may be performed on a Tektronix 4662 pen plotter or on Tektronix 4013, 4015, and 4027 graphics terminals. The procedure is similar to that for the DRAW option in WIPS-RSLT (see Section A10).

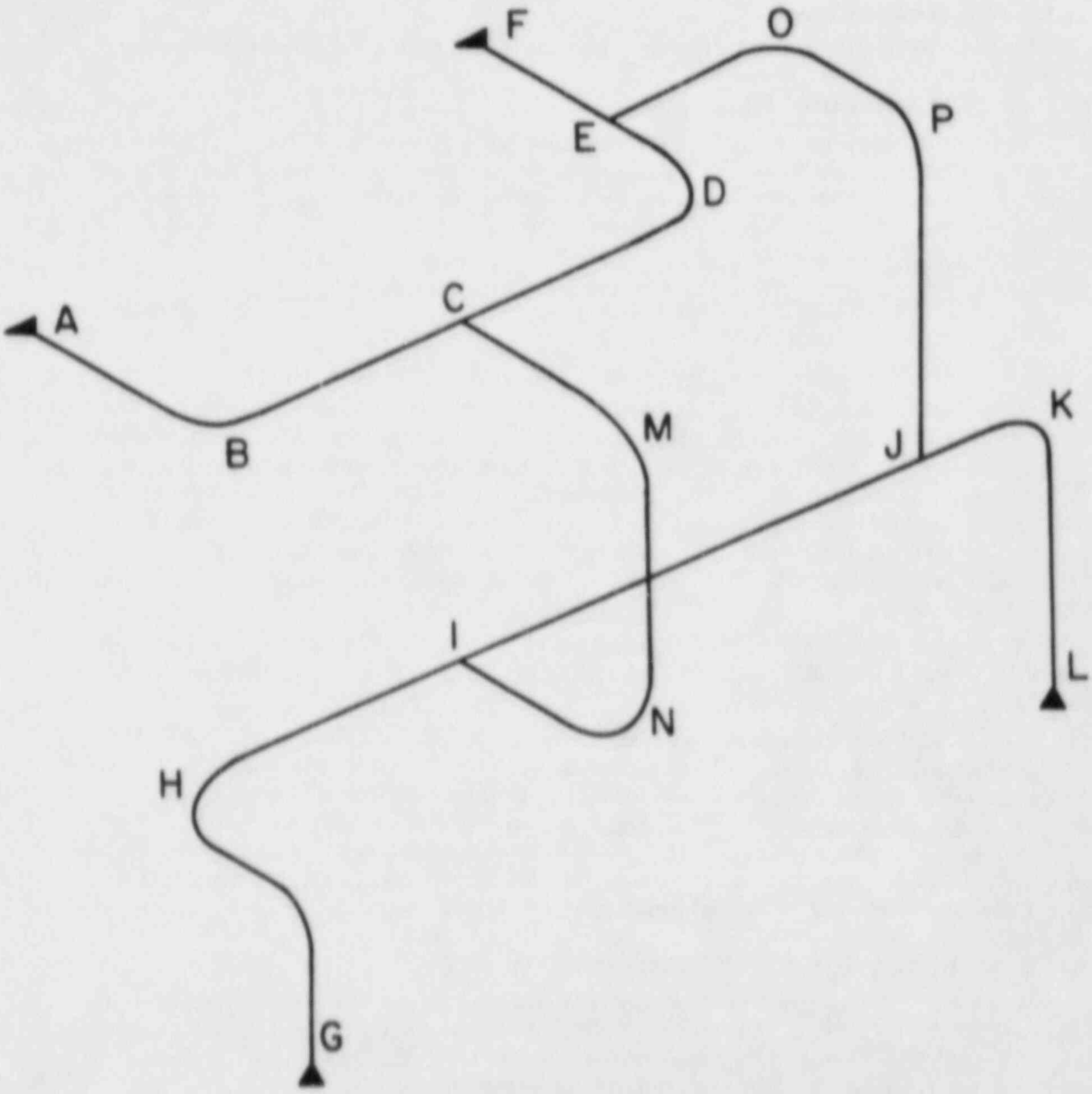


FIG. A4.1.1 - SUBDIVISION INTO PIPE RUNS

A4.2 EXAMPLE

The coordinate specification sequence is shown in Table A4.2.1 for the small piping system in Fig. A4.2.1. Table A4.2.1 is a listing of the session log. A number of "help" requests have been entered to show the type of help messages provided. Errors have been made deliberately to illustrate the procedure for correcting mistakes.

TABLE A4.2.1

WIPS-GEOM EXAMPLE

```
EXEC - WIPS EXECUTIVE
Creating problem no. 1
Problem description: Illustrative Example, Fig. A4.2.1

NEXT WIPS-EXEC COMMAND : geom

GEOM - SPECIFICATION OF SYSTEM GEOMETRY

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Specify new GEOM data

START RUN NO. 1
Enter c.p. data in sequence along run

  c.p. name and type : help
Identify c.p. by name or number, max. 4 characters
C.p. type = df, ti, tn, br, or rf. Default = df

  c.p. name and type : a1
  coord option : help
Options are: di, of, tn, du, st, and cu
  coord option : di
  x,y,z :
  Any changes? :

  c.p. name and type : t1a tn
  no. of extra nodes : 3
  coord option :
  tn generated automatically
  Any changes? :

  c.p. name and type : ti1
  no. of extra nodes :
  coord option :
  c.p., dx,dy,dz :
  offset point defaults to preceding c.p.
  Any changes? : y      DID NOT DECLARE TO BE "ti"
  reenter data for this c.p.

  c.p. name and type : ti1 ti
  bend radius : 15
  coord option :
  c.p., dx,dy,dz : a1 20 0 -40
  Any changes? :
```

```
   c.p. name and type : t1b tn
   no. of extra nodes : 1
   coord option :
   tn generated automatically
   Any changes? :

   c.p. name and type : b1 br
   no. of extra nodes :
   coord option :
   c.p., dx,dy,dz : ti1 40
   Any changes? :

   c.p. name and type : b2 br
   no. of extra nodes : 1
   coord option :
   c.p., dx,dy,dz : ,20          NOTE DEFAULT ON OFFSET POINT
   offset point defaults to preceding c.p.
   Any changes? :

   c.p. name and type : t2a tn
   no. of extra nodes : 1
   coord option :
   tn generated automatically
   Any changes? :

   c.p. name and type : ti2 ti
   bend radius : 10
   coord option :
   c.p., dx,dy,dz : b2,30
   Any changes? :

   c.p. name and type : t2b tn
   no. of extra nodes :
   coord option :
   tn generated automatically
   Any changes? :

   c.p. name and type : a2
   no. of extra nodes :
   coord option :
   c.p., dx,dy,dz : ti2,,20
   Any changes? :

   c.p. name and type :
Last c.p. in this run? : y
Display GEOM data for this run? :

START RUN NO.  2
Enter c.p. data in sequence along run

   c.p. name and type : b2 br
   coord option :
*** error - c.p. name used before - must use "du" option

   c.p. name and type : b2 br
```

```
   coord option : du
   Any changes? :

   c.p. name and type : t4a tn
   no. of extra nodes :
   coord option :
   tn generated automatically
   Any changes? :

   c.p. name and type : ti4 ti
   bend radius : 10
   coord option :
   c.p., dx,dy,dz : b2 0 -20
   Any changes? :

   c.p. name and type : b3
   no. of extra nodes : 1
   coord option :
   c.p., dx,dy,dz : a3,,-10        NOTE CAN OFFSET FROM A LATER C.P.
   Any changes? :

   c.p. name and type : a3
   no. of extra nodes :
   coord option :
   c.p., dx,dy,dz  : ti4 0 0 40
   Any changes?

   c.p. name and type :
Last c.p. in this run? : y
Display GEOM data for this run? :

START RUN NO.  3
Enter c.p. data in sequence along run

   c.p. name and type
Last run? : y

Display new GEOM data? : y

NEW GEOM DATA
```

run no.	c.p. name	c.p. type	bend radius	xtra nods	coord opt	c.p. i	c.p. j	coordinate x,dx,p	data y,dy	z,d
1										
	a1			0	di			0.	0.	0.
	t1a	tn		3	tn			0.	0.	0.
	ti1	ti	15.000	0	of	a1		20.000	0.	-40.00
	t1b	tn		1	tn			0.	0.	0.
	b1	br		0	of	ti1		40.000	0.	0.
	b2	br		1	of	b1		20.000	0.	0.
	t2a	tn		1	tn			0.	0.	0.
	ti2	ti	10.000	0	of	b2		30.000	0.	0.
	t2b	tn		0	tn			0.	0.	0.
	a2			0	of	ti2		0.	20.000	0.

```
Hit RETURN for more
```

```
  2
     b2      br                0     du                     0.          0.          0.
     t4a     tn                0     tn                     0.          0.          0.
     ti4     ti    10.000      0     of    b2               0.       -20.000        0.
     b3                        1     of    a3               0.       -10.000        0.
     a3                        0     of    ti4              0.          0.         40.00
END OF DATA

Modify GEOM data? : y       CORRECT RADIUS AT ti4
Specify modifications
Change data for a c.p.? : y
   c.p. name : ti4
   enter revised data
   c.p. type : ti
   bend radius : 5
   coord option :
   c.p., dx,dy,dz : b2,,-20
   Any changes? :

More changes? :

Display modified GFOM data? :
Write in session log? : y          NOT DISPLAYED ON SCREEN

MODIFIED GEOM DATA

run  c.p.   c.p.    bend    xtra  coord  c.p.  c.p.          coordinate  data
no.  name   type   radius   nods   opt    i     j       x,dx,p       y,dy        z,d

  1
     a1                        0     di                     0.          0.          0.
     t1a     tn                3     tn                     0.          0.          0.
     ti1     ti    15.000      0     of    a1              20.000       0.        -40.00
     t1b     tn                1     tn                     0.          0.          0.
     b1      br                0     of    ti1             40.000       0.          0.
     b2      br                1     of    b1              20.000       0.          0.
     t2a     tn                1     tn                     0.          0.          0.
     ti2     ti    10.000      0     of    b2              30.000       0.          0.
     t2b     tn                0     tn                     0.          0.          0.
     a2                        0     of    ti2              0.         20.000       0.
  2
     b2      br                0     du                     0.          0.          0.
     t4a     tn                0     tn                     0.          0.          0.
     ti4     ti     5.000      0     of    b2               0.       -20.000        0.
     b3                        1     of    a3               0.       -10.000        0.
     a3                        0     of    ti4              0.          0.         40.00

Save current GEOM data? : y       SAVE FOR SAFETY IN CASE OF CRASH
Comment for file catalog :
GEOM DATA SAVED. FILE NAME = GEOM0101

Produce COOR data? : no           PRODUCE WHEN FULL SYSTEM DEFINED

Modify GEOM data? :  no       ADD THIRD RUN IN A LATER SESSION

DATA COMPLFTE FOR THIS SESSION
```

```
Save final GEOM data? : y           ADD FILE COMMENT
Comment for file catalog : Example. Fig.A4.2.1. First 2 runs.
GEOM DATA SAVED. FILE NAME = GEOM0101

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : list

Problem 1: Illustrative Example, Fig. A4.2.1

file   sequence          date                          description
type    number
GEOM       1    Wed Dec  9 17:57:29 1981   Example. Fig.A4.2.1. First 2 runs.

NEXT WIPS-EXEC COMMAND : quit

 - -- ----- ------ - ----- --------------- -----------------------------

EXEC - WIPS EXECUTIVE

Purge WIPSLOG file? : n

NEXT WIPS EXEC COMMAND : geom               ADD THIRD RUN

GEOM - SPECIFICATION OF SYSTEM GEOMETRY

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Modify existing GEOM file? : y          GEOM0101 TO BE CHANGED
Sequence no. : 1

Display existing GEOM data? :
Write in session log? :

Modify GEOM data? : y
Specify modifications
Change data for a c.p.? :
Insert new c.p's? :
Delete a c.p.? :
Add more pipe runs? : y

START RUN NO.  3
Enter c.p. data in sequence along run

   c.p. name and type : b1 br
   coord option : du
   Any changes? :

   c.p. name and type : t3a tn
   no. of extra nodes :
   coord option :
   tn generated automatically
   Any changes? :
```

```
   c.p. name and type : ti3 ti
   bend radius : 10
   coord option :
   c.p., dx,dy,dz : b1,,,20
   Any changes? :

   c.p. name and type : t3b tn
   no. of extra nodes :
   coord option :
   tn generated automatically
   Any changes? :

   c.p. name and type : b3 br
   no. of extra nodes : 1
   coord option : du
   Any changes? :

   c.p. name and type :
Last c.p. in this run? : y
Display GEOM data for this run? :

START RUN NO.  4
Enter c.p. data in sequence along run

   c.p. name and type :
Last run? : y

More changes? :

Display modified GEOM data? :
Write in session log? : y

MODIFIED GEOM DATA
```

run no.	c.p. name	c.p. type	bend radius	xtra nods	coord opt	c.p. i	c.p. j	coordinate data x,dx,p	y,dy	z,d
1										
	a1			0	di			0.	0.	0.
	t1a	tn		3	tn			0.	0.	0.
	ti1	ti	15.000	0	of	a1		20.000	0.	-40.00
	t1b	tn		1	tn			0.	0.	0.
	b1	br		0	of	ti1		40.000	0.	0.
	b2	br		1	of	b1		20.000	0.	0.
	t2a	tn		1	tn			0.	0.	0.
	ti2	ti	10.000	0	of	b2		30.000	0.	0.
	t2b	tn		0	tn			0.	0.	0.
	a2			0	of	ti2		0.	20.000	0.
2										
	b2	br		0	du			0.	0.	0.
	t4a	tn		0	tn			0.	0.	0.
	ti4	ti	5.000	0	of	b2		0.	-20.000	0.
	b3			1	of	a3		0.	-10.000	0.
	a3			0	of	ti4		0.	0.	40.00
3										
	b1	br		0	du			0.	0.	0.

```
    t3a     tn                  0     tn                    0.          0.          0.
    ti3     ti     10.000       0     of     b1             0.          0.          20.000
    t3b     tn                  0     tn                    0.          0.          0.
    b3      br                  1     du                    0.          0.          0.

Save current GEOM data? : y        SAVE FOR SAFETY IN CASE OF CRASH
Comment for file catalog :
GEOM DATA SAVED. FILE NAME = GEOM0102

Produce COOR data? : y

*** error - no following tn, ti = ti4

Modify GEOM data? : y        CORRECT ERROR
Specify modifications
Change data for a c.p.? : y
  c.p. name : t4b
*** error - no c.p. with this name - reenter
  c.p. name :
Change data for a c.p.? : no        CONTROL POINT t4b WAS OMITTED
Insert new c.p's? : y
  name of preceding c.p. : ti4
  enter data for new c.p's

  c.p. name and type : t4b tn
  no. of extra nodes :
  coord option :
  tn generated automatically
  Any changes? :

  c.p. name and type :

More changes? :

Display modified GEOM data? :
Write in session log? : y

MODIFIED GEOM DATA

run   c.p.   c.p.    bend    xtra   coord   c.p.   c.p.        coordinate  data
no.   name   type   radius   nods    opt     i      j      x,dx,p       y,dy        z,dz

 1
      a1                      0      di                    0.          0.          0.
      t1a     tn              3      tn                    0.          0.          0.
      ti1     ti     15.000   0      of      a1            20.000      0.          -40.000
      t1b     tn              1      tn                    0.          0.          0.
      b1      br              0      of      ti1           40.000      0.          0.
      b2      br              1      of      b1            20.000      0.          0.
      t2a     tn              1      tn                    0.          0.          0.
      ti2     ti     10.000   0      of      b2            30.000      0.          0.
      t2b     tn              0      tn                    0.          0.          0.
      a2                      0      of      ti2           0.          20.000      0.
 2
      b2      br              0      du                    0.          0.          0.
      t4a     tn              0      tn                    0.          0.          0.
```

```
     ti4     ti     5.000     0     of    b2              0.        -20.000       0.
     t4b     tn               0     tn                    0.          0.          0.
     b3                       1     of    a3              0.        -10.000       0.
     a3                       0     of    ti4             0.          0.         40.000
  3
     b1      br               0     du                    0.          0.          0.
     t3a     tn               0     tn                    0.          0.          0.
     ti3     ti    10.000     0     of    b1              0.          0.         20.000
     t3b     tn               0     tn                    0.          0.          0.
     b3      br               1     du                    0.          0.          0.

Save current GEOM data? : y        SAVE FOR SAFETY IN CASE OF CRASH
Comment for file catalog :
GEOM DATA SAVED. FILE NAME = GEOM0102

Produce COOR data? : y

Display COOR data? : y


COOR DATA

 run      c.p.   c.p.     bend    node        x           y           z
 no.      name   type    radius   no.       coord       coord       coord

  1
         a1                        1          0.          0.         0.
                                   2          3.964       0.        -7.927
                                   3          7.927       0.       -15.854
                                   4         11.891       0.       -23.781
         t1a     tn                5         15.854       0.       -31.708
         ti1     ti      15.000              20.000       0.       -40.000
                         center              29.271       0.       -25.000
                                   6         21.385       0.       -37.760
         t1b     tn                7         29.271       0.       -40.000
         b1      br                8         60.000       0.       -40.000
Hit RETURN for more
                                   9         70.000       0.       -40.000
         b2      br               10         80.000       0.       -40.000
                                  11         90.000       0.       -40.000
         t2a     tn               12        100.000       0.       -40.000
         ti2     ti      10.000             110.000       0.       -40.000
                         center             100.000      10.000    -40.000
         t2b     tn               13        110.000      10.000    -40.000
         a2                       14        110.000      20.000    -40.000
  2
         b2      br               10         80.000       0.       -40.000
         t4a     tn               15         80.000     -15.000    -40.000
Hit RETURN for more
         ti4     ti       5.000              80.000     -20.000    -40.000
                         center              80.000     -15.000    -35.000
         t4b     tn               16         80.000     -20.000    -35.000
                                  17         80.000     -25.000    -17.500
         b3                       18         80.000     -30.000      0.
         a3                       19         80.000     -20.000      0.
  3
```

```
        b1      br                      8       60.000      0.        -40.000
        t3a     tn                     20       60.000      0.        -25.888
        ti3     ti      10.000                  60.000      0.        -20.000
                        center                  65.547     -8.320     -25.888
Hit RE1URN for more
        t3b     tn                     21       62.856     -4.284     -17.144
                                       22       71.428    -17.142      -8.572
        b3      br                     18       80.000    -30.000       0.
END OF DATA

Plot geometry? : no        NO PLOTTER ON THIS TERMINAL . PLOT LATER

Modify GEOM data? : y      CORRECT DATA FOR b3
Specify modifications
Change data for a c.p.? : y
   c.p. name : b3
   enter revised data
   c.p. type : br
   no. of extra nodes : 1
   coord option :
   c.p., dx,dy,dz : a3 0 0 -10          PREVIOUSLY HAD 0 -10 0
   Any changes? :

More changes? :

Display modified GEOM data? :
Write in session log? : y

MODIFIED GEOM DATA
```

run no.	c.p. name	c.p. type	bend radius	xtra nods	coord opt	c.p. i	c.p. j	coordinate data x,dx,p	y,dy	z,dz
1										
	a1			0	di			0.	0.	0.
	t1a	tn		3	tn			0.	0.	0.
	ti1	ti	15.000	0	of	a1		20.000	0.	-40.000
	t1b	tn		1	tn			0.	0.	0.
	b1	br		0	of	ti1		40.000	0.	0.
	b2	br		1	of	b1		20.000	0.	0.
	t2a	tn		1	tn			0.	0.	0.
	ti?	ti	10.000	0	of	b2		30.000	0.	0.
	t2b	tn		0	tn			0.	0.	0.
	a2			0	of	ti2		0.	20.000	0.
2										
	b2	br		0	du			0.	0.	0.
	t4a	tn		0	tn			0.	0.	0.
	ti4	ti	5.000	0	of	b2		0.	-20.000	0.
	t4b	tn		0	tn			0.	0.	0.
	b3	br		1	of	a3		0.	0.	-10.000
	a3			0	of	ti4		0.	0.	40.000
3										
	b1	br		0	du			0.	0.	0.
	t3a	tn		0	tn			0.	0.	0.
	ti3	ti	10.000	0	of	b1		0.	0.	20.000
	t3b	tn		0	tn			0.	0.	0.

```
    b3     br                1     du                     0.          0.          0.

Save current GEOM data? : y
Comment for file catalog :
GEOM DATA SAVED. FILE NAME = GEOM0102

Produce COOR data? : y

Display COOR data? : y


COOR DATA

 run      c.p.   c.p.     bend    node        x          y          z
 no.      name   type    radius    no.      coord      coord      coord

  1
          a1                        1      0.          0.         0.
                                    2      3.964       0.        -7.927
                                    3      7.927       0.       -15.854
                                    4     11.891       0.       -23.781
          t1a    tn                 5     15.854       0.       -31.708
          ti1    ti      15.000           20.000       0.       -40.000
                         center           29.271       0.       -25.000
                                    6     21.385       0.       -37.760
          t1b    tn                 7     29.271       0.       -40.000
          b1     br                 8     60.000       0.       -40.000
Hit RETURN for more
                                    9     70.000       0.       -40.000
          b2     br                10     80.000       0.       -40.000
                                   11     90.000       0.       -40.000
          t2a    tn                12    100.000       0.       -40.000
          ti2    ti      10.000          110.000       0.       -40.000
                         center          100.000     10.000     -40.000
          t2b    tn                13    110.000     10.000     -40.000
          a2                       14    110.000     20.000     -40.000
  2
          b2     br                10     80.000       0.       -40.000
          t4a    tn                15     80.000    -15.000     -40.000
Hit RETURN for more
          ti4    ti       5.000           80.000    -20.000     -40.000
                         center           80.000    -15.000     -35.000
          t4b    tn                16     80.000    -20.000     -35.000
                                   17     80.000    -20.000     -22.500
          b3     br                18     80.000    -20.000     -10.000
          a3                       19     80.000    -20.000       0.
  3
          b1     br                 8     60.000       0.       -40.000
          t3a    tn                20     60.000       0.       -27.071
          ti3    ti      10.000           60.000       0.       -20.000
                         center           67.071     -7.071     -27.071
Hit RETURN for more
          t3b    tn                21     64.714     -4.714     -17.643
                                   22     72.357    -12.357     -13.821
          b3     br                18     80.000    -20.000     -10.000
END OF DATA
```

```
Plot geometry? :

Modify GEOM data? :

DATA COMPLFTE FOR THIS SESSION
Save final GEOM data? : y
Comment for file catalog : Example. Fig.A4.2.1. Full system.
GEOM DATA SAVED. FILE NAME = GEOM0102
Save COOR data? : y
Comment for file catalog : From GEOM0102
COOR DATA SAVED. FILE NAME = COOR0101

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : list

Problem 1: Illustrative Example, Fig. A4.2.1

file    sequence            date                          description
type     number
GEOM        1    Wed Dec  9 17:57:29 1981   Example. Fig.A4.2.1. First 2 runs.
GEOM        2    Wed Dec  9 18:27:02 1981   Example. Fig.A4.2.1. Full system.
COOR        1    Wed Dec  9 18:27:02 1981   From GEOM0102

NEXT WIPS EXEC COMMAND : delete        GEOM0101 NO LONGER NEEDED
File type: GEOM
Sequence number: 1
GEOM0101 deleted
File type:

NEXT WIPS-EXEC COMMAND : quit

-- -- --- - ----- --- ----- --------------- -- -------------------------
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : geom        WIPS-GEOM TO PLOT GEOMETRY ONLY

GEOM - SPECIFICATION OF SYSTEM GEOMETRY

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Modify existing GEOM file? : y
Sequence no. : 2

Display existing GEOM data? :
Write in session log? :

Modify GEOM data? :

Produce COOR data? : y              MUST PRODUCE COORDINATES TO PLOT

Display COOR data? :
Write in session log? :
```

```
Plot geometry? : y
Plot complete system? : y            COMPLETE SYSTEM. FIG.A4.2.2

Plot device (dflt=4662) :
Baud rate (dflt=1200) :

Viewing direction (+x,-x,+y,-y,+z,-z or incl) : incl
Direction cosines (3 values) : -1 -1 -1
Vertical axis in plot (dflt=+y) :
Max. plot dimensions (current length units)
   Width  =  0.1061e+03
   Height =  0.5591e+02
Specify drawing size and border widths
   Max. allowable width and height =  15.000,   10.000
   Width and height (incl. borders) : 11 8.5        8.5 * 11 PAPER
   Left and right border widths : 2 2
   Top and bottom border widths : 2 2
Scale (current length units per plot unit) =  0.1515e+02
Any changes ? :

Add text to drawing? : y
Enter text (max.60 characters)
 : FIG.A4.2.2  GEOMETRY PLOT. COMPLETE SYSTEM
Symbol height (inches) : .15
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : Run 1
Symbol height (dflt=no change) : .1
Horizontal lettering? : n
Specify angle (degrees) : -30
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : Run 2
Symbol height (dflt=no change) :
Horizontal lettering? : n
Specify angle (degrees) : 30
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : Run 3
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? :
Add X,Y,Z axis symbol? : y
Axis length (dflt=0.25in) :
Symbol location (bl,tl,br or tr) : bl
```

```
More plots? : y
New parts of system may be specified, or current
parts redrawn with new view, drawing size, etc.
Specify new parts? : y
Plot complete system? : n

SPECIFY PARTS OF SYSTEM TO BE PLOTTED
PART  1
   Run number : 1
   First c.p.  (dflt=start of run) : b1
   Last c.p.   (dflt=end of run)   :
More parts? : y
PART  2
   Run number : 2
   First c.p.  (dflt=start of run) :
   Last c.p.   (dflt=end of run)   : b3
More parts? :

Plot device (dflt=no change) :

Viewing direction (+x,-x,+y,-y,+z,-z or incl) : incl
Direction cosines (3 values) : 1 -1 -1
Vertical axis in plot (dflt=+y) :
Max. plot dimensions (current length units)
   Width  =  0.3536e+02
   Height =  0.5715e+02
Change drawing size or borders? :
Scale (current length units per plot unit) =  0.1270e+02
Any changes ? :

Add text to drawing? : y
Enter text (max.60 characters)
 : FIG. A4.2.3  GEOMETRY PLOT. PART OF SYSTEM
Symbol height (inches) : .15
Horizontal lettering? : y
Position pen, then hit RETURN

More text? :
Add X,Y,Z axis symbol? :
More plots? :

Modify GEOM data? :

DATA COMPLETE FOR THIS SESSION
GEOM data not changed
Save COOR data? :

End of this GEOM session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

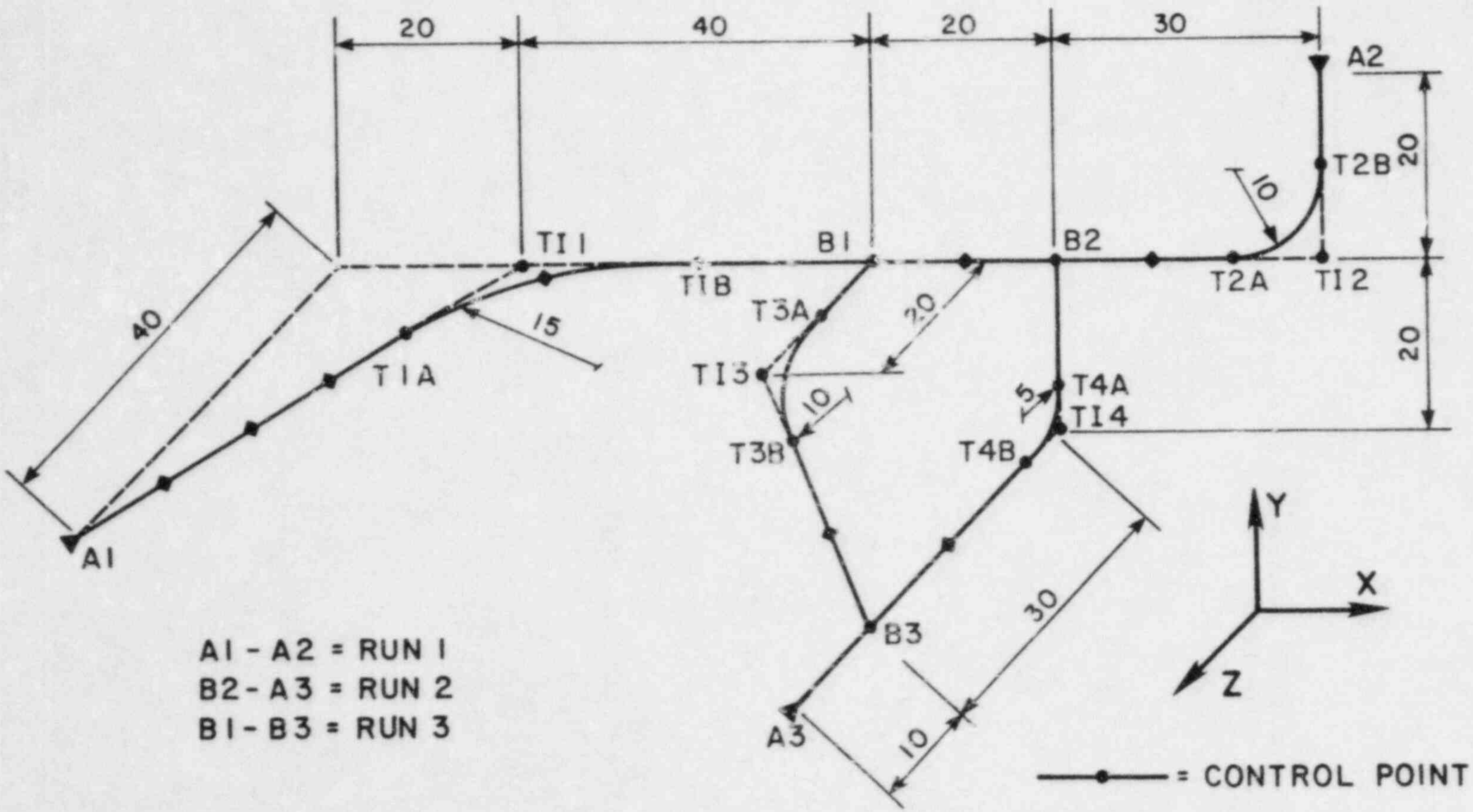


FIG. A4.2.1 - ILLUSTRATIVE EXAMPLE

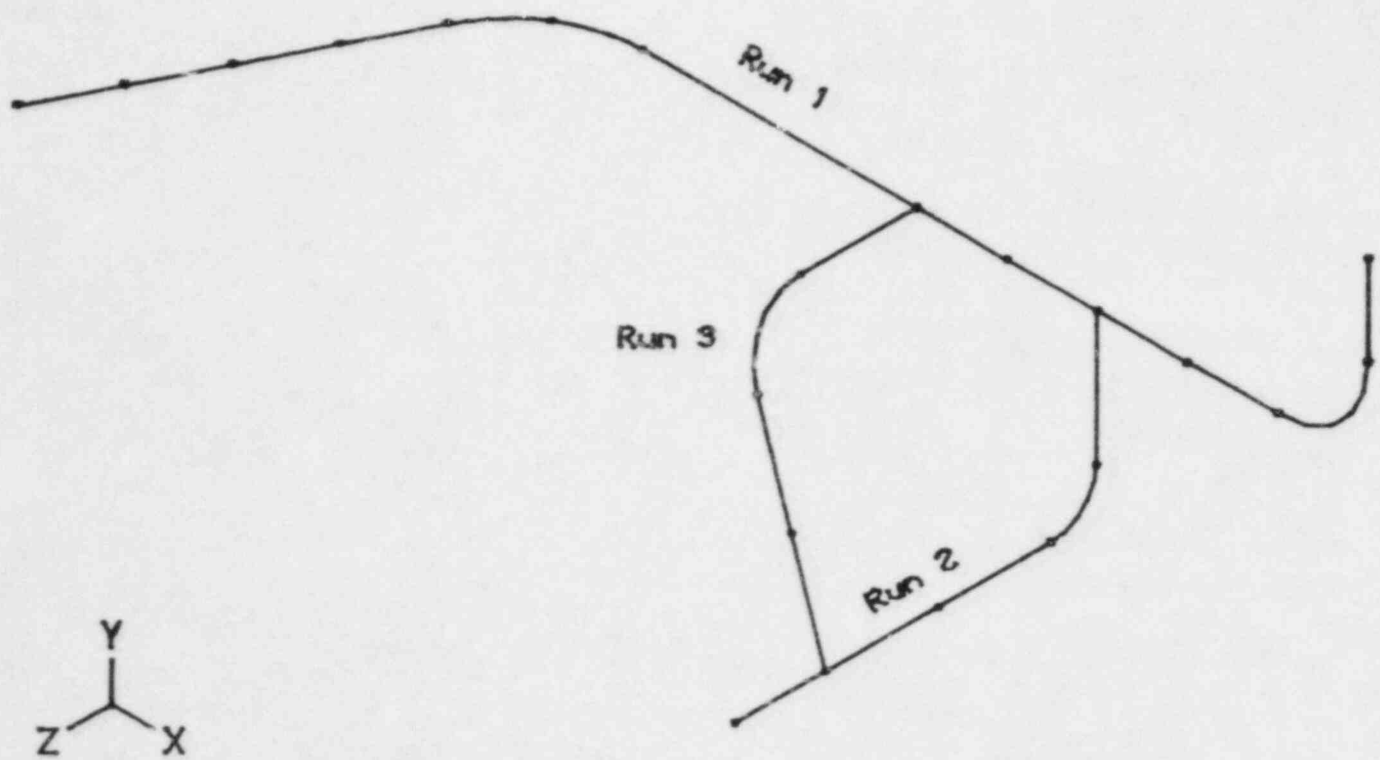


FIG. A4.2.2 GEOMETRY PLOT. COMPLETE SYSTEM

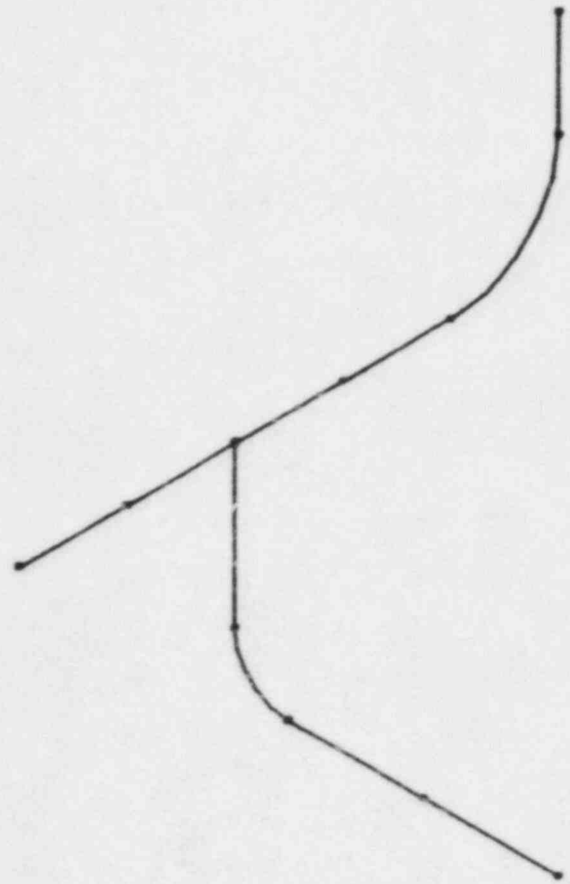


FIG. A4.2.3 GEOMETRY PLOT. PART OF SYSTEM

A5. WIPS-MATL AND WIPS-FREC USER GUIDES

SUMMARY

This section presents instructions and examples on the use of the WIPS-MATL and WIPS-FREC modules.

CONTENTS

A5.1 WIPS-MATL USER GUIDE

A5.1.1 PURPOSE

WIPS-MATL accepts "sets" of material property data and stores them in the MATL file. The materials can then be used for PIPE elements (in WIPS-PIPE) or for shell elements (in WIPS-ELBO, WIPS-STRP, and WIPS-SLAB). Only one material model can currently be specified, namely the Mroz model for steel.

Material property sets may be specified in a single WIPS-MATL session or in a series of separate sessions. New property sets are added at the end of the MATL file. Property sets are numbered in the order in which they are specified.

A5.1.2 MROZ MATERIAL MODEL

A5.1.2.1 General

The theory of the Mroz material model is presented in Section B2. A typical WIPS user does not need to be concerned with the details of the theory. The input data for WIPS-MATL will typically consist only of stress-strain and stress-strain-rate relationships for the material in uniaxial tension. The theory also requires that Poisson's ratio and the material density be defined and that tolerances be specified for implementation of the numerical scheme. Typically, however, the default values will apply for these quantities.

A5.1.2.2 Stress vs. Strain

The stress-strain relationship must be a multilinear curve (Fig. A5.1.1). The relationship may have up to 5 segments for materials to be used for STRP, ELBO, and SLAB substructures, and a maximum of 3 segments for materials to be used for PIPE elements. The static moduli E_1, E_2, etc. and the yield strengths S_1, S_2, etc. must be input.

A5.1.2.3 Stress vs. Strain Rate

If strain rate effects are specified, a multilinear relationship must be defined between stress *increase* (strength in excess of static strength) and strain rate (Fig. A5.1.2). The relationship may have up to 3 segments. The strain rate stiffnesses E_{d1}, E_{d2}, E_{d3} and the strain rate limits $\dot{\epsilon}_1$, $\dot{\epsilon}_2$ must be specified.

A5.1.2.4 Numerical Tolerances

Three tolerances to control the numerical computation may be specified if desired. These tolerances control the determination of nonlinear "events" and thus affect the accuracy and the amount of computer time. A large tolerance may save computation effort but may adversely affect the accuracy or even lead to numerical instability. The default values are recommended.

The tolerances define allowable "overshoots" before a nonlinearity is classified as an event. Tolerances may be specified for yield, stiffness reformulation, and strain rate effects. The yield tolerance defines the allowable overshoot of a yield event as a proportion of the yield stress, with a default of 0.02. The stiffness tolerance defines the allowable change in plastic flow direction before the material stiffness is assumed to be significantly affected, with a default of 0.05 radians. The strain rate tolerance defines the allowable overshoot at a change in strain rate stiffness, as a proportion of the strain rate. The default value is 0.02.

A5.1.2.5 Other Data

Poisson's ratio (default = 0.3) and the material weight density (default = steel = 0.284 lb/cu. in.) may be specified if desired.

A5.1.3 EXAMPLE

Table A5.1.1 is a listing of a WIPS-MATL session log. It illustrates the procedure for materials with properties similar to those shown in Fig. A5.1.1. Two materials are first defined, one with and one without strain rate effects. Control is then returned to WIPS-EXEC, and WIPS-MATL is called again to add a third material. Help is requested and deliberate errors are made to illustrate the help and error messages. For purposes of illustration, the units are changed in the second call to WIPS-MATL.

TABLE A5.1.1

WIPS-MATL EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : matl


MATL - SPECIFICATION OF MATERIAL PROPERTIES

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Start new MATL file

Specify a new property set? : help
Sorry, no help on this item

Specify a new property set? : y

SET NO. =  1
Property set description : help
Optional description, max. 40 characters
Property set description : ASTM A106 Grade B. Trilinear.
  Static moduli (min=2,max=5)
   : help
Slopes of stress-strain curve. Specify as many
as needed, on one line. First = elastic modulus
  Static moduli (min=2,max=5)
   : 3.e5 1.715e6 120
*** error - must decrease progressively
  Static moduli (min=2,max=5)
   : 30000 1715 120
  Yield strengths (no. of moduli minus 1)
   : 45 41
*** error - must increase progressively
  Yield strengths (no. of moduli minus 1)
   : 45 51
  Strain rate stiffnesses (min=0,max=3)
   : help
Slopes of curve relating stress increase to
strain rate. As many as needed, on one line
  Strain rate stiffnesses (min=0,max=3)
   : 1.5 .0522 .01125
  Strain rate limits (no. of stiffnesses minus 1)
   : help
Strain rates at which stiffnesses change
  Strain rate limits (no. of stiffnesses minus 1)
   : 20 250
```

```
  Use default tolerances? : y
  Poisson ratio (dflt = .3) :
  Weight density (dflt=steel) :

Any errors? :

This set added to MATL file

Specify a new property set? : y

SET NO. =  2
Property set description : ASTM A312 Type 304L. Trilinear.
  Static moduli (min=2,max=5)
   : 30000 1625 212
  Yield strengths (no. of moduli minus 1)
   : 32 45
  Strain rate stiffnesses (min=0,max=3)
   :
  No strain rate effect
  Use default tolerances? : n
  Yield tolerance (dflt=.02) : .04
  Stiffness tolerance (dflt=.05) :
  Poisson ratio (dflt = .3) :
  Weight density (dflt=steel) :

Any errors? :

This set added to MATL file

Specify a new property set? :

No. of property sets in MATL file =  2

Display property set descriptions? : y

MATL PROPERTY DESCRIPTIONS

  Set No.  Type  Description
     1     mroz  ASTM A106 Grade B. Trilinear.
     2     mroz  ASTM A312 Type 304L. Trilinear.

Display new property set data? : y

MATL PROPERTY DATA

SET NO.  1.  ASTM A106 Grade B. Trilinear.
```

Matl Type	Data Type	Segm No.	Modulus or Coefficient	Stress/Strain Limit
mroz				
	static	1	0.3000e+05	0.4500e+02
		2	0.1715e+04	0.5100e+02
		3	0.1200e+03	
	str. rate	1	0.1500e+01	0.2000e+02
		2	0.5220e-01	0.2500e+03
		3	0.1125e-01	

```
               yld. tol.            0. 2000e-01
               stif tol             0. 5000e-01
               rate tol             0. 5000e-01
               poisson              0. 3000e+00
               density              0. 2840e-03
Hit RETURN for next set

SET NO.  2.  ASTM A312 Type 304L. Trilinear.

        Matl     Data    Segm     Modulus  or     Stress/Strain
        Type     Type    No.      Coefficient         Limit
        mroz
               static     1       0. 3000e+05       0. 3200e+02
                          2       0. 1625e+04       0. 4500e+02
                          3       0. 2120e+03
               yld. tol.          0. 4000e-01
               stif tol           0. 5000e-01
               rate tol           0.      e+00
               poisson            0. 3000e+00
               density            0. 2840e-03
Hit RETURN for next set
END OF DATA

Display all property set data? : no         ALREADY DISPLAYED
Write in session log? :

New MATL file created
Comment for file catalog : Example, Section A5. 1

End this MATL session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : matl         RETURN TO ADD A THIRD MATERIAL


MATL - SPECIFICATION OF MATERIAL PROPERTIES

Define units
  Length (ft, in, m, mm) : mm              NOTE S. I. UNITS
  Force (k, lb, kgf, kN) : kN

Existing MATL file being extended.
No. of existing property sets =  2

Specify a new property set? : y

SET NO. =  3
Property set description : Elastic-Plastic, Fy = 280 MPa.
  Static moduli (min=2, max=5)
   : 206. 85 2. 06
  Yield strengths (no. of moduli minus 1)
   : . 28
  Strain rate stiffnesses (min=0, max=3)
   :
  No strain rate effect
```

```
  Use default tolerances? : y
  Poisson ratio (dflt = .3) :
  Weight density (dflt=steel) :

Any errors? :

This set added to MATL file

Specify a new property set? :

No. of property sets in MATL file =  3

Display property set descriptions? : y

MATL PROPERTY DESCRIPTIONS

  Set No.  Type  Description
     1     mroz  ASTM A106 Grade B. Trilinear.
     2     mroz  ASTM A312 Type 304L. Trilinear.
     3     mroz  Elastic-Plastic, Fy = 280 MPa

Display new property set data? : y

MATL PROPERTY DATA

SET NO.  3.  Elastic-Plastic, Fy = 280 MPa.

       Matl     Data     Segm     Modulus  or     Stress/Strain
       Type     Type     No.      Coefficient        Limit
       mroz
              static      1       0.2069e+03       0.2800e+00
                          2       0.2060e+01
              yld.tol.            0.2000e-01
              stif tol            0.5000e-01
              rate tol            0.     e+00
              poisson             0.3000e+00
              density             0.7709e-07
Hit RETURN for next set
END OF DATA

Display all property set data? : n
Write in session log? : n

Existing MATL file extended
Comment for file catalog : Example, Section A5.1

End this MATL session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : matl          DISPLAY PROPERTIES ONLY


MATL - SPECIFICATION OF MATERIAL PROPERTIES

Define units
```

Length (ft, in, m, mm) : in NOTE KIP, INCH UNITS
Force (k, lb, kgf, kN) : k

Existing MATL file being extended.
No. of existing property sets = 3

Specify a new property set? :

No. of property sets in MATL file = 3

Display property set descriptions? : y

MATL PROPERTY DESCRIPTIONS

Set No.	Type	Description
1	mroz	ASTM A106 Grade B. Trilinear.
2	mroz	ASTM A312 Type 304L. Trilinear.
3	mroz	Elastic-Plastic, Fy = 280 MPa.

Display all property set data? : y

MATL PROPERTY DATA

SET NO. 1. ASTM A106 Grade B. Trilinear.

Matl Type	Data Type	Segm No.	Modulus or Coefficient	Stress/Strain Limit
mroz				
	static	1	0. 3000e+05	0. 4500e+02
		2	0. 1715e+04	0. 5100e+02
		3	0. 1200e+03	
	str. rate	1	0. 1500e+01	0. 2000e+02
		2	0. 5220e-01	0. 2500e+03
		3	0. 1125e-01	
	yld. tol.		0. 2000e-01	
	stif tol		0. 5000e-01	
	rate tol		0. 5000e-01	
	poisson		0. 3000e+00	
	density		0. 2840e-03	

Hit RETURN for next set

SET NO. 2. ASTM A312 Type 304L. Trilinear.

Matl Type	Data Type	Segm No.	Modulus or Coefficient	Stress/Strain Limit
mroz				
	static	1	0. 3000e+05	0. 3200e+02
		2	0. 1625e+04	0. 4500e+02
		3	0. 2120e+03	
	yld. tol.		0. 4000e-01	
	stif tol		0. 5000e-01	
	rate tol		0. e+00	
	poisson		0. 3000e+00	
	density		0. 2840e-03	

Hit RETURN for next set

```
SET NO.  3.  Elastic-Plastic, Fy = 280 MPa.

       Matl     Data     Segm      Modulus  or      Stress/Strain
       Type     Type     No.       Coefficient          Limit
       mroz
             static       1        0.3000e+05        0.4061e+02
                          2        0.2988e+03
             yld.tol.              0.2000e-01
             stif tol              0.5000e-01
             rate tol              0.     e+00
             poisson               0.3000e+00
             density               0.2840e-03
Hit RETURN for next set
END OF DATA

MATL file not changed

End this MATL session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : quit
```

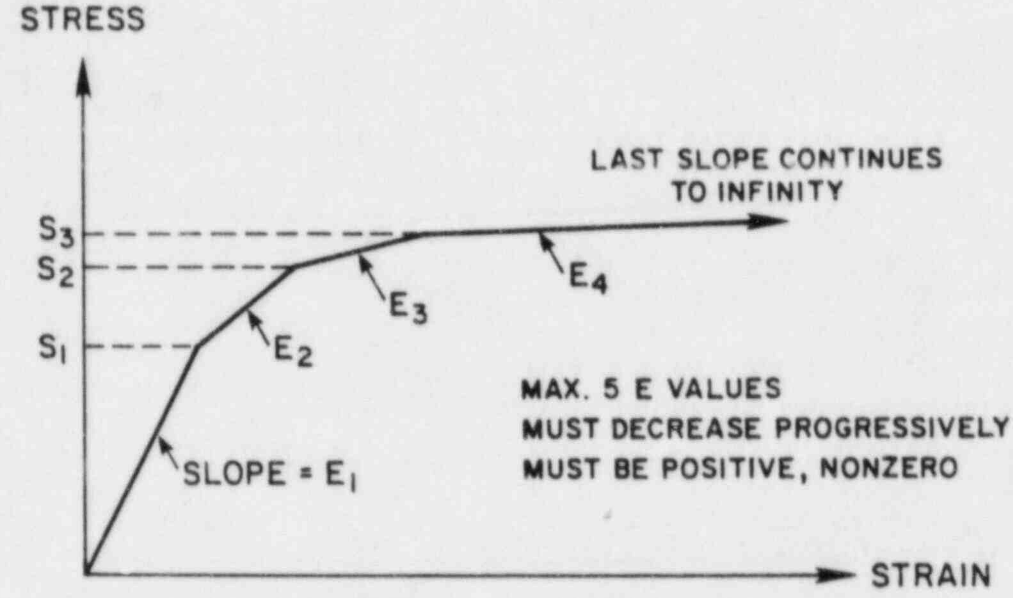


FIG. A5.1.1 - STRESS-STRAIN RELATIONSHIP

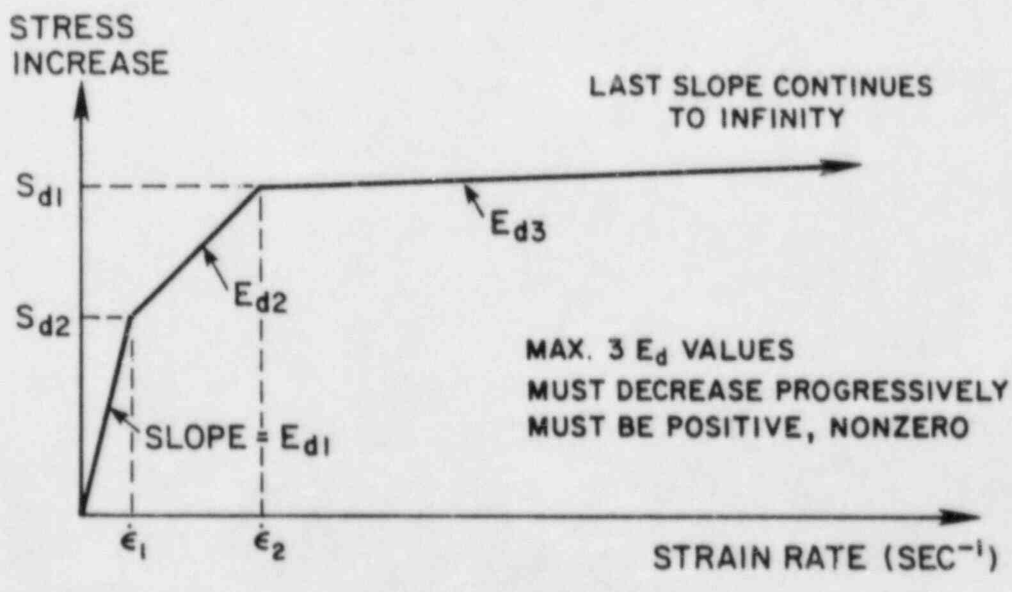


FIG. A5.1.2 - RELATIONSHIP BETWEEN STRESS INCREASE AND STRAIN RATE

A5.2 WIPS-FREC USER GUIDE

A5.2.1 PURPOSE

WIPS-FREC accepts records defining variation of force with time and stores them in the FREC file. Particular records can then be incorporated into a DATA file in the WIPS-DATA phase. Only force records are set up in WIPS-FREC: the loaded points and the force directions are specified in WIPS-DATA. The force magnitudes may also be scaled in WIPS-DATA, if desired.

The force records may be specified in a single WIPS-FREC session or in a series of separate sessions. New records are added at the end of the FREC file. Each record is identified by a four-character name specified by the WIPS-FREC user. In addition, the records are numbered in the order in which they are specified.

A5.2.2 FORCE RECORD

Each force record is specified as a series of time-force pairs. The times (in seconds) must increase progressively. The first time-force pair is automatically set to zero-zero. Data input begins with the second pair.

A5.2.3 EXAMPLE

Table A5.2.1 is a listing of a WIPS-FREC session log to illustrate the procedure for specifying FREC data.

TABLE A5.2.1

WIPS-FREC EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : frec

FREC - SPECIFY DYNAMIC FORCE RECORDS

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Start new FREC file

Specify a new record ? : y

RECORD NO.  1

Record name (4 characters) : rec1
Description (max. 40 char.) : Constant 15k force.

Enter Time-Force pairs
First pair automatically set to 0,0
  Pair no.  2 : help
Enter time-force pairs one per line
Enter blank line to end record
  Pair no.  2 : 1.e-6 15
  Pair no.  3 : 1. 15
  Pair no.  4 :
Last pair? : y
Any errors ? :

Specify a new record ? : y

RECORD NO.  2

Record name (4 characters) : rec2
Description (max. 40 char.) : Variable force, 20k peak.

Enter Time-Force pairs
First pair automatically set to 0,0
  Pair no.  2 : 1.e-6 15
  Pair no.  3 : .004 20
  Pair no.  4 : .01 10
  Pair no.  5 : 1. 10
  Pair no.  6 :
Last pair? : y
Any errors ? :
```

```
Specify a new rec    . :

Display any records ? : y
Record number (dflt=all) :

RECORD NO.  1.  Name = rec1
     Time           Force
   0.          0.      e+00
   0.00000     0.1500e+02
   1.00000     0.1500e+02
END OF RECORD
Hit RETURN for next record

RECORD NO.  2.  Name = rec2
     Time           Force
   0.          0.      e+00
   0.00000     0.1500e+02
   0.00400     0.2000e+02
   0.01000     0.1000e+02
   1.00000     0.1000e+02
END OF RECORD

Write records in session log? : no      DISPLAY AUTOMATICALLY WRITES TO LOG

End this FREC session? : y
New FREC file created
Comment for file catalog : Example, Section A5.2

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : frec             RETURN TO LOOK AT A RECORD

FREC - SPECIFY DYNAMIC FORCE RECORDS

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Extend existing FREC file
Data on existing file
  Rec.  Rec.  No.of
   No.  Name  Times  Description
    1   rec1      3  Constant 15k force.
    2   rec2      5  Variable force, 20k peak.

Specify a new record ? :

Display any records ? : y
Record number (dflt=all) : 2

RECORD NO.  2.  Name = rec2
     Time           Force
   0.          0.      e+00
   0.00000     0.1500e+02
   0.00400     0.2000e+02
   0.01000     0.1000e+02
```

```
   1.00000    0.1000e+02
END OF RECORD

Display any records ? :

Write records in session log? :

End this FREC session? : y
FREC file not changed

EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : quit
```

A6. ELEMENT TYPE USER GUIDES

SUMMARY

This section contains user guides for defining the properties for each different element type. Four element types may be defined, namely straight or curved pipes (WIPS-PIPE, Section A6.1), straight beam-columns (WIPS-BEAM, Section A6.2), U-bar restraints (WIPS-UBAR, Section A6.3), and gap-friction elements (WIPS-GAPF, Section A6.4).

CONTENTS

A6.1 WIPS-PIPE USER GUIDE

A6.1.1 PURPOSE

WIPS-PIPE accepts "sets" of property data for pipe elements and stores them in the PIPE file. Pipe elements with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by defining the location in a pipe run and the property set to be used. WIPS-MODL extracts the appropriate data from the PIPE file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-PIPE session or in a series of separate sessions. New property sets are added at the end of the PIPE file. Property sets are numbered in the order they are specified.

A6.1.2 INPUT DATA

A6.1.2.1 General

The theory of the PIPE element is presented in Section B3. A typical WIPS user should be familiar with the main features of the element but need not be concerned with the details of the theory.

A6.1.2.2 Basic Properties

The basic properties which must be defined are the outside diameter, the wall thickness, the number of subelements in the pipe cross section (typically 8, maximum 12), the weight per unit length (default = steel pipe), and the material number (in the MATL file). The material must have been previously defined, using WIPS-MATL and must have *no more than three linear segments* in its stress-strain relationship.

A6.1.2.3 Ovalling Properties

To enable curved elements to be considered, additional information on the ovalling properties is needed. Typically, the default values will apply. The PIPE element accounts for ovalling using an approximate theory, as explained in Section B3. The contribution of the pipe wall bending stiffness to the ovalling resistance is defined by means of a multi-linear relationship between ovalling deformation and a generalized ovalling force (Fig. A6.1.1). The generalized ovalling force is defined only in a virtual work sense. The stiffness K_1 and the strength Ω_1 in Fig. A6.1.1 are calculated by WIPS-PIPE from the pipe diameter, the wall thickness, and the material properties. The stiffnesses h_1K_1 and h_2K_1 may be assigned by WIPS-PIPE (default option), or they may be input by specifying the values of h_1 and h_2. The default values are h_1=0.4 and h_2=0.05. The value Ω_2 is set equal to the strength when the ovalling, ω, equals 4 times the initial yield ovalling, ω_y (Fig. A6.1.1).

If desired, the values of K_1 and Ω_1 may be modified, by specifying a stiffness factor, f_1, and a strength factor, f_2. The value of K_1 is then increased to f_1K_1, and the value of Ω_1 to $f_1f_2\Omega_1$ (i.e., $f_2f_1K_1\omega_y$, where ω_y remains constant). The value of Ω_2 is automatically modified to keep the ratio Ω_2/Ω_1 unchanged. The default values of f_1 and f_2 are both unity. Values larger than unity might be specified to account for the stiffening and strengthening effects exerted on elbow ovalling by adjacent straight pipes. It is generally recommended that the default values be used.

A6.1.2.4 Large Ovalling Deformations

If substantial ovalling occurs, the shape of the cross section may depart substantially from a circle. As a result, the in-plane flexural strength of an elbow for bending which collapses the cross section can be significantly less than for bending which stretches the cross section. This effect may be taken into account (large ovalling = "YES") or ignored (large ovalling = "NO").

The default is "YES".

A6.1.2.5 Large Displacements

Analyses may be carried out by WIPS-ANAL for either small displacements (typically restrained pipes) or large displacements (typically unrestrained pipes). The choice is made in the WIPS-MODL phase.

A6.1.3 EXAMPLE

Table A6.1.1 is a listing of a session log to illustrate the procedure for specifying PIPE properties.

TABLE A6.1.1

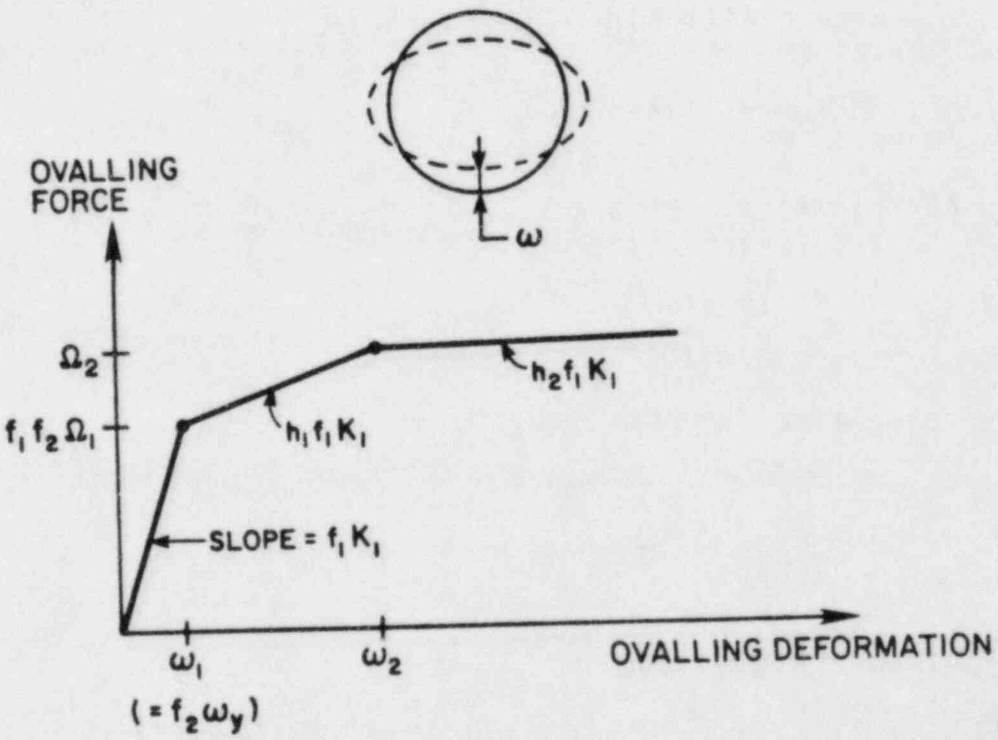
WIPS-PIPE EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : pipe


PIPE - SPECIFICATION OF PIPE PROPERTIES

Define units
  Length (ft/in/m/mm) : in
  Force (k/lb/kgf/kN) : k

No. of MATL property sets available =  3
Display material descriptions? : y

MATL PROPERTY DESCRIPTIONS
  Set No.  Type  Description
     1     mroz  ASTM A106 Grade B. Trilinear.
     2     mroz  ASTM A312 Type 304L. Trilinear.
     3     mroz  Elastic-Plastic, Fy = 280 MPa.

Start new PIPE file

Specify a new property set? : y

SET NO  =  1
Property set description : help
Optional description, max. 40 characters
Property set description : 6sch40, A106-B, with strain rate.
  Outside diameter : 6.625
  Wall thickness : .28
  Weight/unit length (dflt=pipe weight) :
  No. of X-section elements (dflt=12) :
  Material number : 1
  Large ovalling (yes or no)? (dflt=no) :
Use default ovalling properties? : y

Any errors? :

This set added to PIPE file

Specify a new property set? : y

SET NO. =  2
Property set description : 4sch40, A312-304L, no strain rate.
  Outside diameter : 4.5
  Wall thickness : .237
  Weight/unit length (dflt=pipe weight) :
```

```
   No. of X-section elements (dflt=12) : 8
   Material number : 2
   Large ovalling (yes or no)? (dflt=no) :
 Use default ovalling properties? : n
   No. of hardening ratios for ovalling (dflt=2) :
   Hardening ratios for ovalling : .38 .14
   Stiffness modification factor :
   Strength modification factor :

 Any errors? :

 This set added to PIPE file

 Specify a new property set? :

 No. of property sets in PIPE file =  2

 Display property set descriptions? : y

 PIPE PROPERTY DESCRIPTIONS
   Set No.   Description
      1      6sch40, A106-B, with strain rate.
      2      4sch40, A312-304L, no strain rate.

 Display new property set data? :
 Write in session log? :

 Display all property set data? : y

  PIPE PROPERTY DATA

 SET NO.  1.  6sch40, A106-B, with strain rate.

       Data                      Segm    Modulus or     Stress/Strain
       Type                      No.     Data Value         Limit

       Outside diameter                  0.6625e+01
       Wall thickness                    0.2800e+00
       Unit weight                       0.1585e-02
       No. of elements                     12

       Stress v strain            1      0.3000e+05     0.4500e+02
                                  2      0.1715e+04     0.5100e+02
                                  3      0.1200e+03
       Stress v strain rate       1      0.1500e+01     0.2000e+02
                                  2      0.5220e-01     0.2500e+03
                                  2      0.5220e-01
       Poisson ratio                     0.3000e+00
       Large ovalling                      no
       Ovalling ratios            1      0.4000e+00
                                  2      0.5000e-01
 Hit RETURN for next set

 SET NO.  2.  4sch40, A312-304L, no strain rate.

       Data                      Segm    Modulus or     Stress/Strain
```

Table A6.1.1 (page 3)

```
     Type                         No.     Data Value          Limit

     Outside diameter                     0.4500e+01
     Wall thickness                       0.2370e+00
     Unit weight                          0.9014e-03
     No. of elements                          8
     No. of slices                            2
     Stress v strain               1      0.3000e+05       0.3200e+02
                                   2      0.1625e+04       0.4500e+02
                                   3      0.2120e+03
     Poisson ratio                        0.3000e+00
     Large ovalling                           no
     Ovalling ratios               1      0.3800e+00
                                   2      0.1400e+00
Hit RETURN for next set
END OF DATA

New PIPE file created
Comment for file catalog : Example, Section A6.1

End this PIPE session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : quit

--------------------------------------------------------------------------------
```



NOTES:

(1) K_1, Ω_1, Ω_2 and ω_y are determined automatically.

(2) h_1, h_2, f_1 and f_2 may be specified if desired, but default values will typically apply.

FIG. A6.1.1 - OVALLING BEHAVIOR

A6.2 WIPS-BEAM USER GUIDE

A6.2.1 PURPOSE

WIPS-BEAM accepts "sets" of property data for beam elements and stores them in the BEAM file. Beam elements with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by defining the location in a pipe run and the property set to be used. WIPS-MODL extracts the appropriate data from the BEAM file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-BEAM session or in a series of separate sessions. New property sets are added at the end of the BEAM file. Property sets are numbered in the order they are specified.

A6.2.2 BEAM PROPERTIES

A6.2.2.1 General

The theory of the BEAM element is presented in Section B4. A typical WIPS user should be familiar with the main features of the element but need not be concerned with the details of the theory.

BEAM elements may be used to model members of any cross section, not necessarily pipes. The basic element properties are specified as four action-deformation relationships, namely, (1) axial force vs. axial strain; (2) torque vs. rate of twist; (3) bending moment about one principal axis vs. corresponding curvature; and (4) moment vs. curvature about second principal axis. A different curve (with some restrictions) may be specified for each of the four actions. Shear deformations may be included, by specifying shear rigidities in the two bending planes.

Strain rate effects may be specified, by means of a single multi-linear curve (dimensionless) which applies for all four actions and deformations. In addition, the weight of the element per unit length must be specified, and tolerances to control the numerical accuracy may be defined. Typically, the default values will apply for these tolerances.

A6.2.2.2 Static Behavior

Multi-linear approximations of the action-deformation relationships must be chosen, as shown in Fig. A6.2.1. These relationships must be calculated separately, taking into account the cross section dimensions and material properties. This may involve a substantial amount of preliminary calculation. BEAM members are all assumed to be straight, and there is no allowance for cross-section distortion (i.e. ovalling). If BEAM elements are to be used to model bends or elbows, the action-deformation relationships must allow for the effects of cross-section distortion.

The sequence in which WIPS-BEAM requests the action-deformation data is as follows.

(1) Elastic axial stiffness (EA) and shear stiffnesses (GA_y in element xy plane; GA_z in element xz plane). Because the element is assumed to remain elastic in shear, these are the only shear properties required.

(2) Elastic torsional stiffness (GJ) and bending stiffnesses (EI_y for bending about y axis; EI_z for bending about z axis).

(3) First yield values F_1, MX_1, MY_1 and MZ_1 (Fig. A6.2.1).

(4) Stiffnesses EA_1, GJ_1, EIY_1, and EIZ_1 (Fig. A6.2.1).

(5) Second yield values F_2, MX_2, MY_2, and MZ_2.

(7) Third yield values F_3, MX_3, MY_3, and MZ_3.

(8) Stiffnesses EA_3, GJ_3, EIY_3, and EIZ_3.

A6.2.2.3 Restriction on Action-Deformation Relationships

As shown in Fig. A6.2.1, there are three changes of slope for each action-deformation relationship. The deformations at which slope changes occur are w_{F1}, w_{F2}, w_{F3} for axial force; w_{M1}, w_{M2}, w_{M3} for the bending moments; and w_{T1}, w_{T2}, w_{T3} for torque. Experience has shown that it is desirable to make the ratio w_1:w_2:w_3 essentially the same for all four relationships. This restriction is needed to obtain realistic hardening behavior if there is substantial axial or torsional yield. It is less important if yielding is primarily in bending.

WIPS-BEAM checks the w_1:w_2:w_3 ratios and displays a warning if they are not substantially similar.

A6.2.2.4 Strain Rate Effects

Strain rate effects, if any, must be defined by means of a dimensionless relationship between strength increase and deformation rate. The usual procedure for establishing this relationship will be as follows.

(1) For the material of which the element is made, construct a multi-linear curve (maximum 3 segments) relating stress *increase* (stress in excess of static stress) to strain rate (Fig. A6.2.2a).

(2) Select a yield stress, σ_y, and a corresponding yield strain, $\epsilon_y = \sigma_y/E$, where E = Young's modulus. The yield stress may be a nominal value (e.g. 0.2% offset).

(3) Divide the stress ordinates of the curve by σ_y and the strain rate ordinates by ϵ_y to get a dimensionless relationship (Fig. A6.2.2b).

The values X_1, X_2, and X_3 in Fig. A6.2.2b are "strain rate factors", and the quantities Y_1, Y_2, and Y_3 are "strength factors". WIPS-BEAM requests these factors.

A6.2.2.5 Tolerances

In the dynamic analysis, tolerances are required to control the stiffness reformulation when nonlinear "events" occur (e.g. yield or unloading from a yielded state). The following tolerances are used.

(1) An angle tolerance (in radians) to control stiffness reformulation as the direction of plastic flow changes. The default is 0.05.

(2) Overshoot (for yield) and reversal (for unloading) tolerances. These are dimensionless values, expressed as proportions of the element strength. The default values are 0.02 (i.e. 2%).

(3) A strain rate tolerance for change in the viscous stiffness associated with strain rate effects. This is a dimensionless value expressed as a proportion of strain rate factor. The default value is 0.02.

A6.2.3 EXAMPLE

Table A6.2.1 is a listing of a session log to illustrate the procedure for specifying BEAM properties.

TABLE A6.2.1

WIPS-BEAM EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : beam


BEAM - SPECIFICATION OF BEAM PROPERTIES

Define units
  Length (ft; in; m; mm) : in
  Force (k, lb, kgf, kN) : k

Start new BEAM file

Specify a new property set? : y

SET NO. =  1
Property set description : 24in. sch.100, with strain rate.
Weight per unit length : .0306

Elastic stiffnesses
  EAx, GAy, GAz : 2.886e6 1.e10 1.e10
  GJx, EIy, EIz : 1.402e8 1.822e8 1.822e8
Strengths at first yield
  Fx, Mxx, Myy, Mzz : 2886 15200 15200 15200
Stiffnesses after first yield
  EA, GJx, EIy, EIz : 23670 2.08e6 2.4e6 2.4e6
Strengths at second yield
  Fx, Mxx, Myy, Mzz : 4755 32700 32700 32700
Stiffnesses after second yield
  EA, GJx, EIy, EIz : 23.7 3.6e5 4.7e5 4.7e5
Strengths at third yield
  Fx, Mxx, Myy, Mzz : 4757 34000 34000 3400
*** error - must exceed previous values
Strengths at third yield
  Fx, Mxx, Myy, Mzz : 4757 34000 34000 34000
Stiffnesses after third yield
  EA, GJx, EIy, EIz : 2.4 48000 62000 62000
*** warning - w1:w2:w3 ratios differ by over 10%
     Fx  =   0.1000e+01  0.7996e+02  0.1643e+03
     Mxx =   0.1000e+01  0.7860e+02  0.1119e+03
     Myy =   0.1000e+01  0.8840e+02  0.1216e+03
     Mzz =   0.1000e+01  0.8840e+02  0.1216e+03
Strain rate factors (0-3, dflt=none) : 20 250 1?20
Corresponding strength factors : .65 .95 1.2
Use default tolerances? : y

Any errors? :
```

```
This set added to BEAM file

Specify a new property set? :

No. of property sets in BEAM file =  1

Display property set descriptions? : y

BEAM PROPERTY DESCRIPTIONS
  Set No.   Description
     1      24in. sch.100, with strain rate.

Display new property set data? : y

BEAM PROPERTY DATA

SET NO.  1.  24in. sch.100, with strain rate.

 Data Type                   Data 1      Data 2      Data 3      Data 4

 Unit weight              0.3060e-01
 Elastic EAx,GAy,GAz      0.2886e+07 0.1000e+11 0.1000e+11
         GJx,EIy,EIz      0.1402e+09 0.1822e+09 0.1822e+09
 First yield
       Fx,Mxx,Myy,Mzz     0.2886e+04 0.1520e+05 0.1520e+05 0.1520e+05
       EA,GJx,EIy,EIz     0.2367e+05 0.2080e+07 0.2400e+07 0.2400e+07
 Second yield
       Fx,Mxx,Myy,Mzz     0.4755e+04 0.3270e+05 0.3270e+05 0.3270e+05
       EA,GJx,EIy,EIz     0.2370e+02 0.3600e+06 0.4700e+06 0.4700e+06
 Third yield
       Fx,Mxx,Myy,Mzz     0.4757e+04 0.3400e+05 0.3400e+05 0.3400e+05
       EA,GJx,EIy,EIz     0.2400e+01 0.4800e+05 0.6200e+05 0.6200e+05
 Strain rate factors      0.2000e+02 0.2500e+03 0.1220e+04
 Strength factors         0.6500e+00 0.9500e+00 0.1200e+01
 Stiffness tolerance      0.5000e-01
 Yield tolerance          0.2000e-01
 Unloading tolerance      0.2000e-01
 Str. rate tolerance      0.5000e-01
Hit RETURN for next set
END OF DATA

Display all property set data? :
Write in session log? :

New BEAM file created
Comment for file catalog : Illustration, Table A6.2.1.

End this BEAM session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : quit
```

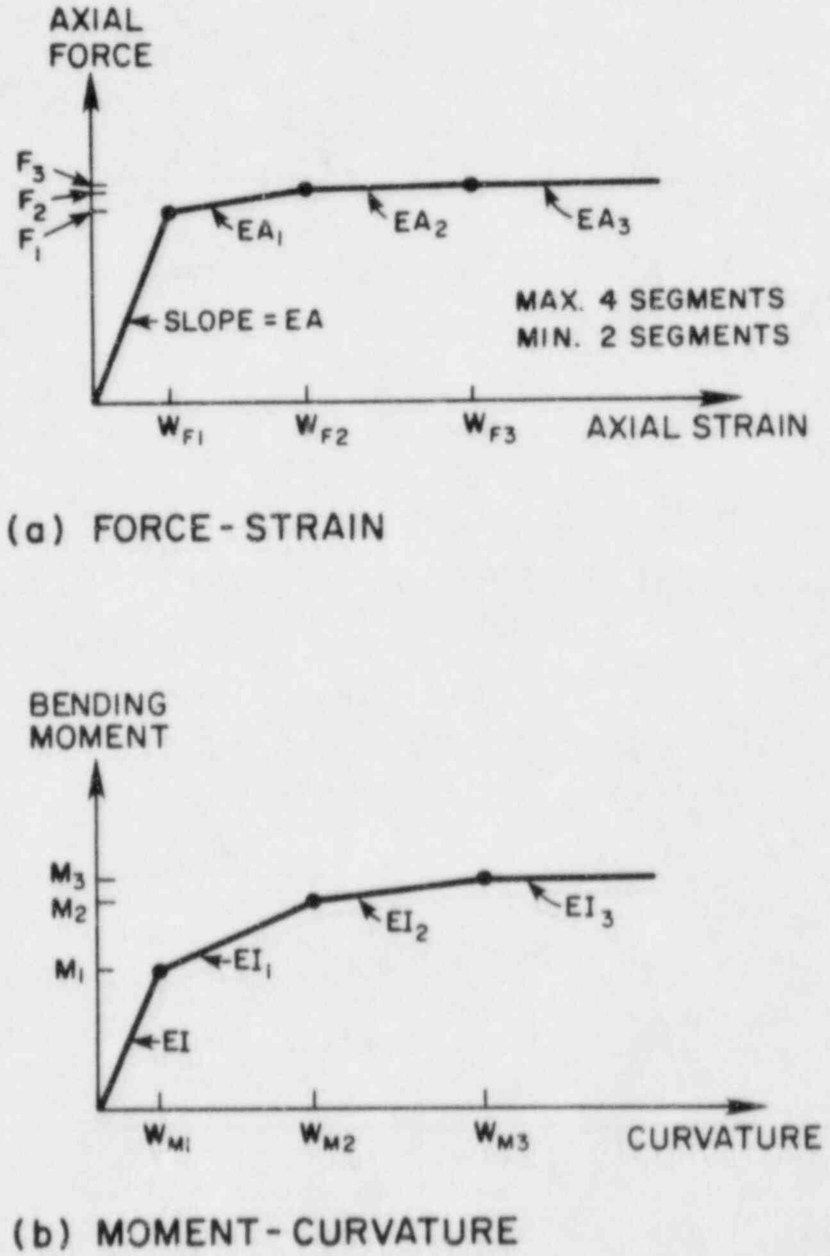
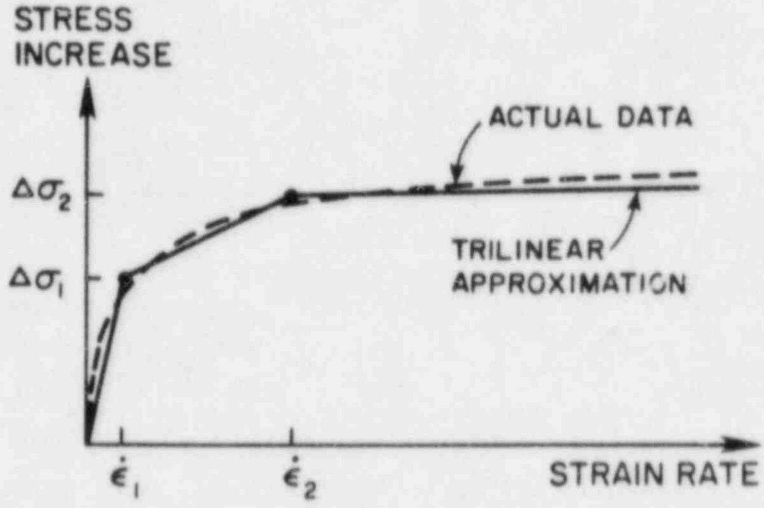
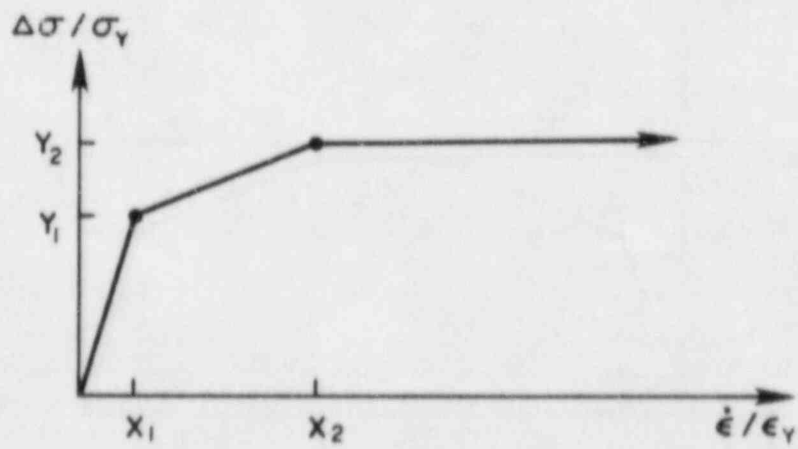


FIG. A6.2.1 - BEAM ACTION-DEFORMATION RELATIONSHIPS



(a) STRESS INCREASE VS STRAIN RATE



(b) DIMENSIONLESS FORM

FIG. A6.2.2 - BEAM STRAIN RATE PROPERTIES

A6.3 WIPS-UBAR USER GUIDE

A6.3.1 PURPOSE

WIPS-UBAR accepts "sets" of property data for U-bar elements and stores them in the UBAR file. U-bars with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by specifying the element location, the element orientation, and the property set to be used. WIPS-MODL extracts the appropriate data from the UBAR file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-UBAR session or in a series of separate sessions. New property sets are added at the end of the UBAR file. Property sets are numbered in the order they are specified.

A6.3.2 INPUT DATA

A6.3.2.1 General

The theory of the UBAR element is presented in Section B6. A typical WIPS user should be familiar with the main features of the element but need not be concerned with the details of the theory.

A U-bar element is modeled essentially as a cable, which can provide only axial tensile resistance. The element is assumed to have an initial gap (in effect, the cable is initially slack), and hence, to have no initial stiffness. If the gap closes, tensile resistance develops. This resistance will typically be nonlinear and can depend on the extension rate of the element, if desired.

WIPS-UBAR first requests data on the static force-extension relationship, then data on extension rate effects (optional), and finally certain other data, as follows.

A6.3.2.2 Static Behavior

For static load, the force-extension relationship follows a multi-linear curve (Fig. A6.3.1), with a minimum of 2 and a maximum of 6 segments. The stiffnesses K_1, K_2, etc. must first be specified, followed by the strengths F_1, F_2, etc. The stiffnesses must decrease progressively ($K_1 > K_2 > K_3$, etc.). For linear elastic behavior, specify K_2 small and F_1 very large.

A6.3.2.3 Extension Rate Effect

For dynamic loading, the strength of the element may depend on the extension rate. From the results of experimental studies, first develop a curve relating strength *increase* to extension rate (Fig. A6.3.2). Next, approximate this curve by a multi-linear curve, with up to 3 segments (Fig. A6.3.2). The extension rate coefficients C_1, C_2, and C_3 must first be specified, followed by the strength increases ΔF_1, ΔF_2. For no extension rate effect, specify no C values.

A6.3.2.4 Default Gap

The initial gap may be specified, if desired, or may be specified in the WIPS-MODL phase, as optional data. If no gap is specified in WIPS-MODL, it defaults to the value specified in WIPS-UBAR.

A6.3.2.5 Nonlinear Tolerance

A tolerance to control error in the nonlinear solution may be specified, if desired. Typically, the default value (100 lb. force) will be reasonable.

A6.3.3 EXAMPLE

Table A6.3.1 is a listing of a session log to illustrate the procedure for specifying UBAR properties.

TABLE A6.3.1

WIPS-UBAR EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : ubar

UBAR - SPECIFICATION OF U-BAR PROPERTIES

Define units
  Length (ft/in/m/mm) : in
  Force (k/lb/kgf/kN) : k

Start new UBAR file

Specify a new property set? : y

SET NO. =  1
Property set description : Illustration. No strain rate.
Static stiffnesses (min=2,max=6)
 : 100 35 10 1
Static strengths (no. of stiffnesses minus 1)
 : 20 25 25.5
Extension rate Stiffnesses (min=0,max=3)
 :
No extension rate effect
Default gap clearance : .5
Use default tolerances? : y

Any errors? :

This set added to UBAR file

Specify a new property set? : y

SET NO. =  2
Property set description : Illustration. With strain rate.
Static stiffnesses (min=2,max=6)
 : 100 35 10 1
Static strengths (no. of stiffnesses minus 1)
 : 20 25 25.5
Extension rate Stiffnesses (min=0,max=3)
 : 10 .2
Strength limits (no. of stiffnesses minus 1)
 : 6
Default gap clearance :
*** error - must be positive
Default gap clearance : .5
Use default tolerances? : no
Stiffness tolerance (dflt=.05) : .1
```

```
Overshoot tolerance (dflt=200lb) : .3
Unloading tolerance (dflt=200lb) : .1

Any errors? :

This set added to UBAR file

Specify a new property set? :

No. of property sets in UBAR file =  2

Display property set descriptions? : y

UBAR PROPERTY DESCRIPTIONS
  Set No.   Description
     1      Illustration. No strain rate.
     2      Illustration. With strain rate.

Display new property set data? :
Write in session log? :

Display all property set data? : y

UBAR PROPERTY DATA

SET NO.  1.  Illustration. No strain rate.

        Data                 Segm  Stiffness or    Force
        Type                  No.   Coefficient    Limit

        Static properties      1     0.1000e+03  0.2000e+02
                               2     0.3500e+02  0.2500e+02
                               3     0.1000e+02  0.2550e+02
                               4     0.1000e+01
        Default gap                  0.5000e+00
        Stiffness tol.               0.5000e-01
        Overshoot tol.               0.2000e+00
        Unloading tol.               0.2000e+00
Hit RETURN for next set

SET NO.  2.  Illustration. With strain rate.

        Data                 Segm  Stiffness or    Force
        Type                  No.   Coefficient    Limit

        Static properties      1     0.1000e+03  0.2000e+02
                               2     0.3500e+02  0.2500e+02
                               3     0.1000e+02  0.2550e+02
                               4     0.1000e+01
        Rate effect            1     0.1000e+02  0.6000e+01
                               2     0.2000e+00
        Default gap                  0.5000e+00
        Stiffness tol.               0.1000e+00
        Overshoot tol.               0.3000e+00
        Unloading tol.               0.1000e+00
Hit RETURN for next set
```

```
END OF DATA

New UBAR file created
Comment for file catalog : Illustration, Table A6.3.1.

End this UBAR session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit

-----------------------------------------------------------------
```

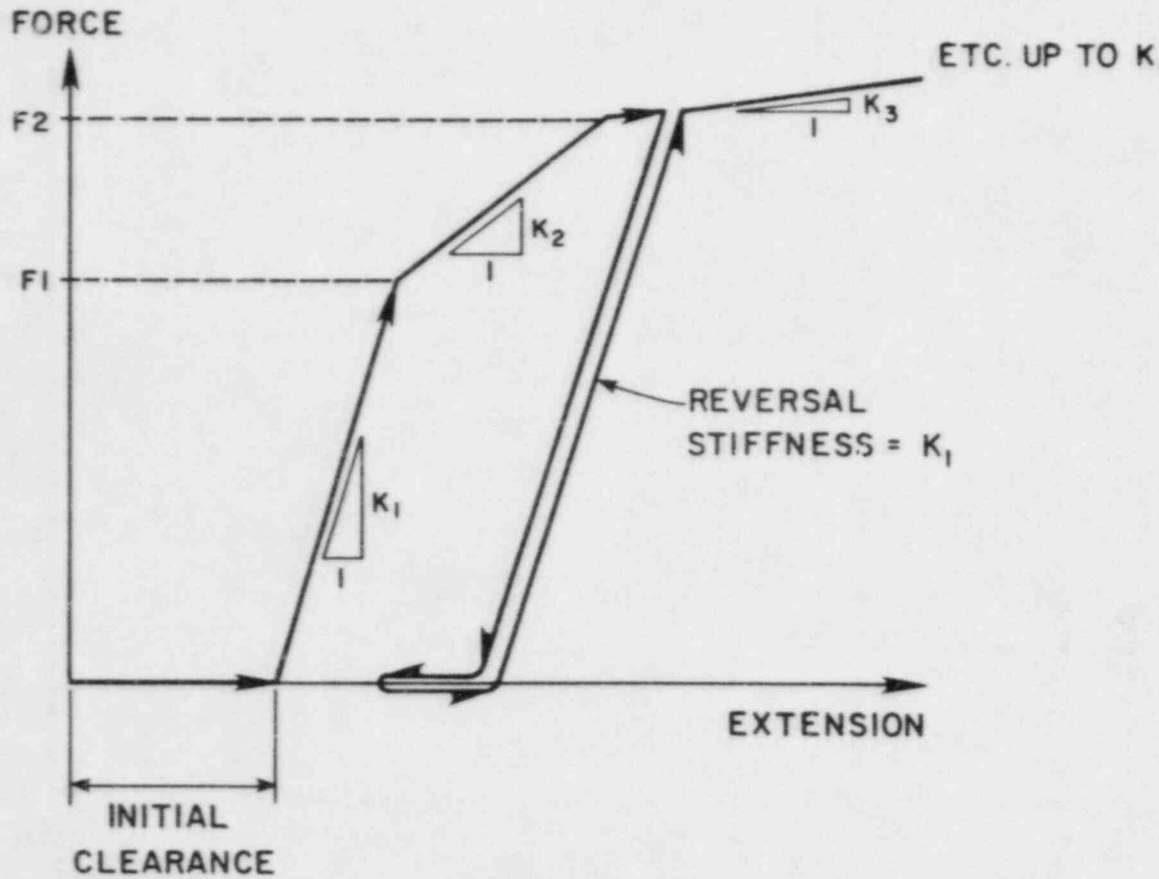


FIG. A6.3.1 - U-BAR FORCE-EXTENSION RELATIONSHIP

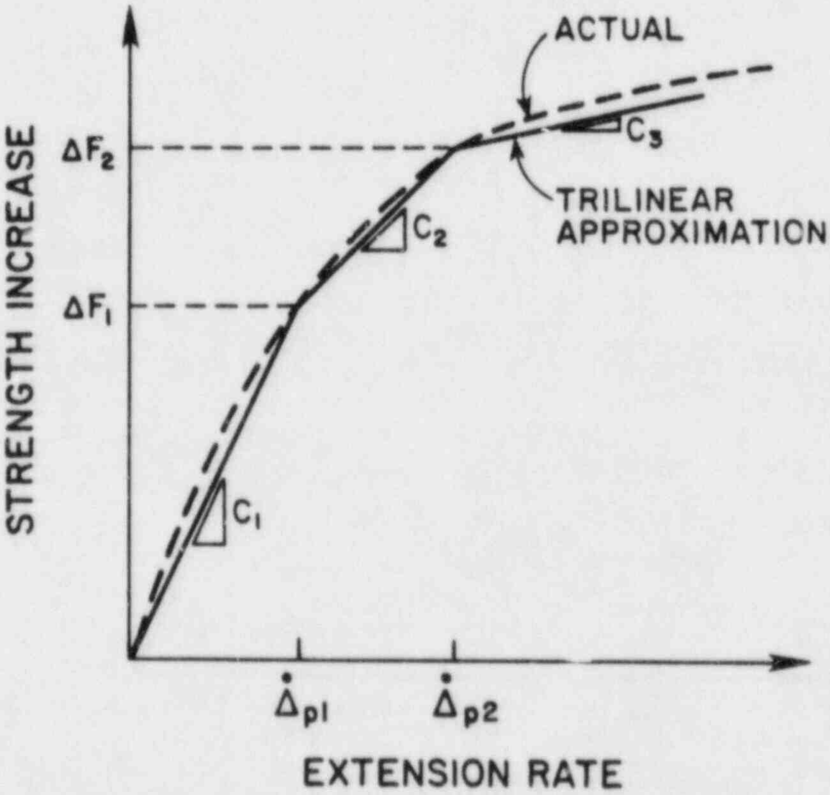


FIG. A6.3.2 - U-BAR STRAIN RATE PROPERTIES

A6.4 WIPS-GAPF USER GUIDE

A6.4.1 PURPOSE

WIPS-GAPF accepts "sets" of property data for gap-friction elements and stores them in the GAPF file. Gap-friction elements with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by specifying the element location, the element orientation, and the property set to be used. WIPS-MODL extracts the appropriate data from the GAPF file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-GAPF session or in a series of separate sessions. New property sets are added at the end of the GAPF file. Property sets are numbered in the order they are specified.

A6.4.2 INPUT DATA

A6.4.2.1 General

The GAPF element can be used to model walls or similar barriers. The theory of the element is presented in Section B8. A typical WIPS user should be familiar with the main features of the element but need not be concerned with the details of the theory.

A gap-friction element is modeled as a linear bearing component plus a nonlinear friction component. The element is assumed to have an initial gap, and hence, to have no initial stiffness. If the gap closes, bearing and friction resistance develops at the barrier surface. If the friction resistance exceeds the bearing force multiplied by the friction coefficient, slip takes place. The bearing component is modeled by an elastic spring oriented normal to the barrier plane. The friction component is modeled by inelastic springs oriented tangential to the barrier plane.

A6.4.2.2 Basic Properties

WIPS-GAPF first requests the normal and tangent spring stiffnesses (i.e. the bearing stiffness and the tangential stiffness at the barrier before slip occurs) and the friction coefficient. The stiffnesses should be reasonably realistic values, recognizing that no structure can be completely rigid. If extremely large stiffnesses are specified, it is possible for numerical instability to develop in the dynamic analysis.

A6.4.2.3 Tolerances

In the dynamic analysis, tolerances are required to control stiffness reformulation when nonlinear "events" (e.g. gap closure) occur. The following two tolerances are used.

(1) An angle tolerance (in radians) to control stiffness reformulation as the direction of slip changes. The default value is 0.05.

(2) An "overshoot" tolerance (in force units) to control stiffness reformulation when gap closure, gap opening, friction slip, or friction slip reversal occur. The default value is 200 lbs.

The default values will typically apply.

A6.4.2.4 Other Data

The remaining data is specified in the WIPS-MODL phase. This includes data defining the initial gap and the orientation of the bearing plane.

A6.4.3 EXAMPLE

Table A6.4.1 is a listing of a session log to illustrate the procedure for specifying GAPF properties.

TABLE A6.4.1

WIPS-GAPF EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : gapf

GAPF - SPECIFICATION OF GAP FRICTION PROPERTIES

Define units
  Length (ft/in/m/mm) : in
  Force (k/lb/kgf/kN) : k

Start new GAPF file

Specify a new property set? : y

SET NO.  1
Property set description : Illustration.
Normal stiffness : 1.e4
Tangent stiffness : 1000
Friction coefficient : .45
Use default tolerances? y
Any errors? :

This set added to GAPF file

Specify a new property set? : y

SET NO.  2
Property set description : Second illustration.
Normal stiffness : 150
Tangent stiffness : 40
Friction coefficient : .3
Use default tolerances? n
Stiffness tolerance (dflt=.05) : .1
Overshoot tolerance (dflt=200lb) : .4
Any errors? :

This set added to GAPF file

Specify a new property set? :

No. of property sets in GAPF file =  2

Display property set descriptions? : y

GAPF PROPERTY DESCRIPTIONS
  Set No.   Description
     1      Illustration.
```

```
     2      Second illustration.

Display new property set data? : y

 QAPF PROPERTY DATA

Set   Data                           Data
No.   Type                           Value
  1
      Normal stiffness        0.1000e+05
      Tangent stiffness       0.1000e+04
      Friction coefficient    0.4500e+00
      Stiffness tolerance     0.5000e-01
      Overshoot tolerance     0.2000e+00
Hit RETURN for next set
  2
      Normal stiffness        0.1500e+03
      Tangent stiffness       0.4000e+02
      Friction coefficient    0.3000e+00
      Stiffness tolerance     0.1000e+00
      Overshoot tolerance     0.4000e+00
Hit RETURN for next set
END OF DATA

Display all property set data? :
Write in session log? :

New QAPF file created
Comment for file catalog : Illustration, Table A6.4.1.

End this CAPF session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

A7. SUBSTRUCTURE USER GUIDES

SUMMARY

This section contains user guides for defining the properties for each different substructure type. Three substructures types may be defined, namely straight pipes (WIPS-STRP, Section A7.1), elbows (WIPS-ELBO, Section A7.2), and flat walls or slabs (WIPS-SLAB, Section A7.3).

CONTENTS

A7.1 WIPS-STRP USER GUIDE

A7.1.1 PURPOSE

WIPS-STRP accepts "sets" of property data for straight pipe substructures and stores them in the STRP file. Substructures with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by defining the location in a pipe run and the property set to be used. WIPS-MODL extracts the appropriate data from the STRP file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-STRP session or in a number of separate sessions. New property sets are added at the end of the STRP file. Property sets are numbered in the order they are specified.

A7.1.2 SUBSTRUCTURE DEFINITION

A7.1.2.1 Geometry

A straight pipe substructure consists of three segments (Fig. A7.1.1). The number of finite element subdivisions around the pipe circumference must be the same in all three segments, but the longitudinal subdivisions may be different. The main purpose of the segments is to permit a fine finite element mesh in the center segment, with progressively coarser (expanding) meshes towards the substructure ends. However, different pipe diameters, wall thicknesses, and material properties may be specified in the three segments, if desired.

A substructure property set is defined using local coordinates, as shown in Fig. A7.1.1. Transformation to the global system takes place in the WIPS-MODL phase when the substructure is incorporated into an analysis model. In the WIPS-MODL phase, it is not necessary for the substructure length in the analysis model to agree exactly with that specified in WIPS-STRP. Specifically, the total length in WIPS-MODL may be up to 30% different from the sum of the segment lengths in WIPS-STRP. This avoids the need to specify the dimensions exactly in WIPS-STRP. In WIPS-MODL, the segment lengths are automatically scaled to match the length in the analysis model.

A7.1.2.2 End Cross Sections: General Case

The substructure is modeled using shell elements. At the end cross sections, the substructure is connected to the rest of the analysis model through nodes on the pipe axis. The shell nodes at the two end cross sections are automatically slaved to these nodes, *assuming plane, circular cross sections.* This same slaving assumption is made even if two substructures connect directly to each other in the analysis model (e.g. an elbow substructure connecting to a straight pipe substructure). That is, if two substructures connect, the cross section between them is assumed to remain plane and circular. The only exception to this rule is that described in the following section.

A7.1.2.3 End Cross Sections: Longitudinal Symmetry

In WIPS-MODL an option is provided to define a longitudinally symmetrical boundary condition at the J end (right-hand end in Fig. A7.1.1) of any straight pipe substructure, to allow modeling of symmetrical pipe runs. In this case, the plane, circular section condition is imposed at end I of the substructure, but at end J only a plane section condition is imposed, and distortion of the cross section is permitted.

Substantial computer time can be saved in WIPS-ANAL if advantage is taken of longitudinal symmetry. For such symmetry to be specified, however, it is necessary for the pipe axis to lie along a global X, Y, or Z axis in the analysis model. This restriction is considered in detail in the WIPS-MODL User Guide.

A7.1.2.4 Transverse Symmetry

In WIPS-MODL, an option is also provided to define transverse symmetry (i.e. consider only one-half of the pipe circumference). For this reason, the number of circumferential subdivisions must be even. The procedure for defining symmetry is described in the WIPS-MODL User Guide.

A7.1.2.5 Gauss Integration Points Through Thickness

A straight pipe substructure is modeled using shell finite elements. Yielding of each element is monitored at a number of Gauss integration points through the element thickness. If an element remains elastic, two integration points are sufficient. If an element may yield, it is recommended that five integration points be specified. For greater accuracy (but at greater cost) up to seven points may be used.

In many cases, yielding will be confined to segment 2 of a straight pipe substructure, with elastic behavior in segments 1 and 3. If this is the case, computer time can be saved by specifying only two Gauss points in the outer segments.

A7.1.2.6 Impact

For specifying impact surfaces, the mesh subdivisions are assumed to be numbered as shown in Fig. A7.1.1. When regions for impact are specified (in WIPS-MODL), this numbering scheme must be used.

A7.1.2.7 Substructure Orientation

In the WIPS-MODL phase, the orientation of a straight pipe substructure is determined as follows. Figure A7.1.2 illustrates the procedure.

(1) Ends I and J must be two control points in a pipe run. This locates the substructure local x axis.

(2) A third control point (Point K) must be specified to define the local xy plane. As shown in Fig. A7.1.2, the points IJK define the xy plane.

If straight pipe substructures are to be specified in WIPS-MODL, an appropriate control point to use as Point K must be present in the WIPS-GEOM data for each substructure. This may have to be a specially defined reference point.

Point K is *not* needed if the substructure has transverse symmetry. In this case the half circumference starting with circumferential point 1 is assumed, and the substructure xy plane is the plane of symmetry.

A7.1.3 REQUIRED INPUT DATA

For each property set, WIPS-STRP requests input data as follows.

(1) Number of circumferential subdivisions.

(2) Length, number of longitudinal divisions and (for segments 1 and 3) mesh expansion factor for each segment. If the expansion factor is, say, 1.2, the length of each longitudinal subdivision is 1.2 times larger than the preceding subdivision, *moving towards the substructure ends.* In Fig. A7.1.1 the expansion factor is 1.2.

(3) Pipe outside diameter and wall thickness for each segment. For segments 2 and 3, the default is the same dimension as segment 1.

(4) Material property set numbers (in MATL file) for each segment. For segments 2 and 3, the default is the same material as segment 1.

All other data (defining the substructure orientation, symmetry conditions, internal pressure, etc.) are specified in the WIPS-MODL phase.

A7.1.4 EXAMPLE

Table A7.1.1 shows the session log for the straight pipe substructure in Fig. A7.1.1. Note that the finite element mesh in this example is very coarse. For actual analyses, a substantially finer mesh will typically be needed.

TABLE A7.1.1

WIPS-STRP EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : strp

STRP - SPECIFICATION OF STRP GEOMETRY

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

No. of MATL property sets available =  3
Display material descriptions? :
Write in session log? :

Start new STRP file

Specify a new property set? : y

Set no. =  1

Substructure description : 6in. sch. 40, 18in. length.
No. of circumf. divisions : 12          NOTE TOO SMALL FOR ACCURATE MODELLIN(
First Segment
   Length : 6
   No. of divisions : 3
   Mesh factor : 1.2
   Outside diameter : 6.625
   Wall thickness : .28
   Integ. pts thru thickness : 2
Center Segment
   Length : 6
   No. of divisions : 4
   Outside diameter : 6.625
   Wall thickness : .28
   Integ. pts thru thickness : 5
Last Segment
   Length : 6
   No. of divisions : 3
   Mesh factor : 1.2
   Outside diameter : 6.625
   Wall thickness : .28
   Integ. pts thru thickness : 2

Material numbers (seg.1, seg.2, seg.3) : 1

Any errors? :
```

```
This set added to STRP file

Specify a new property set? :

No. of property sets in STRP file =  1

Display pre.erty set descriptions? : y

STRP PROPERTY DESCRIPTIONS
  Set No.   Description
     1      6in. sch. 40, 18in. length.

Display new property set data? : y

STRP PROPERTY DATA

SET NO.  1.  6in. sch. 40, 18in. length.

  Circumferential divisions = 12

                 Segment 1    Segment 2    Segment 3
  length         0.6000e+01   0.6000e+01   0.6000e+01
  divisions          3            4            3
  mesh factor    0.1200e+01                0.1200e+01
  diameter       0.6625e+01   0.6625e+01   0.6625e+01
  thickness      0.2800e+00   0.2800e+00   0.2800e+00
  integ. order       2            5            2
  material no.       1            1            1
Hit RETURN for next set
END OF DATA

Display all property set data? : no        ALREADY DISPLAYED
Write in session log? : no                 DISPLAY WRITES TO LOG ALSO

New STRP file created
Comment for file catalog : Illustration. Table A7.1.1.

End this STRP session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

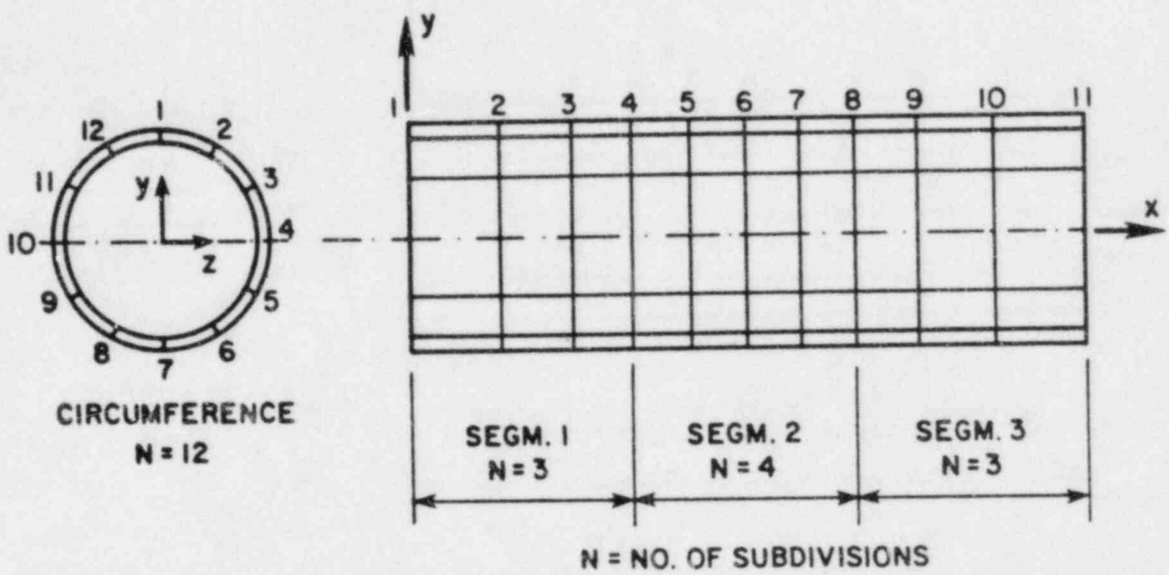


FIG. A7.1.1 - STRP SUBSTRUCTURE

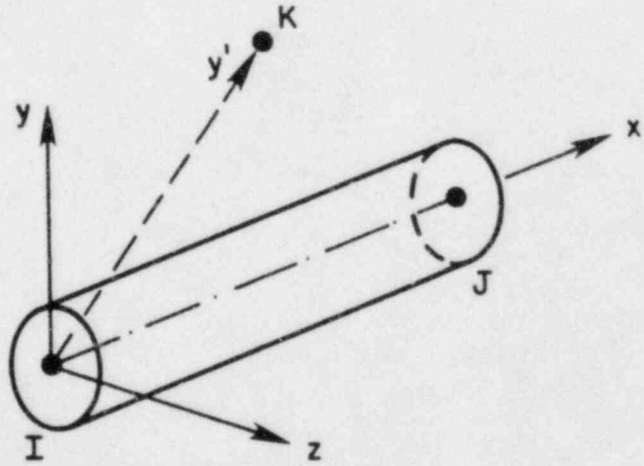


FIG. A7.1.2 - STRP LOCAL AXES

A7.2 WIPS-ELBO USER GUIDE

A7.2.1 PURPOSE

WIPS-ELBO accepts "sets" of property data for elbow substructures and stores them in the ELBO file. Substructures with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase by defining the location in a pipe run and the property set to be used. WIPS-MODL extracts the appropriate data from the ELBO file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-ELBO session or in a number of separate sessions. New property sets are added at the end of the ELBO file. Property sets are numbered in the order they are specified.

A7.2.2 SUBSTRUCTURE DEFINITION

A7.2.2.1 Geometry

An elbow substructure consists of an elbow connected to tangent lengths of straight pipe (Fig. A7.2.1). The substructure is initially defined in a local coordinate system, as shown. Transformation to the global system takes place automatically when a substructure is specified as part of a pipe run in the WIPS-MODL phase.

The three parts of the substructure (tangent 1, elbow, and tangent 2) may have different pipe diameters, wall thicknesses, and material properties. The number of finite element subdivisions around the pipe circumference must be the same in all three parts, but the numbers of longitudinal subdivisions may be different. The longitudinal subdivision is uniform within the elbow. However, in the tangents the element sizes may increase progressively away from the elbow, as indicated in Fig. A7.2.1. This is done to allow a fine element subdivision in the elbow and in the tangents close to the elbow, but progressively coarser subdivisions in the less critical regions away from the elbow.

When an elbow substructure is incorporated into an analysis model in the WIPS-MODL phase, it is not necessary for the model dimensions to agree exactly with the substructure dimensions. Specifically, the tangent lengths, bend radius, and bend angle may all vary by up to 30% from the values specified in the WIPS-ELBO phase. This avoids the need to specify the dimensions exactly in WIPS-ELBO.

A7.2.2.2 End Cross Sections: General Case

The tangents and elbows are modeled using shell elements. At the end cross sections, the substructure is connected to the rest of the analysis model through nodes on the pipe axis. The shell nodes at the two end cross sections are automatically slaved to these nodes, *assuming plane, circular cross sections.* This same slaving assumption is made even if two substructures connect directly to each other in the analysis model (e.g. an elbow substructure connecting to a straight pipe substructure). That is, if two substructures connect, the cross section at the junction between them is assumed to remain plane and circular. The only exception to this rule is that described in the following section.

A7.2.2.3 End Cross Sections: Longitudinal Symmetry

In WIPS-MODL an option is provided to define a longitudinally symmetrical boundary condition at the J end of any elbow substructure (right-hand end in Fig. A7.2.1), to allow modeling of longitudinally symmetrical configurations. In this case, the plane, circular section condition is imposed at end I of the substructure, but at end J only a plane section condition is imposed, and distortion of the cross section is permitted.

A7.2.2.4 Transverse Symmetry

In WIPS-MODL, an option is also provided to define transverse symmetry (i.e. consider only one-half of the pipe circumference. For this reason, the number of circumferential subdivisions must be even. The procedure for defining symmetry is described in the WIPS-MODL User Guide.

A7.2.2.5 Zero Length for Tangent 2

If desired, the length of tangent 2 may be specified to be zero. This makes it possible to model longitudinally symmetrical elbows by introducing a plane of symmetry at the elbow midpoint. Tangent 2 should have zero length only for such symmetrical situations. If the length of tangent 2 is zero and symmetry is not specified, ovalling of the end J cross section will be prevented, and substantially incorrect results are likely to be obtained.

A7.2.2.6 Gauss Integration Points Through Thickness

An elbow substructure is modeled using shell finite elements. Yielding of each element is monitored at a number of Gauss integration points through the element thickness. If an element remains elastic, two integration points are sufficient. If an element may yield, it is recommended that five integration points be specified. For greater accuracy (but at greater cost) up to seven points may be used.

In many cases, yielding will be confined to the elbow part of an elbow substructure, with elastic behavior in tangent 1 and tangent 2. If this is the case, computer time can be saved by specifying only two Gauss points in the tangent regions.

A7.2.2.7 Impact

For specifying impact surfaces, the mesh subdivisions are assumed to be numbered as shown in Fig. A7.2.1. When regions for impact are specified (in WIPS-MODL), this numbering scheme must be used.

A7.2.3 REQUIRED INPUT DATA

For each property set, WIPS-ELBO requests input data as follows.

(1) Number of circumferential subdivisions.

(2) Length, number of longitudinal subdivisions, and mesh expansion factor for tangent 1. If the mesh expansion factor is 1.0, a uniform mesh is produced. If the factor is, say, 1.2, the length of each longitudinal subdivision is 1.2 times larger than the length of the preceding subdivision, *moving away from the elbow.* In Fig. A7.2.1 the expansion factor is 1.2.

(3) Pipe outside diameter, wall thickness, and number of Gauss points through thickness for tangent 1.

(4) Bend radius (to pipe axis), bend angle (angle between tangents, in degrees), and number of longitudinal subdivisions for the elbow.

(5) Diameter, wall thickness, and number of Gauss points for the elbow. The default is the same dimension as tangent 1.

(6) Length, number of longitudinal subdivisions, and mesh expansion factor for tangent 2.

(7) Diameter, wall thickness, and number of Gauss points for tangent 2. The default is the same dimension as tangent 1.

(8) Material property set numbers (in MATL file) for tangent 1, the elbow, and tangent 2, respectively. For the elbow and tangent 2, the default is the same material as tangent 1.

All other data (defining the substructure orientation, symmetry conditions, internal pressure, etc.) are specified in the WIPS-MODL phase.

A7.2.4 EXAMPLE

Table A7.2.1 shows the session log for definition of the elbow substructure in Fig. A7.2.1. Note that the finite element mesh in this example is very coarse. For actual analyses, a substantially finer mesh will typically be needed.

TABLE A7.2.1

WIPS-ELBO EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : elbo


ELBO - SPECIFICATION OF ELBO PROPERTIES

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

No. of MATL property sets available =  3
Display material descriptions? : y

MATL PROPERTY DESCRIPTIONS
   Set No.  Type  Description
     1      mroz  ASTM A106 Grade B. Trilinear.
     2      mroz  ASTM A312 Type 304L. Trilinear.
     3      mroz  Elastic-Plastic, Fy = 280 MPa.

Start new ELBO file

Specify a new property set? : y

SET NO.  1

Substructure description : 6sch40, 45degLR, A106-B, w strain rate.
No. of circumf. divisions : 12       NOTE: TOO SMALL FOR ACCURATE MODELING
First Tangent
   Length : 9
   No. of divisions : 5
   Mesh factor : 1.2
   Outside diameter : 6.625
   Wall thickness : .28
   No. of integ. pts thru thickness : 5
Elbow
   Bend radius : 9
   Bend angle (deg) : 45
   No. of divisions : 6
   Outside diameter : 6.625
   Wall thickness : .28
   No. of integ. pts thru thickness : 5
Second Tangent
   Length : 9
   No. of divisions : 5
   Mesh factor : 1.2
   Outside diameter : 6.625
```

```
    Wall thickness : .28
    No. of integ. pts thru thickness : 5
Material numbers (tang.1, elbow, tang.2) : 1

Any errors? : no              IF "YES" MUST RE-ENTER DATA

This set added to ELBO file

Specify a new property set? :

No. of property sets in ELBO file =  1

Display property set descriptions? : y

ELBO PROPERTY DESCRIPTIONS
   Set No.   Description
      1      6sch40, 45degLR, A106-B, w strain rate.

Display new property data? : y

ELBO PROPERTY DATA

SET NO.  1.  6sch40, 45degLR, A106-B, w strain rate.
   Circumferential divisions = 12

                    Tangent 1        Elbow     Tangent 2
   Length          0.9000e+01                 0.9000e+01
   Divisions            5              6           5
   Mesh factor     0.1200e+01                 0.1200e+01
   Diameter        0.6625e+01   0.6625e+01   0.6625e+01
   Thickness       0.2800e+00   0.2800e+00   0.2800e+00
   Bend radius                  0.9000e+01
   Angle (deg)                  0.4500e+02
   Integ. order         5              5           5
   Material no.         1              1           1
Hit RETURN for next set
END OF DATA

Display all property data? :
Write in session log? :

New ELBO file created
Comment for file catalog : Example, Section A7.2

End this ELBO session? : y

EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : quit
```

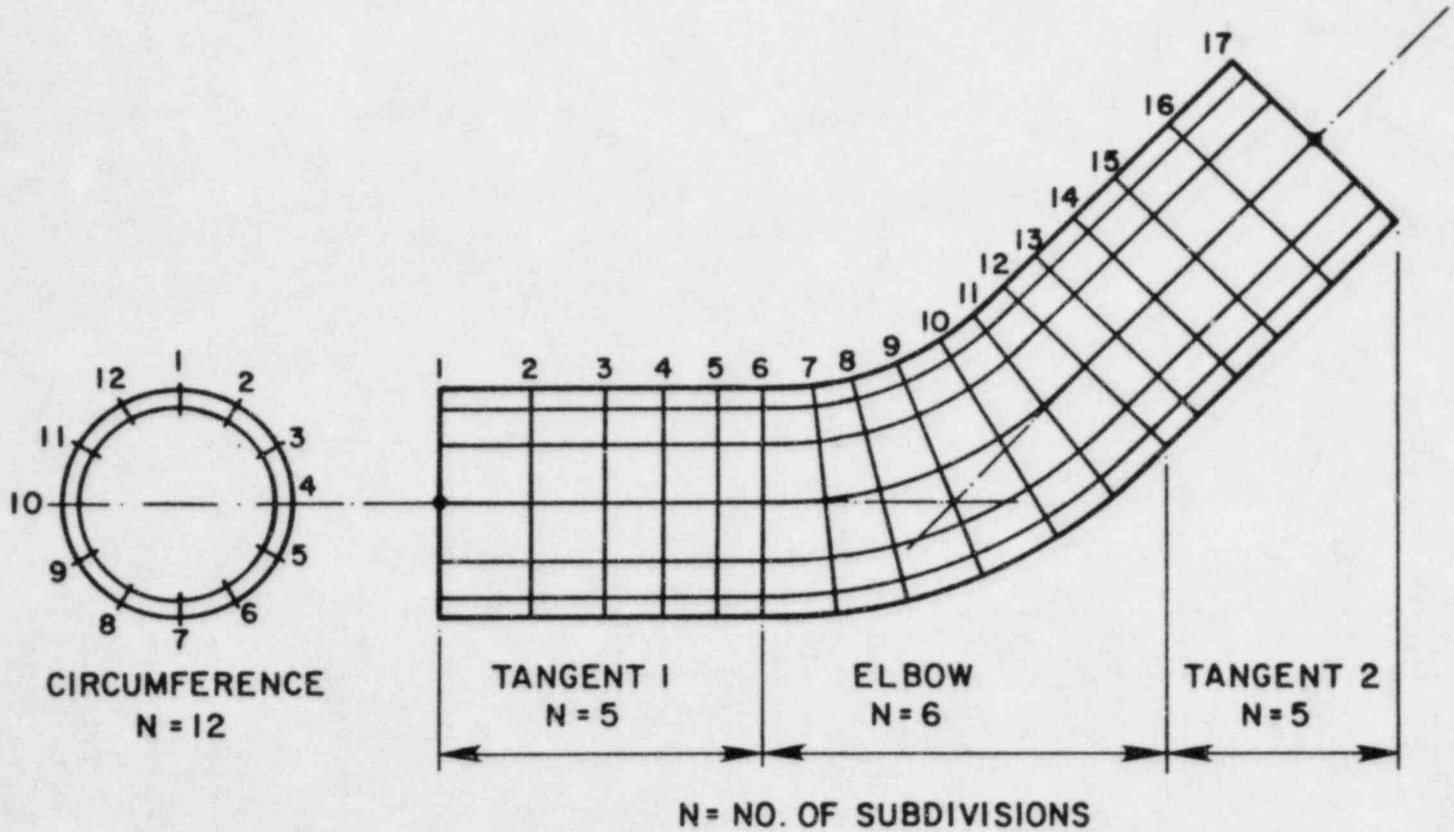


FIG. A7.2.1 - ELBOW SUBSTRUCTURE

A7.3 WIPS-SLAB USER GUIDE

A7.3.1 PURPOSE

WIPS-SLAB accepts "sets" of property data for flat slab substructures and stores them in the SLAB file. Substructures with particular properties can then be incorporated into an analysis model in the WIPS-MODL phase, by defining the property sets to be used. WIPS-MODL extracts the appropriate data from the SLAB file and incorporates it into a MODL file.

The property sets may be specified in a single WIPS-SLAB session or in a number of separate sessions. New property sets are added at the end of the SLAB file. Property sets are numbered in the order they are specified.

A7.3.2 SUBSTRUCTURE PROPERTIES

A7.3.2.1 Geometry

A flat slab substructure must be flat and of rectangular or parallelogram shape. A substructure may be arbitrarily oriented in space, as shown in Fig. A7.3.1. The property data includes data defining the finite element mesh, coordinate data to locate the substructure in the global X,Y,Z system, and data on the support conditions along the substructure boundaries.

The surface OACB (Fig. A7.3.1) defines the *midthickness* of the slab. The finite element mesh is divided into three segments along each of the local x and y directions, defining nine regions. The mesh must be uniform in each region but may be coarser in the outer regions than in the center region. This allows small finite elements near the center of the substructure, with larger elements towards the boundaries. The segment lengths are defined as proportions of the lengths OA and OB. The length of any outer segment may be zero so that only two (or one) segments are present along any edge.

The location of a substructure in the global X,Y,Z coordinate system is defined by specifying the following.

(1) The lengths of OA and OB.

(2) The orientations in X,Y,Z space of OA and OB, defined by their direction cosines. The direction cosines need not be normalized (i.e. they need only define vectors along OA and OB, not necessarily unit vectors). If the orientation specified for OB is not perpendicular to OA (i.e. if the OB vector defines only a line in the OACB plane, not necessarily edge OB), an option is provided for WIPS-STRP to make the slab rectangular by adjusting the OB vector. If this option is not exercised, the slab will have a parallelogram shape.

(3) The X,Y,Z coordinates of point O. These coordinates may be specified in WIPS-SLAB, or alternatively, in WIPS-MODL. If no values are specified in WIPS-MODL, the values specified in WIPS-SLAB are used. Note, however, that there is no provision in WIPS-MODL to redefine the orientations of OA and OB.

A7.3.2.2 Boundary Conditions

Supports may be specified along the substructure edges only. Any edge may be free, clamped, or hinged, or may have symmetrical boundary conditions corresponding to symmetry about the global XY, YZ, or ZX planes. A symmetrical boundary condition is permitted for an edge only if it lies in the symmetry plane.

A7.3.2.3 Gauss Integration Points Through Thickness

A flat slab substructure is modeled using shell finite elements. Yielding of each element is monitored at a number of Gauss integration points through the element thickness. If an element remains elastic, two integration points are sufficient. If an element may yield , it is recommended that five points be specified. For greater accuracy (but at greater cost), up to seven points may be used. Different numbers of Gauss points may be specified for the center region and outer regions (the same in all outer regions).

A7.3.2.4 Material Yield

The shell finite element is based on the Mroz material. This material is suitable for modeling the yield of steel but not cracking and crushing of concrete.

A7.3.2.5 Impact

For specifying impact surfaces, the mesh subdivisions are assumed to be numbered as shown in Fig. A7.3.1. When regions for impact are specified (in WIPS-MODL), this numbering scheme must be used.

A7.3.3 REQUIRED INPUT DATA

For each property set, WIPS-STRP requests input data as follows (refer to Fig. A7.3.1).

(1) The lengths of OA and OB.

(2) The lengths of a_1 and a_3 as proportions of OA. Either or both of these proportions may be zero.

(3) The lengths of b_1 and b_3 as proportions of OB. Either or both of these proportions may be zero.

(4) Numbers of element subdivisions na_1, na_2, and na_3.

(5) Numbers of element subdivisions nb_1, nb_2, and nb_3.

(6) Slab thickness (assumed to be uniform throughout).

(7) Number of Gauss integration points through thickness in center region.

(8) Number of Gauss points in outer regions.

(9) Direction cosines of OA (or components of any vector directed along OA).

(10) Direction cosines of OB (or components).

(11) Global X,Y,Z coordinates of O. These values may be over-ridden in the WIPS-MODL phase.

(12) Boundary condition codes for sides OA, BC, OB, and AC.

(13) Number of material property set (in MATL file). All elements in the slab are assumed to have the same material.

A7.3.4 EXAMPLE

Table A7.3.1 shows a typical session log for a WIPS-SLAB session.

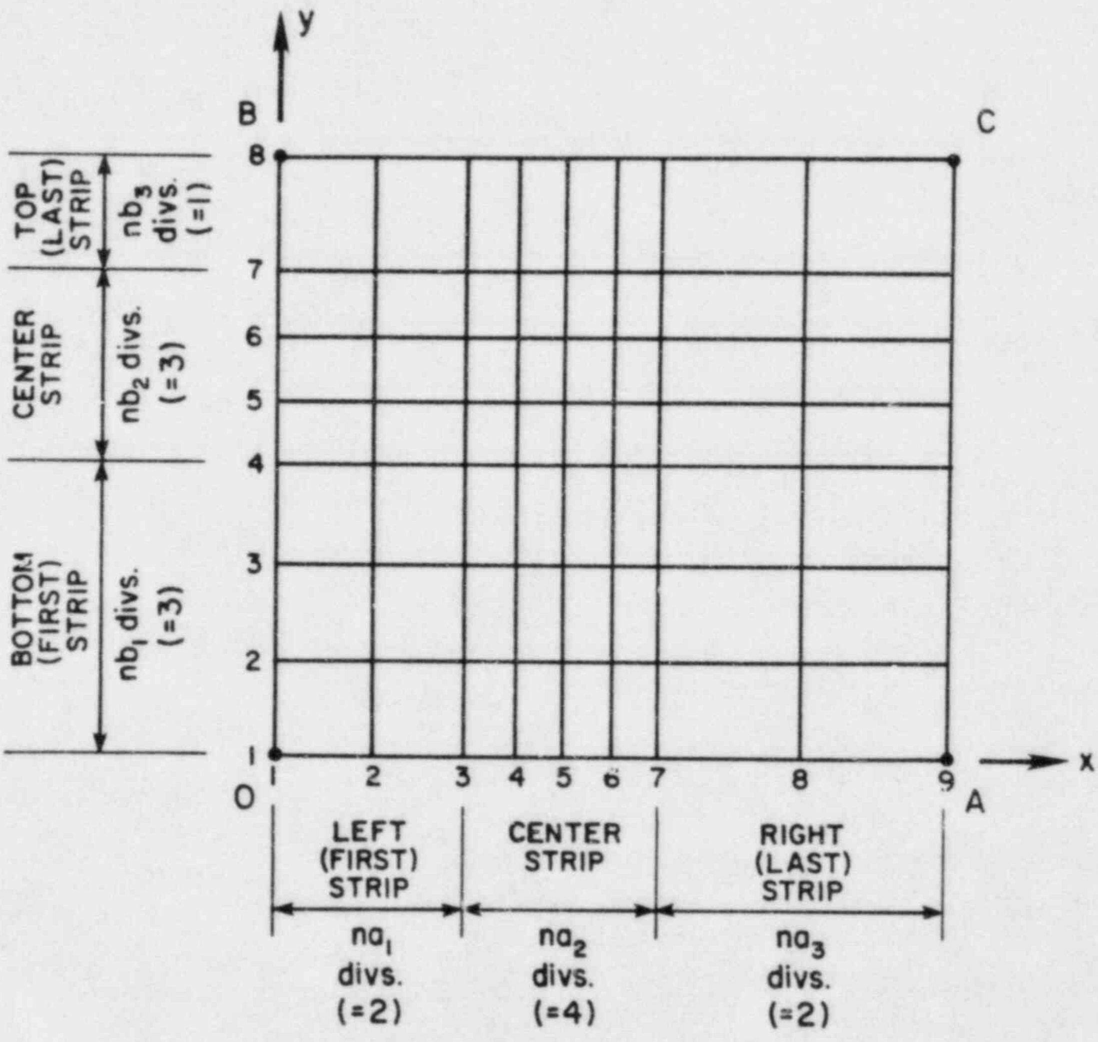
TABLE A7.3.1

WIPS-SLAB EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : slab


SLAB - SPECIFICATION OF SLAB PROPERTIES

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

No. of MATL property sets available =  3
Display material descriptions? :
Write in session log? :

Start new SLAB file

Specify a new property set? : y

SET NO.  1

Substructure description : Illustration, Fig.A7.3.1
Length of side OA : 30
Length of side OB : 25
Outer strip widths (proportions of OA)
  Left strip  : .3
  Right strip : .4
Outer strip widths (proportions of OB)
  Bottom strip : .5
  Top strip    : .2
No. of subdivisions along OA
  First strip  : 2
  Center strip : 4
  Last strip   : 2
No. of subdivisions along OB
  First strip  : 3
  Center strip : 3
  Last strip   : 1
Slab thickness : .625
No. of Gauss points thru thickness
  Center region : 5
  Outer regions : 2
Direction cosines of OA,OB
  OA (3 values) : 1                        ALONG GLOBAL X AXIS
  OB (3 values) : 0 1 1                    45 DEGREE INCLINATION
Coords of point O (X,Y,Z values) : 0     SPECIFY COORDS IN WIPS-MODL
Boundary codes (free,hing,clmp,syxy,syyz,or syzx)
```

```
  OA : free
  BC : hing
  OB : clmp
  AC : syxy
*** error - edge must be in symmetry plane
  AC : syyz
Material set number : 1

Any errors? :

This set added to SLAB file

Specify a new property set? :

No. of property sets in SLAB file =  1

Display property set descriptions? :
Write in session log? : y

SLAB PROPERTY DESCRIPTIONS
  Set No.   Description
     1      Illustration, Fig.A7.3.1

Display new property data? :
Write in session log? :

Display all property data? : y

SLAB PROPERTY DATA

SET NO.  1.  Illustration, Fig.A7.3.1
                              First       Center         Last
                              Strip        Strip        Strip
  Widths along OA         0.9000e+01  0.9000e+01  0.1200e+02
  Widths along OB         0.1250e+02  0.7500e+01  0.5000e+01
  Elements along OA            2           4            2
  Elements along OB            3           3            1
  Thicknesses             0.6250e+00  0.6250e+00  0.6250e+00
  No. of Causs points          2           5            2

                               X           Y            Z
  Diraction cosines
     Edge OA              0.1000e+01  0.    e+00  0.    e+00
     Edge OB              0.    e+00  0.7071e+00  0.7071e+00
  Coordinates of O        0.    e+00  0.    e+00  0.    e+00

  Boundary codes OA,BC = free, hing
  Boundary codes OB,AC = clmp, syyz
Hit RETURN for next set
END OF DATA

New SLAB file created
Comment for file catalog : Illustration

End this SLAB session? : y
```

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
---------------------------------------------------------------------------
```



NOTE: Left, right, bottom and/or top strips may have zero width and number of divisions.

FIG. A7.3.1 - SLAB SUBSTRUCTURE

A8. WIPS-MODL USER GUIDE

SUMMARY

This section describes how an *analysis model* is created and presents instructions and examples on the use of the WIPS-MODL module.

CONTENTS

A8.1 GEOMETRICAL DEFINITION OF ANALYSIS MODEL

A8.1.1 ANALYSIS MODEL

An analysis model is a finite element representation of all or part of the piping system. The data defining the model contains information on the node coordinates and the locations and properties of the elements making up the model. This data is set up in a file (the MODL file) which consists essentially of input data "cards" for the WIPS-ANAL module. To set up an analysis model, it is necessary to use coordinate data from a COOR file, plus element and substructure property data from PIPE, BEAM, UBAR, GAPF, STRP, ELBO, and/or SLAB files.

In general, an analysis model can include parts of any number of pipe runs from the piping system, plus one or more SLAB substructures. It is necessary for the user to specify which parts of which runs; which PIPE, BEAM, UBAR, and GAPF elements and STRP and ELBO substructures are present in the pipe runs; and which SLAB substructures are to be included. This is done by a series of *commands*, each of which defines a single substructure or one or more elements. The commands are processed by WIPS-MODL to produce a MODL file. The MODL file is subsequently processed by WIPS-DATA to produce an input DATA file for WIPS-ANAL.

A8.1.2 SEGMENTS AND COMMANDS

Those parts of the analysis model which lie within pipe runs must be specified as a series of *segments*. A segment is any part of any pipe run, beginning and ending at a control point. A segment may comprise the entire pipe run, if desired. Typically, only a single segment will be used in any pipe run. However, several segments may be defined, if desired. The following rules must be observed.

(1) Each segment must lie entirely within a single pipe run.

(2) If STRP or ELBO substructures are used, each substructure must lie entirely within a segment.

(3) A pipe break must be the end point of a segment.

(4) The two halves of a double-ended break must not be the same Control Point (that is, separate control points must be defined in the WIPS-GEOM phase).

(5) Segments should not overlap. WIPS-MODL does *not* check for overlapping.

The first command for any segment specifies the control point (c.p.) at the beginning of the segment and the boundary conditions to be assigned to the corresponding node. The first command also specifies whether the segment can deform in 3D or whether it is constrained, because of symmetry, to move in a particular symmetry plane.

Each subsequent command specifies the structural elements to be used, either at a single node or over a length of the pipe run between two c.p.'s. This is done by specifying the c.p. at the end of the length, the type of element or substructure (PIPE, ELBO, etc.), and certain other information about the element. The commands must progress along the pipe run from beginning to end of the segment. A new command is needed each time the element properties change (i.e., *not necessarily at each control point)*. A blank command indicates the end of the segment.

A8.1.3 EXTERNAL SUBSTRUCTURES

After the last segment has been specified, "external" substructures, which do not lie in pipe runs, may be specified. At present, only SLAB substructures may be specified. The commands for specifying external substructures are similar to the commands for segments, except that no c.p. is specified.

A8.2 COMMAND FORMAT

A8.2.1 REQUIRED AND OPTIONAL DATA

The first two items in a command are *required*, as follows.

(1) First Item: The name of a c.p. in the pipe run. This may be either (a) the same as the c.p. in the preceding command, in which case the command defines an element of *single node* type (e.g. GAPF, UBAR), or (b) a c.p. further along the pipe run, in which case the command defines a substructure or series of elements of *two node* type (e.g. ELBO, PIPE).

(2) Second Item: The element or substructure type (UBAR, PIPE, ELBO, etc.).

The remaining items in a command (if any) are *optional* items. Each optional item is preceded by an option identifier, consisting of a 4-character word (or just the first two characters of the word) plus an "=" sign. Optional items may be specified in any convenient sequence. For most commands, default values will apply for most optional items, and no data needs to be specified.

The optional items for each element and substructure type are listed in Tables A8.1. Note that a ZERO element type is included, primarily to allow lumped weights and boundary conditions to be specified.

A8.2.2 EXTERNAL SUBSTRUCTURES

For an external substructure, the first item in the command must be the substructure type (SLAB). The remaining items are *optional*, as before.

A8.2.3 COMMAND LISTINGS

After a segment has been defined, the commands for the segment may be displayed. The display will show the values of all optional data items, including defaulted values. The display should be studied carefully to ensure that the commands are correct. If there are any errors, the segment must be redefined (i.e. there are no provisions for editing commands).

After all segments have been defined, the complete set of commands may be displayed and/or written in the session log. As a matter of sound practice, the commands should always be written to the session log, and a printed record obtained.

TABLE A8.2.1

ELEMENT AND SUBSTRUCTURE OPTIONS FOR WIPS-MODL

The following tables describe the data options for WIPS elements and substructures. It is important to note that the default options will usually apply, so that few options will need to be specified for most WIPS-MODL commands.

To specify optional data, enter the option name (either all four characters or just the first two characters), followed immediately by "=" (no blanks), followed immediately by the data. The data may be integer (I), real (R), or alpha (A), as shown in the DATA column for each table. The optional data may be entered in *any sequence*.

Two default columns are given in each table, namely DFLT.A and DFLT.B. The DFLT.A column applies if an element or substructure of the same type (PIPE, UBAR, STRP, etc.) has appeared in an earlier command. The DFLT.B column applies if this is the first element or substructure of its type.

A default of "same" means that the value defaults to that for the *nearest preceding* element or substructure of the same type (i.e. WIPS-MODL scans backwards over the commands until it finds a command with the same element or substructure type).

A default of "none" means that there is no default value, and the optional quantity *must* be specified.

A default of "auto" means that values are assigned automatically by WIPS-MODL. A default of "file" means that the value defaults to the value specified in the corresponding property set file. Explanatory NOTES follow the tables.

TABLE A8.2.1(a) - PIPE OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in PIPE file (I).	Same	None	(1)
BCON	Boundary code (I).			
	(a) 3D motion.	Zero	Zero	(2)
	(b) 2D motion.	Auto	Auto	(3)
LOCL	Local axis code (A).			
	(a) 3D motion (y axis): point K name or "AUTO"	Same	"AUTO"	(4)
	(b) 2D motion (z axis): "+" or "-"	Same	"+"	(5)
LDIS	Large displacement code (A): "YES" or "NO".	Same	"NO"	(6)
THIS	Time history code (A): "YES" or "NO".	Same	"YES"	(7)
WFAC	Self-weight multiplier (R).	Same	1.0	(8)
WDIS	Extra distributed weight per unit length (R).	Same	0.0	(9)
INTP	Internal pressure (R).	Same	0.0	(10)

TABLE A8.2.1(b) - BEAM OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in BEAM file (I).	Same	None	(1)
BCON	Boundary code (I). (a) 3D motion (b) 2D motion	 Zero Auto	 Zero Auto	 (2) (3)
LOCL	Local axis code (A). (a) 3D motion (y axis): point K name or "AUTO". (b) 2D motion (z axis): "+" or "-".	 Same Same	 "AUTO" "+"	 (4) (5)
LDIS	Large displacements code (A): "YES" or "NO"	Same	"NO"	(11)
THIS	Time history code (A): "YES" or "NO".	Same	"YES"	(7)
WFAC	Self-weight multiplier (R).	Same	1.0	(8)
WDIS	Extra distributed weight per unit length (R).	Same	0.0	(9)

TABLE A8.2.1(c) - UBAR OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in UBAR file (I).	Same	None	(1)
BCON	Boundary code (I). (a) 3D motion. (b) 2D motion.	 Zero Auto	 Zero Auto	 (2) (3)
JNOD	Point J name (A).	Blank	Blank	(12)
DXIJ	X projection of IJ (R).	0.0	0.0	(13)
DYIJ	Y projection of IJ (R).	0.0	0.0	(13)
DZIJ	Z projection of IJ (R).	0.0	0.0	(13)
GAP	Initial gap (R).	File	File	(14)
KNOD	Point K name (A).	Blank	Blank	(15)
LDIS	Large displacements code. (A): "YES" or "NO"	Same	"NO"	(16)
THIS	Time history code (A): "YES" or "NO"	Same	"YES"	(7)
LUMW	Lumped weight at node (R).	0.0	0.0	(17)

TABLE A8.2.1(d) - GAPF OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in GAPF file (I).	Same	None	(1)
BCON	Boundary code (I).			
	(a) 3D motion.	Zero	Zero	(2)
	(b) 2D motion.	Auto	Auto	(3)
DXIJ	X projection of IJ(R).	Same	Zero	(27)
DYIJ	Y projection of IJ(R).	Same	Zero	(27)
DZIJ	Z projection of IJ(R).	Same	Zero	(27)
GAP	Initial gap.	Same	None	
THIS	Time history code (A): "YES" or "NO".	Same	"YES"	(7)
LUMW	Lumped weight at node (R).	0.0	0.0	(17)

TABLE A8.2.1(e) - STRP OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in STRP file (I).	None	None	(1)
BCON	Boundary code (I).			
	(a) 3D motion	Zero	Zero	(2)
	(b) 2D motion	Auto	Auto	(3)
NAME	Substructure name in analysis model (A).	Auto	Auto	(18)
LOCL	Local axis code (A).			
	(a) 3D motion (y axis): Point K name.	None	None	(19)
	(b) 2D motion (z axis): "+" or "-".	"+"	"+"	(20)
LDIS	Large displacements code (A): "YES" or "NO".	"YES"	"YES"	(21)
THIS	Time history code (A): "YES" or "NO"	"YES"	"YES"	(22)
WFAC	Self-weight multiplier (R).	1.0	1.0	(8)
WDIS	Extra distributed weight per unit length (R).	0.0	0.0	(9)
INTP	Internal pressure (R).	0.0	0.0	(23)
SYMM	Longitudinal symmetry code (A): "YES" or "NO".	"NO"	"NO"	(24)

TABLE A8.2.1(f) - ELBO OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in ELBO file (I).	None	None	(1)
BCON	Boundary code (I). (a) 3D motion (b) 2D motion	 Zero Auto	 Zero Auto	 (2) (3)
NAME	Substructure name in analysis model (A).	Auto	Auto	(18)
LDIS	Large displacements code (A): "YES" or "NO".	"YES"	"YES"	(21)
THIS	Time history code (A): "YES" or "NO".	"YES"	"YES"	(22)
WFAC	Self-weight multiplier (R).	1.0	1.0	(8)
WDIS	Extra distributed weight per unit length (R).	0.0	0.0	(9)
INTP	Internal pressure (R).	0.0	0.0	(23)
SYMM	Longitudinal symmetry code (A): "YES" or "NO".	"NO"	"NO"	(25)

TABLE A8.2.1(g) - SLAB OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
PROP	Property set number in SLAB file (I).	None	None	(1)
NAME	Substructure name in analysis model (A).	Auto	Auto	(18)
LDIS	Large displacements code (A): "YES" or "NO".	Same	"YES"	(21)
THIS	Time history code (A): "YES" or "NO".	"YES"	"YES"	(22)
WFAC	Self-weight multiplier (R).	1.0	1.0	(8)
WDIS	Extra distributed weight per unit area (R).	0.0	0.0	(9)
XORG	X coordinate of point O(R).	File	File	(26)
YORG	Y coordinate of point O(R).	File	File	(26)
ZORG	Z coordinate of point O(R).	File	File	(26)

TABLE A8.2.1(b) - ZERO ELEMENT OPTIONS FOR WIPS-MODL

OPTION	DATA	DFLT.A	DFLT.B	NOTES
BCON	Boundary code (I).			
	(a) 3D motion	Zero	Zero	(2)
	(b) 2D motion	Auto	Auto	(3)
LUMW	Lumped Weight (R).	Zero	Zero	(17)

For a ZERO element, either specify "ZERO" for the element type or specify a blank type (i.e., c.p. name, followed by two successive commas, followed by optional data).

TABLE A8.2.1 - NOTES

(1) In each of the property set files (PIPE, BEAM, ELBO, etc.), the property sets are numbered in the order they are specified. For the first element of any type, PROP must be specified. For later elements, if PROP is omitted, it defaults to the property set number for the *nearest preceding element* of the same type.

(2) For 3D motion, the boundary code is a 6-digit number of ones and zeros. The digits correspond to X,Y,Z global displacements and X,Y,Z global rotations, respectively. A value of 0 for any digit indicates the corresponding displacement is free; a value of 1 that it is fixed. The default is free motion for all 6 displacements.

WARNING. If the BCON option is specified for a PIPE or BEAM element, and if the command defines several elements (and hence covers several nodes), the boundary code applies not only for the control point (c.p.) being specified but also for all nodes between the current c.p. and the c.p. in the preceding command. This may not be what is wanted. If a boundary code is to be defined for a single c.p., it is advisable to specify a ZERO element at the c.p., with the BCON option.

(3) For 2D motion, the boundary code defaults to motion in the symmetry plane only. For example, for motion in the XY plane, the default is 001110. If a code is specified and it is less restrictive than the default code for any displacement, it is automatically modified. For example, if a code 101010 is specified and motion is in the XY plane, it is modified automatically to 101110.

(4) The local coordinate system is defined as follows.

(a) Point I is the node at the beginning of the element, and Point J the node at the end.

(b) Line IJ defines the local x axis.

(c) Line IK defines a line (axis y') in the local xy plane. Note that IK must not be parallel to IJ.

(d) Axis y is constructed perpendicular to axis x, and hence, axis z is perpendicular to axes x and y.

If LOCL=AUTO, Point K is assigned automatically as follows.

(a) If IJ is not parallel to the global Y axis, Point K is assigned a very large +Y value.

(b) If IJ is parallel to the global Y axis, Point K is assigned a very large +X value.

(5) For 2D motion, the local element y axis lies in the plane of motion. The item to be specified is then the direction of the local z axis. For, say, motion in the global YZ plane, "LOCL=+" defines the local +z axis to be parallel to the global +X axis, whereas "LOCAL=-" defines the local +z axis to be parallel to the global -X axis.

(6) Small displacement analyses (LDIS=NO) are computationally more efficient than large displacement analyses (LDIS=YES). The assumption of small displacements will generally be reasonable for piping systems with restraints which prevent substantial deformation. For unrestrained pipes, it will usually be necessary to specify large displacements. For a PIPE element, the LDIS option controls only large displacements behavior of BEAM type (see NOTE 11). The PIPE element also includes an option to allow for cross section distortion (large ovalling). This option is controlled by the WIPS-PIPE input data, not by the LDIS option.

(7) If the time history code for any element is "YES", results for that element are saved in the RSLT file. If the code is "NO", results are not saved. The length of the RSLT file (and hence, data storage costs and some execution time) can be reduced by specifying "NO" for elements for which the response is not of interest. Note that nodal displacements, velocities and accelerations are automatically saved for all nodes so that deflected shapes can always be drawn from the RSLT file. The THIS option affects only element

response (stresses, strains, etc.).

(8) The self-weight is a part of the element property set and will typically include the weight of the steel element only. If desired, this weight may be increased or decreased, by specifying a WFAC multiplier larger or smaller than 1.0. Note that additional distributed weight can also be specified using the WDIS option, and additional lumped weights using the LUMW option (see ZERO element type).

(9) The WDIS option defines extra distributed weight (in addition to the element self-weight), for example, to allow for contained fluid and/or insulation. The weight and length units must be those specified at the beginning of the current WIPS-MODL session. Additional lumped weights can be specified using the LUMW option (see ZERO element type).

(10) The PIPE element includes an approximate theory for considering internal pressure effects. This theory allows for the effects of hoop and axial tension on yield of the element and for the stiffening effect of internal pressure on curved pipe elements (decreased ovalling). The pressure is assumed to be *constant* during the analysis. Hence, the value specified should be an estimate of the average internal pressure at the element during the response.

(11) The large displacements theory accounts for large rigid body *displacements* (i.e. rotations) of a BEAM element but assumes small *deformations* within the element. The small deformation assumption is accurate for elements which are short in length but not for very long elements.

Small displacement analyses (LDIS=NO) are computationally more efficient than large displacement analyses (LDIS=YES). The assumption of small displacements will typically be reasonable for piping systems with restraints which prevent substantial deformation. For unrestrained systems, or for supporting structures which may buckle or otherwise deform substantially, large displacements should be specified.

(12) The point on the pipe at which the U-bar provides restraint is Point I. Point J is a fixed point at which the U-bar is anchored. The line IJ defines the element axis (or, for large U-bar rotations, the initial position of this axis). If the length of the U-bar is not important (i.e. if large U-bar rotations will not occur), Point J may be any point along the element axis. Note that IJ may alternatively be specified by its X,Y,Z projections (see NOTE 13). Line IJ must be defined either by specifying Point J or by specifying the X,Y,Z projections, but not both. A blank name means that Point J is not specified.

(13) See NOTE 12 for definition of Points I and J. DXIJ, DYIJ, and DZIJ define the global X,Y,Z projections of IJ. If the length of IJ is not important, DXIJ, DYIJ, and DZIJ may be the direction cosines of IJ. Note that the default is zero. If IJ is along a global axis, only one projection needs to be specified.

(14) The GAP value is the initial U-bar gap. If GAP is not specified, it defaults to the value in the UBAR file for this property set (see WIPS-UBAR User Guide).

(15) See NOTE 12 for definition of Points I and J. Point K may be used to define the normal direction to the U-bar plane. If Point K is specified and LDIS=YES, the pipe (i.e. Point I) is allowed to move freely through the U-bar (i.e. in the axial direction of the pipe) until the gap is closed. After gap closure, axial displacement is restrained. The computational procedure is described in Section B6.

(16) If small displacements are specified (LDIS=NO), the axis of the U-bar element is assumed to remain parallel to its initial direction (see NOTE 12). This option will be suitable for cases where the pipe motion is essentially parallel to the U-bar. If large displacements are specified, the direction of the U-bar axis (and hence, the direction of the restraint force) is allowed to change during the analysis. This option must be used if the pipe motion is inclined to the U-bar axis. A large displacements analysis is more expensive computationally.

(17) Lumped weights may be specified to account for the extra weights of valves, flanges, or any pipe whip restraint hardware *attached to the pipe*. The WIPS U-bar and gap-friction elements are assumed to have zero mass (e.g. the weight of any saddles or other hardware attached to a U-bar is assumed to be negligible).

(18) Each substructure in the analysis model must be assigned a unique 4-character name. If no name is specified, successive substructures are automatically assigned the names S001, S002, etc. The substructure name is used for specifying impact surfaces (later in WIPS-MODL) and for results post-processing (in WIPS-RSLT).

(19) For 3D motion, the orientation of each STRP substructure in space must be defined using the "Point K" procedure (see NOTE 4). There is no default on this item. If impact is to be specified for any STRP substructure, particular care should be taken to ensure that the orientation is specified correctly.

(20) For 2D motion, the local xy axis of the substructure is automatically in the symmetry plane. The direction of the local z axis must, however, be defined. For, say, motion in the global YZ plane, "LOCL=+" defines the local +z axis to be parallel to the global +X axis, whereas "LOCL=-" defines the local +z axis to be parallel to the global -X axis.

(21) Large displacements analyses of substructures are much more expensive computationally than small displacements analyses. If the small displacements option is specified (LDIS=NO), yielding of the substructure is considered, but no change of shape. Because substructures will typically be specified only if impact or other local deformation effects are important, it will be usual to specify large displacements (LDIS=YES). Note that the substructure default is "LDIS=YES", whereas the element default is "LDIS=NO".

(22) If the time history code for any substructure is "YES", results for *all* shell elements in the substructure are saved in the RSLT file. If the code is "NO", no element results are saved.

(23) The shell element used to model substructures includes an approximate theory for considering internal pressure effects. This theory allows for the effects of hoop and axial tension on element yield and for the resistance which pressure offers to distortion of the pipe cross section. The pressure is assumed to be *constant* during the analysis. Hence, the value specified should be an estimate of the average internal pressure during the response.

(24) If "SYMM=YES" is specified, the STRP cross section at end J of the substructure is assumed to lie on a plane of symmetry, as described in the WIPS-STRP User Guide. The current segment is automatically assumed to end at this point. The cross section at end I of the substructure will be constrained to remain circular. However, the cross section at end J is *not* constrained in this way. Substructure boundary conditions are automatically constructed to confine the motion of end J to the symmetry plane but not to restrict cross section deformation. The SYMM=YES option can be used only if IJ is parallel to a global axis (X, Y, or Z).

(25) The "SYMM=YES" option for an ELBO substructure is intended to allow modeling of the type described in the WIPS-ELBO User Guide. If the length of "tangent 2" (see WIPS-ELBO User Guide) is zero, a single curved pipe is modeled. If this length is not zero, a pair of elbows separated by a tangent is modeled. The boundary conditions at end J of the substructure are as described in NOTE 24. The direction of "tangent 2" must be parallel to a global coordinate axis.

(26) If the XORG, YORG, and ZORG values are *all* blank, the coordinates of Point O are those specified for the SLAB property set in WIPS-SLAB. If one or more of XORG, YORG, and/or ZORG is nonzero, the coordinates specified in WIPS-SLAB are ignored.

(27) The point on the pipe at which the gap-friction element may provide restraint is Point I. Point J is any point along the normal from I to the restraint plane, *directed towards the restraint plane*. The values DXIJ, DYIJ, and DZIJ thus define a vector directed from Point I in the direction of gap closure. Note that this is opposite to the IJ direction for a U-bar element.

A8.3 IMPACT AND INITIAL VELOCITIES

A8.3.1 MODL FILE

After the segments and external substructures for the analysis model have been defined, a MODL file can be produced. For small models, the files will be short and will require only a few seconds of computer time to set up. For models involving substructures, the files may be long and may require up to a few minutes.

The node and element data defining the model is first set up. Then, if any substructures are present, surfaces for pipe-to-pipe or pipe-to-slab contact may be defined, and initial velocity patterns may be specified. WIPS-MODL requests data as described in the following sections.

A8.3.2 IMPACT DATA

A8.3.2.1 Primary and Secondary Surfaces

For surface-to-surface contact, one surface is the *primary* surface and one the *secondary*. Any number of contact surface *pairs* may be specified, each pair consisting of a primary surface and a secondary surface.

The primary surface must be a quadrilateral grid. This grid must be all or part of the finite element grid for a STRP, ELBO, or SLAB substructure. The secondary "surface" is not treated in the analysis as an actual surface but as a number of separate nodes. During the analysis, the geometrical relationship between the secondary surface nodes and primary surface grid is monitored, and the secondary surface is prevented from penetrating the primary surface. The theory is described in Section B1.2.

A8.3.2.2 Surface Pair Identification

Each surface pair must be named (e.g. SUR1). The name is used to identify the surface in the WIPS-RSLT phase.

A8.3.2.3 Primary Surface

The primary surface must be all or part of the finite element grid for a STRP, ELBO, or SLAB substructure. The term "grid" is used in this report to refer to the lines of nodes in a substructure mesh. The term "mesh" is used to refer to the rows of elements.

In the definition of model segments and external substructures, each substructure is assigned a name (either by use of the NAME option or by defaulting to S001, S002, etc.). The primary surface substructure must be named. WIPS-MODL displays the allowable grid ranges in the circumferential (or, for a SLAB, the OA) and longitudinal (or OB) directions. The grid ranges to be used must be specified by the WIPS-MODL user. The specified grid ranges should cover enough of the surface to ensure that all contact occurs within the specified area. Each range is defined by a beginning and ending grid location.

For STRP and ELBO substructures, the circumferential grid range must be specified, moving in a clockwise direction when viewed along the pipe axis. If the range covers grid line number 1, the beginning grid location will be numerically larger than the ending location. In all other cases, the ending location must be the larger.

A8.3.2.4 Secondary Surface

The secondary surface may be either another substructure or a series of nodes defining pipe or beam elements along a pipe run. In this second case, the secondary "substructure" is the "main" structure.

If the secondary surface is a substructure, the grid range must be specified as for the primary surface. If the secondary surface is the main structure, one or more ranges of nodes must be specified. For each range, the run number, beginning c.p. and ending c.p., must be specified. All nodes in the range (including the end c.p.'s) become nodes of the secondary surface.

A8.3.2.5 Surface Thicknesses

The coordinates of nodes along a pipe run define the pipe centerline, and the substructure coordinates stored for SLAB substructures define the slab midsurface. To account for finite pipe dimensions and slab thicknesses, adjustments must be made in the impact analysis. This is done, in WIPS-ANAL, by calculating *nominal clearances* (in effect the node-to-node distances) between the surfaces, and subtracting off *surface half-thicknesses* to obtain *actual clearances*. For a node in the main structure, the half-thickness is the pipe radius, and for a SLAB substructure it is one-half of the slab thickness. For STRP and ELBO substructures, the node locations are assumed to be at the pipe wall midthickness for the purpose of calculating nominal clearances. Hence, the surface half-thickness is one-half the pipe wall thickness.

Surface half-thicknesses must be specified for both the primary and secondary surfaces. The default value is zero (i.e. WIPS-MODL does *not* look at the dimensions of the elements or substructures to obtain half-thickness values).

A8.3.2.6 Friction Coefficient

If friction effects are likely to be important, a surface-to-surface friction coefficient may be specified. In addition, a sliding velocity tolerance must be defined. The purpose of this tolerance is as follows.

In the impact analysis, friction forces are applied in the direction opposing the current sliding velocity. If the sliding velocity is small, it may fluctuate from positive to negative in successive time steps, with the result that the friction forces can suddenly change direction. Such sudden changes can lead to instability in the solution scheme. To avoid this, the friction coefficient is assumed to attain its full value only if the sliding velocity exceeds a certain tolerance. For sliding velocities less than this tolerance, the friction coefficient is assumed to reduce and to be zero for zero velocity. A default tolerance of 50 in/sec. is built into WIPS-MODL. A different value may be specified if desired.

A8.3.3 INITIAL VELOCITIES

A8.3.3.1 Purpose

In certain cases the motion of the ruptured pipe up to impact, and the impact velocity, will be known, and the primary purpose of the analysis will be to study the behavior following impact. In such cases, it may be possible to define the initial geometry of the system as that just before impact and to specify a pattern of initial velocities. It is then not necessary to perform any analysis for the motion between rupture and impact, and substantial computer time may be saved.

WIPS-MODL includes an option for specifying initial velocities. The option is necessarily limited in scope, however. In particular, it is assumed that the pipe moves as a rigid body before impact (for example, as a length of pipe pivoting about a known hinge, with negligible deformation before impact).

A8.3.3.2 Procedure

Initial velocities may be specified for one or more *segments* of pipe (as defined in Section A8.1.2). The initial motion of a segment may be purely translational or may involve pivoting about a known axis. WIPS-MODL requests the segment number and the name of the c.p. about which the segment pivots. If no pivot is named, the segment initially moves as a rigid body with only translational velocity, and the global X,Y,Z velocities must be specified. If a pivot point is named, the pivot axis and rigid body rotational velocity must be specified. The pivot axis must be one of the global X,Y,Z axes. WIPS-MODL automatically calculates the initial velocities for all nodes in the segment, including the nodes in STRP and ELBO substructures.

The initial velocity option must be used very cautiously. Its main purpose is to permit analyses of experimental studies in which a length of pipe is swung about a pivot until it strikes a target pipe, or in which a length of pipe is propelled as a projectile to strike the containment wall or another pipe.

A8.4 EXAMPLES

A8.4.1 RESTRAINED SYSTEM

Pipe run number 2 in Fig. A4.2.1 extends from a branch point at control point B2 to an anchor at control point A3. Table A8.4.1 is a listing of the WIPS-MODL session log to set up an analysis model for this run. The model has a break at B2, a U-bar restraint at T4B, and assumed full fixity at branch point B3. The part of the pipe run from B3 to A3 is not included in the model.

The pipe run lies in the YZ plane and moves only in this plane. Hence, a symmetric condition is specified for the analysis model, with symmetry about the YZ plane. When such a symmetric condition is specified for any segment, the analysis model contains only one-half of the structure in the segment (i.e. one-half of the pipe circumference and one-half of the U-bar in this example). This allows full 3D motion to be specified for some segments of an analysis model, and only 2D motion (i.e. symmetry) for other segments. The WIPS-ANAL analysis also requires less execution time, because a 2D structure has fewer degrees of freedom (node displacements) than a 3D structure, and because some *pipe* element computations can be omitted. It is important to note, however, that if jet forces are specified for control points in a 2D segment (in WIPS-DATA), these forces must be multiplied by 0.5 to account for the symmetry.

A8.4.2 UNRESTRAINED SYSTEM

Table A8.4.2 shows the WIPS-MODL commands to set up an unrestrained analysis model consisting of pipe runs 2 and 3 in Fig. A4.2.1. A break is assumed at A3, and full fixity at control points B1 and B2. There are no pipe whip restraints, and large displacements are specified. Full 3D motion is allowed.

A8.4.3 SYSTEM WITH SUBSTRUCTURES AND IMPACT

A series of experiments involving pipe-to-pipe impact have been conducted at Battelle Northwest Laboratories. The test configuration is illustrated in Fig. A8.4.1. Table A8.4.3 shows the session log (GEOM, MATL, STRP, PIPE, and MODL phases) to construct an analysis model for a configuration of this type. Figures A8.4.2 and A8.4.3 show the finite element mesh which is constructed. Note that transverse symmetry is specified for the swinging pipe, and longitudinal symmetry for the target pipe, to reduce the size of the finite element mesh. The geometry (in WIPS-GEOM) is specified such that there is a small initial clearance between the pipes, and initial velocities are specified (in WIPS-MODL).

This example is for illustration only, and coarse finite element meshes have been specified to avoid cluttering the figures. For an actual analysis, it would be necessary to specify much finer meshes.

TABLE A8.4.1

WIPS-MODL EXAMPLE

RESTRAINED MODEL. RUN 2 FROM FIG. A4.2.1

```
EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : modl


MODL - SPECIFICATION OF ANALYSIS MODEL

Define units
  Length (ft, in, m, mm) : in
  Force (k, lb, kgf, kN) : k

Sequence no. of COOR file : 1

Available element property sets
  Type     No. of Sets
  pipe         2
  ubar         1
Available substructure property sets
  Type     No  of Sets
  elbo         1

DEFINE MODEL IN SEGMENTS

SEGMENT NO.   1

Pipe run no. : 2
First c.p. of segment : b2
Full 3D motion ? : no                      RESTRICT TO Y-Z PLANE
Displacement plane (xy, yz or zx) : yz
Any substructures in this segment? :
Boundary condition code for first c.p. :
Symmetric - boundary code reset to 100011

Specify c.p. name + elem type + optional data
 1: t4b pipe pr=2                          REMAINING OPTIONS DEFAULT
 2: t4b ubar pr=1 gap=.4
*** error - direction not defined - use 'jnod' or 'd*ij'
 2: t4b ubar pr=1 gap=.4 dy=1              U-BAR IN +Y DIRECTION
 3: b3 pipe                                ALL OPTIONS DEFAULT
 4: b3,,bc=111111
 5:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE
```

```
SEGMENT NO.   1.   Run No. =  2
  c.p.   elem     optn      data          optn      data          optn      data
  b2
                  bcon    100011
  t4b    pipe
                  prop         2
                  locl         +          ldis       no           this       yes
                  wfac    1.0000
  t4b    ubar
                  prop         1
                  ldis        no          this      yes
                  dyij    1.0000          gap    0.4000
  b3     pipe
                  prop         2
                  locl         +          ldis       no           this       yes
                  wfac    1.0000
Hit RETURN for more
  b3
                  bcon    111111
Any changes? : yes                                  ADD LUMPED WEIGHT AT t4b
 Re-enter data for this segment

SEGMENT NO.    1

Pipe run no. : 2
First c.p. of segment : b2
Full 3D motion ? : n
Displacement plane (xy,yz or zx) : yz
Any substructures in this segment? :
Boundary condition code for first c.p. :
Symmetric - boundary code reset to 100011

Specify c.p.name + elem type + optional data
 1: t4b pipe pr=2
 2: t4b ubar pr=1 gap=.4 dy=1 lumw=.02
 3: b3 pipe
 4: b3 zero bc=111111
 5:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO.   1.   Run No. =  2
  c.p.   elem     optn      data          optn      data          optn      data
  b2
                  bcon    100011
  t4b    pipe
                  prop         2
                  locl         +          ldis       no           this       yes
                  wfac    1.0000
  t4b    ubar
                  prop         1
                  ldis        no          this      yes
                  lumw    0.2000e-01      dyij   1.0000           gap     0.4000
```

```
  b3    pipe
                 prop        2
                 locl        +          ldis       no         this       yes
                 wfac    1.0000
Hit RETURN for more
  b3    zero
                 bcon    111111
Any changes? :

SEGMENT NO.    2

Pipe run no. :
Last segment? : y

Any external substructures? :

Display commands for all segments? :
Write in session log? :

Produce MODL file? : y
Wait while data is processed

Specify initial velocities ? :
Comment for file catalog : Run2, b2 (break) to b3 (anchor).

MODL file saved. File name = MODL0101
End of this MODL session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

TABLE A8.4.2

WIPS-MODL EXAMPLE

UNRESTRAINED MODEL. RUNS 2,3 FROM FIG. A4.2.1

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : modl

MODL - SPECIFICATION OF ANALYSIS MODEL

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Sequence no. of COOR file : 1

Available element property sets
  Type     No. of Sets
  pipe         2
  ubar         1
Available substructure property sets
  Type     No. of Sets
  elbo         1

DEFINE MODEL IN SEGMENTS

SEGMENT NO     1

Pipe run no. : 2
First c.p. of segment : b2
Full 3D motion ? : y
Boundary condition code for first c.p. : 111111

Specify c.p.name + elem type + optional data
 1: a3 pipe pr=2 ld=y intp=1.2
 2:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO    1.   Run No. = 2
  c.p.  elem    optn      data        optn      data        optn      data
  b2
                bcon   111111
  a3    pipe
                prop         2
                locl      auto        ldis      yes         this      yes
```

```
               wfac    1.0000        intp     1.200
Any changes? :

SEGMENT NO.    2

Pipe run no. : 3
First c.p. of segment : b1
Full 3D motion ? : y
Boundary condition code for first c.p. : 11111

Specify c.p.name + elem type + optional data
 1: b3 pipe                         ALL OTHER DATA DEFAULTS
 2:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO.  2.  Run No. =  3
  c.p.  elem    optn      data        optn      data        optn      data
  b1
                bcon     11111
  b3    pipe
                prop         2
                locl      auto        ldis      yes         this      yes
                wfac    1.0000        intp    1.200
Any changes? : y              INCORRECT BCON CODE FOR b1
 Re-enter data for this segment

SEGMENT NO.    2

Pipe run no. : 3
First c.p. of segment : b1
Full 3D motion ? : y
Boundary condition code for first c.p. : 111111

Specify c.p.name + elem type + optional data
 1: b3 pipe
 2:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO.  2.  Run No. =  3
  c.p.  elem    optn      data        optn      data        optn      data
  b1
                bcon    111111
  b3    pipe
                prop         2
                locl      auto        ldis      yes         this      yes
                wfac    1.0000        intp    1.200
Any changes? :

SEGMENT NO.    3
```

```
Pipe run no. :
Last segment? : y

Any external substructures? :

Display commands for all segments? :
Write in session log? : y

MODL COMMAND TABLE

SEGMENT NO.  1.  Run No. =  2
  c.p.   elem    optn     data        optn      data        optn      data
  b2
                 bcon   111111
  a3     pipe
                 prop        2
                 locl     auto        ldis       yes        this       yes
                 wfac   1.0000        intp     1.200
SEGMENT NO.  2.  Run No. =  3
  c.p.   elem    optn     data        optn      data        optn      data
  b1
                 bcon   111111
  b3     pipe
                 prop        2
                 locl     auto        ldis       yes        this       yes
                 wfac   1.0000        intp     1.200

Produce MODL file? : y
Wait while data is processed

Specify initial velocities ? :
Comment for file catalog : Runs 2 and 3, unrestrained, break at a3.

MODL file saved. File name = MODL0102
End of this MODL session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

TABLE A8.4.3

WIPS-MODL EXAMPLE

MODEL WITH SUBSTRUCTURES. COARSE MESH MODEL OF BATTELLE TEST.

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : modl


MODL - SPECIFICATION OF ANALYSIS MODEL

Define units
  Length (ft, in, m, mm) : in
  Force (k, lb, kgf, kN) : k

Sequence no. of COOR file : 1

Available element property sets
  Type    No. of Sets
  pipe        1
Available substructure property sets
  Type    No. of Sets
  strp        2

DEFINE MODEL IN SEGMENTS

SEGMENT NO.   1

Pipe run no. : 1
First c.p. of segment : s1
Full 3D motion ? : n
Displacement plane (xy, yz or zx) : xy
Any substructures in this segment? : y
They must lie on +z or -z side of plane
Specify side (+ or -) : +
Boundary condition code for first c.p. : 111000
Symmetric - boundary code reset to 111110

Specify c.p. name + elem type + optional data
 1: s2 pipe pr=1 ld=y
 2: s3 strp pr=1
 3: s4 pipe
 4:
End of segment? : y
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO.  1.  Run No. =  1
  c.p.  elem    optn     data        optn     data        optn      data
```

```
  s1
                 bcon    111110
  s2    pipe
                 prop         1
                 locl         +          ldis       yes        this       yes
                 wfac    1.0000
  s3    strp
                 prop         1
                 name      s001          ldis       yes        this       yes
                 wfac    1.0000
  s4    pipe
                 prop         1
                 locl         +          ldis       yes        this       yes
                 wfac    1.0000
Hit RETURN for more
Any changes? :

SEGMENT NO.   2

Pipe run no. : 2
First c.p. of segment : t1
Full 3D motion ? : y
Boundary condition code for first c.p. : 111001

Specify c.p.name + elem type + optional data
 1: t2 pipe
*** error - must use "locl" to define local y axis
 1: t2 pipe lo=rf1
 2: t3 strp pr=2 sy=y lo=rf1
 Symmetric condition. Automatic segment end
Display commands for this segment? : y

MODL COMMAND TABLE

SEGMENT NO.   2.  Run No. =  2
  c.p.  elem    optn      data       optn      data       optn      data
  t1
                bcon    111001
  t2    pipe
                prop         1
                locl       rf1       ldis      yes        this      yes
                wfac    1.0000
  t3    strp
                prop         2
                name      s002       ldis      yes        this      yes
                locl       rf1       symm      yes
                wfac    1.0000
Any changes? :

SEGMENT NO.   3

Pipe run no. :
Last segment? : y

Any external substructures? :
```

```
Display commands for all segments? :
Write in session log? :

Produce MODL file? : y
Wait while data is processed

Impact analysis required ? : y
No. of impact surface pairs : 1

SURFACE PAIR NO.  1
Surface pair name : sur1

Define primary surface
  Substructure name : s002
STRP substructure, property set no.  2
  Circumf. divisions = 12
  Longitl. divisions =  6  (  2,  2,  2)
Full 3D motion
Allowable longitudinal grid range = 1 thru  7
Define grid ranges
  Circumferential : 10 4
  Longitudinal : 3 7
Thickness from nodes to surface : .14

Define secondary nodes
  Substructure name (dflt = main) : s001
STRP substructure, property set no.  1
  Circumf. divisions = 12
  Longitl. divisions = 10  (  3,  4,  3)
Only second circumferential half of mesh used
Allowable circumferential grid range = 7 thru  1
Allowable longitudinal grid range = 1 thru 11
Define grid ranges
  Circumferential : 7 10
  Longitudinal : 4 8
Thickness from nodes to surface : .14
Friction coefficient (dflt=zero) :
Sliding veloc. for full friction (dflt=50in/sec) :
Any errors ? :

Specify initial velocities ? : y
Specify initial velocities for affected segments
Warning - only partial consistency check is performed on data
  SEGMENT NO. : 1
   Segment has transverse symmetry
  Motion in xy plane only
  Name of pivot point (dflt = no pivot) : s1
  Pivot axis = global z
  Angular velocity (rad/sec, r.h. rule) : 14
  SEGMENT NO. :
  Last affected segment ? : y
   Wait while data is processed

Comment for file catalog : Example. Battelle type configuration.

MODL file saved. File name = MODL0101
```

```
End of this MODL session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit

----- -- ----- -- -- ----- ----- ------------------------------
```

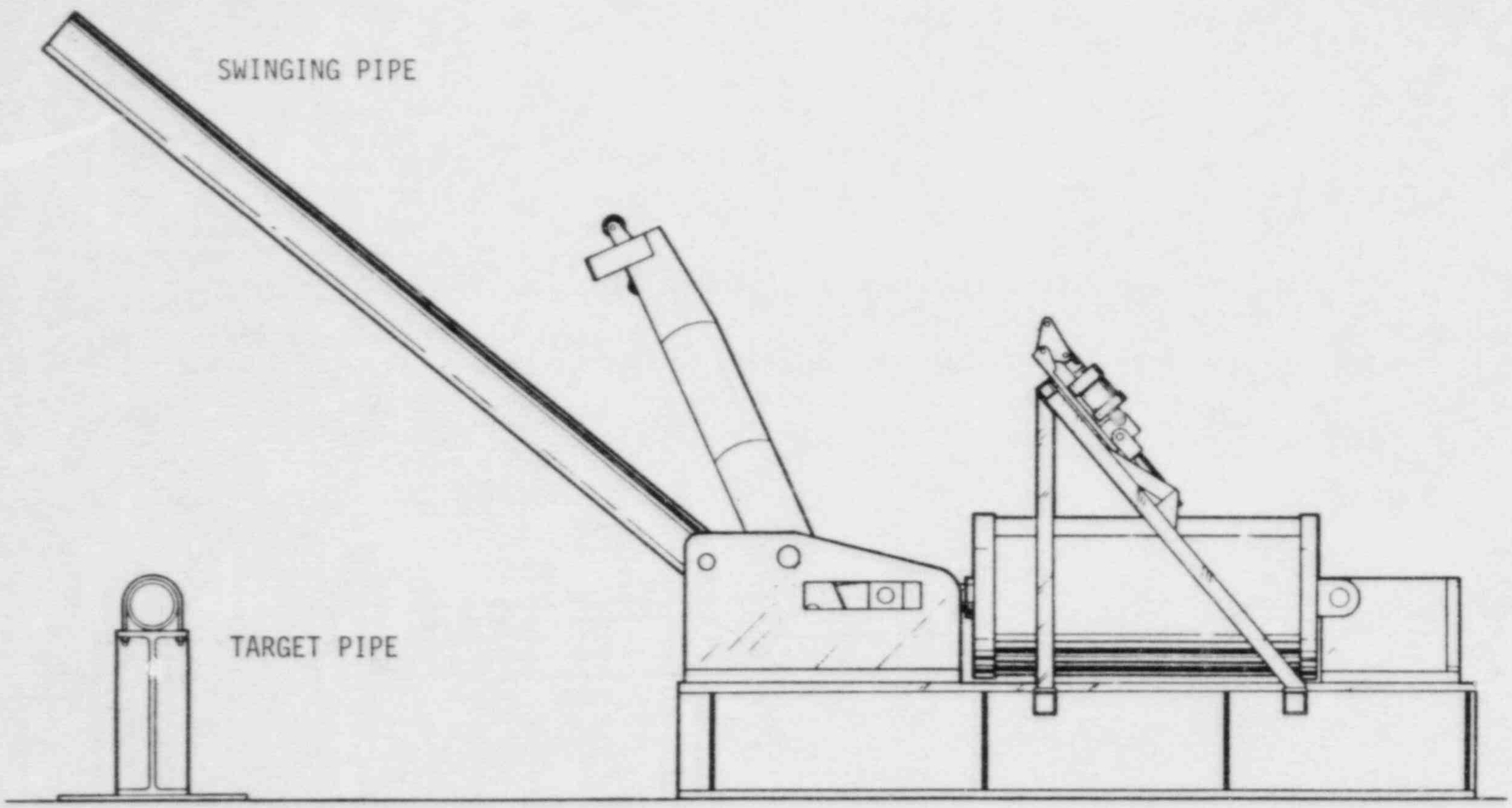
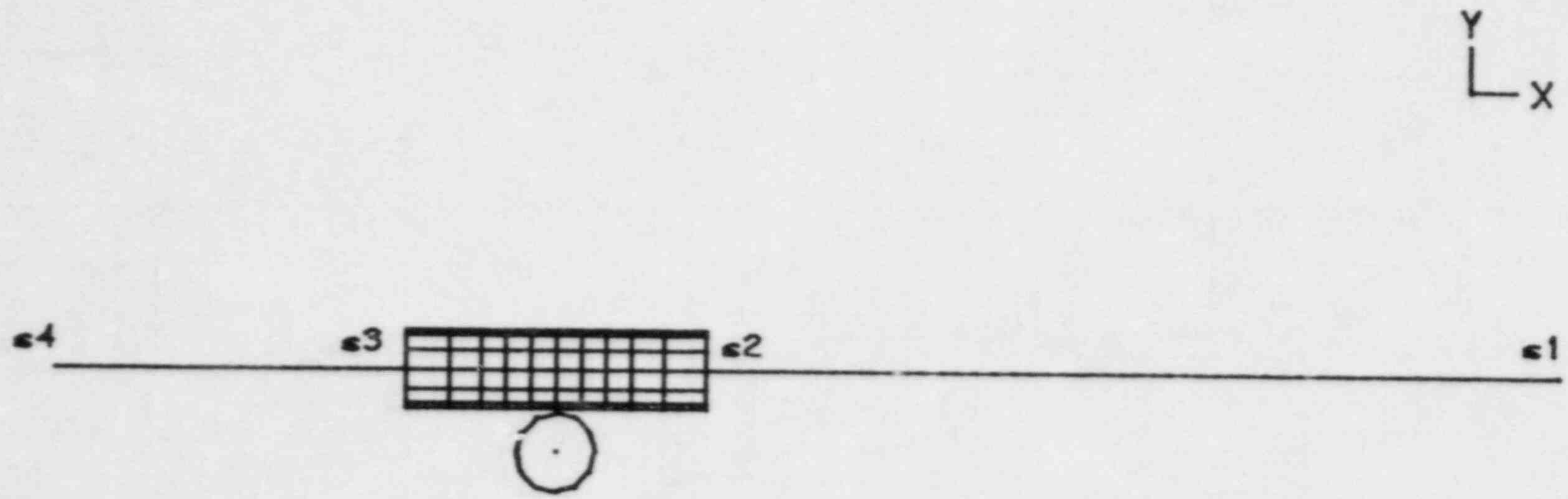


FIG. A8.4.1 - TEST EQUIPMENT (COURTESY OF BATTELLE NORTHWEST LABORATORIES)



Notes:

(1) Geometry is specified so that pipes have small clearance at time zero. Initial velocity option is used.

(2) The finite element mesh in this example is coarser than would be needed for an accurate analysis.

FIG. A8.4.2 - WIPS ANALYSIS MODEL

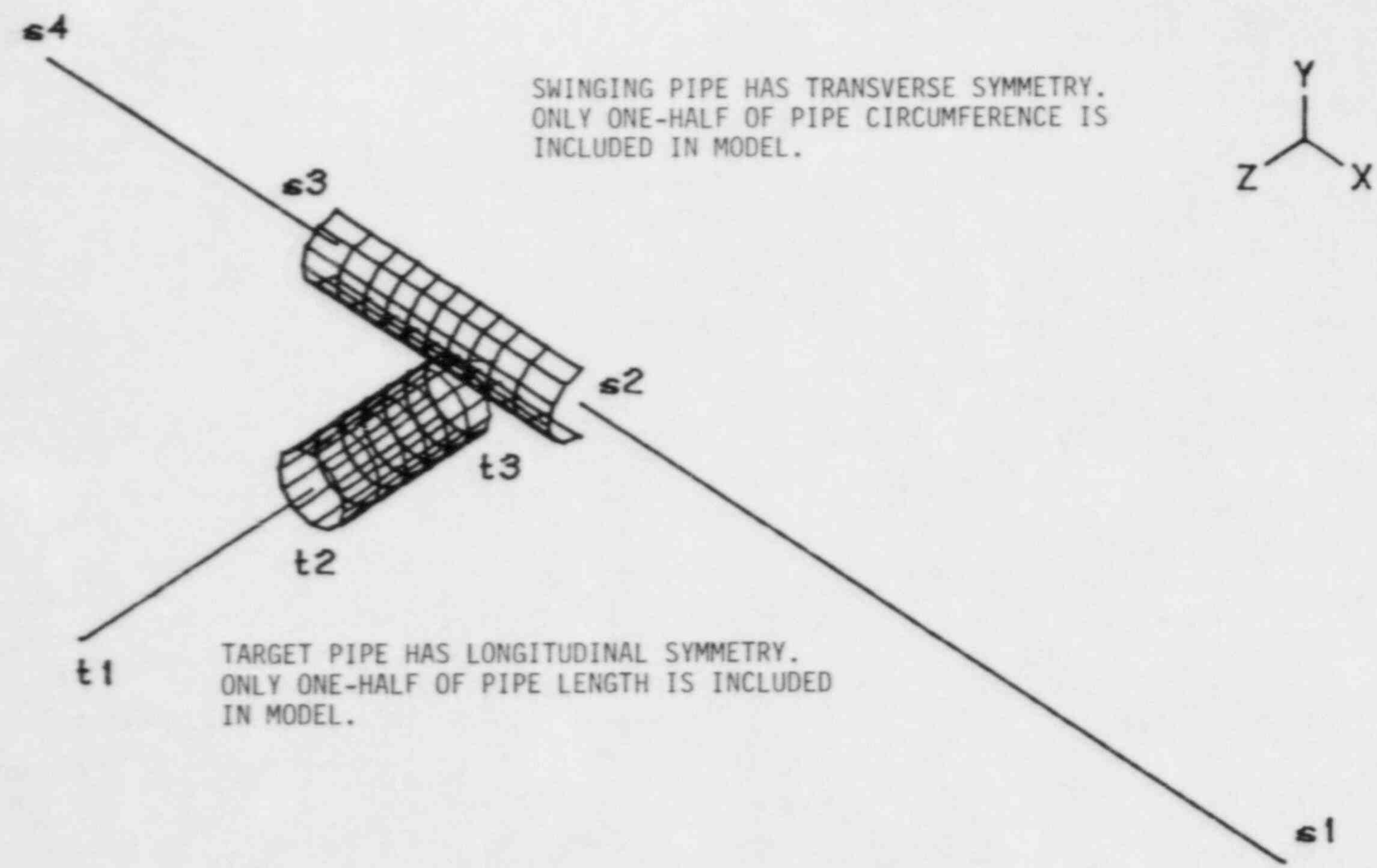


FIG. A8.4.3 - WIPS ANALYSIS MODEL. INCLINED VIEW.

A9. WIPS-DATA USER GUIDE

SUMMARY

WIPS-DATA takes the data in a MODL file and modifies it to produce a DATA file for WIPS-ANAL. WIPS-DATA also initializes ECHO, SLOG, RSLT, PAUS, and PAUZ files which make up the DATA file set. This section describes the procedure for using WIPS-DATA.

In addition, this section contains a chapter on the execution of WIPS-ANAL. Because WIPS-ANAL execution is system-dependent, this chapter must be supplemented by additional information for each different operating system.

CONTENTS

A9.1 WIPS-DATA USER GUIDE

A9.1.1 PURPOSE

A MODL file containing the geometric and mechanical data for an analysis model is produced in the WIPS-MODL phase. In one mode of operation (the *original data* mode), WIPS-DATA extends the MODL file to include information on loading, time step size, etc. and creates a DATA file. The DATA file becomes the input file for the WIPS-ANAL phase. At the same time, WIPS-DATA initializes ECHO, SLOG, RSLT, and (if requested) PAUS and PAUZ files for use by WIPS-ANAL.

If the DATA file instructs WIPS-ANAL to pause at the end of the execution, PAUS and PAUZ files are set up to allow restart. In a second mode of operation (the *restart* mode), WIPS-DATA modifies an existing DATA file for use in a restart run by WIPS-ANAL. In this mode, WIPS-DATA can also be used to purge the ECHO, SLOG, and/or RSLT files, to reduce their size and save disk storage.

A9.1.2 PROCESSING MODE

If any PAUSE files (i.e pairs of PAUS and PAUZ files) exist for the current problem, WIPS-DATA first asks whether existing PAUSE files are to be modified. If the answer is YES, the sequence number of the DATA file set is requested, and WIPS-DATA enters the restart mode. If the answer is NO, or if there are no PAUSE files, WIPS-DATA requests the sequence number of a MODL file and enters the original data mode.

A9.1.3 ORIGINAL DATA MODE

A9.1.3.1 Initial Data

In the original data mode, the following data is requested after the MODL file sequence number.

(1) Problem description: 4 lines, up to 60 characters per line, describing the problem to be solved.

(2) Whether or not PAUSE files are to be created. If the answer is YES, a pair of PAUSE files is set up at the end of the subsequent WIPS-ANAL execution, to allow later restart. If the answer is NO, no PAUSE files are set up (that is, it is assumed that the analysis will be completed in a single WIPS-ANAL run).

WIPS-DATA then asks the user to wait while the data on the MODL file is processed. For a small problem, this will require only a few seconds. For a large problem, it may require a minute or two.

A9.1.3.2 Initial Velocity Scaling

During processing of the MODL file, any initial velocities specified in the WIPS-MODL phase may be scaled. If initial velocity data are present on the MODL file, WIPS-DATA requests a velocity scale factor, with a default value of 1.0. All initial velocities are scaled by this factor.

A9.1.3.3 Load Data

After initial processing of the data, information on the loading is requested. A force acting in a particular direction at a point on the analysis model constitutes one *load*. Any number of loads may be specified. The following data is requested for each load:

(1) Name of force record (in FREC file). A blank response ends the load data.

(2) Name of the control point (c.p.) at which the force acts. This must be a c.p. in the main structure. It is not possible to specify loads at nodes within a substructure.

(3) Direction of force. The options are (global) X, Y, or Z, or FOLLOWER (which may be abbreviated to FOLL).

(4) For the FOLL option, the name of a second c.p. (the follower c.p.) to define the direction of the force. This must be a c.p. which is a *node in the main structure*. The force then acts at the loaded c.p. and is always directed towards the follower c.p. This permits the direction of a jet force to be varied if the piping system deforms substantially. The option is also convenient for specifying pressure pulse effects, as described in Section A9.1.3.4.

(5) Scale factor, by which the forces in the specified force record are to be multiplied. This factor also controls the direction of the force. For a force in the X direction, a positive scale factor means a force in the +X global direction and a negative factor a force in the -X direction. Similarly, for the FOLL option, a positive factor means that the force acts towards the follower c.p. and a negative factor that it acts away from the follower c.p.

(6) Time delay, which will typically be zero. This permits a traveling pulse load to be modeled, as described in Section A9.1.3.4.

A9.1.3.4 Traveling Pulse

In most cases only a jet force will be considered, and a force will be applied only at the pipe break. It is possible, however, to model (approximately) the influence of a traveling pressure pulse (i.e. water hammer effect).

As a pressure pulse passes around a pipe bend, it exerts an essentially radial force at the bend. Because of the pulse travel time between bends, significant vibrations may be excited in the piping. Also, because of the possible large magnitude of the pressure load, large dynamic forces may be generated in piping anchors and supports.

The effect of a pressure pulse in a bend can be represented approximately by a pair of dynamically applied forces acting parallel to the tangents at the tangent points of the bend. In the data of Section A9.1.3.3 the forces can be specified as follows.

(1) Specify two loads for each bend (one for each tangent point).

(2) Specify the FOLL option. The loaded c.p. is the loaded tangent point, and the follower c.p. is any c.p. along the adjacent tangent (typically the next tangent point). Note that the tangent intersection point *cannot* be used as the follower c.p., because it is not a node in the analysis model.

(3) Specify *negative* scale factors, so that the forces act outwards on the bends. If desired, attenuation of the pulse can be modeled by specifying a progressively smaller factor for each successive bend.

(4) Specify a time delay for each force equal to the time taken for the pulse to arrive at the bend.

If the water hammer loading involves both initial and reflected pulses, it will be necessary to specify a different force record for each pulse. Time delays can then be specified so that the pulses arrive at the bends at appropriate times.

A9.1.3.5 Time Step Data

Following the load data, the following time step data is requested.

(1) Initial time step. This is the time step used at the beginning of the WIPS-ANAL analysis. As the analysis proceeds, the time step is automatically varied and may increase or decrease.

(2) Maximum time step. The time step is not permitted to be increased above this value.

(3) Minimum time step. The time step is not permitted to be reduced below this value.

Typical time steps for a pipe whip analysis are as follows:

(1) Initial: 0.0005 seconds.

(2) Maximum: 0.001 seconds.

(3) Minimum: 0.00003 seconds.

The minimum value is likely to be reached only during short periods following very stiff impact. To enforce a uniform time step, specify maximum and minimum values slightly larger and smaller, respectively, than the initial value.

A9.1.3.6 Termination Limits

Limits to use for termination of the WIPS-ANAL execution are requested as follows.

(1) Maximum number of time steps.

(2) Maximum time.

When the number of time steps completed in the analysis equals the maximum number, or when the sum of all completed time steps exceeds the maximum time (whichever occurs first), WIPS-ANAL execution ceases. Note that because the average time step is not known in advance, whereas the duration of the pipe whip event is usually known approximately, it will be usual to control the termination by the maximum time. However, to avoid waste of computer time if WIPS-ANAL should "hang up" with a large number of very small time steps, it is wise to specify a reasonable limit on the number of steps.

A9.1.3.7 Error Tolerances

During a WIPS-ANAL analysis, the time step is automatically increased and decreased, depending on the value of the "midstep equilibrium error". The concept is that if this error never exceeds a specified tolerance, the computed solution will never violate dynamic equilibrium excessively, and hence, will be accurate. The term "accurate" is, however, relative. If a large tolerance is specified, substantial equilibrium violations are permitted and accuracy will suffer. If a small tolerance is specified, smaller time steps will be required to satisfy the tolerance, and execution times will increase.

A certain amount of experience and judgement is needed to select an appropriate tolerance. If experience is lacking, a value equal to 25% of the average jet force is suggested. If no jet force is used (i.e. in an analysis with initial velocities), estimate the jet force required to produce the initial velocities.

WIPS-DATA requests two tolerances (upper and lower). The upper tolerance is the value described above. If the midstep error exceeds this tolerance in any step, the time step is reduced. The lower tolerance should be specified to be approximately 20% of the upper tolerance. If the midstep error is below the lower tolerance for two successive steps, the time step is increased.

A9.1.3.8 Results Output Intervals

At specified intervals during a WIPS-ANAL analysis, results are output to the RSLT file. This output interval may be controlled in two ways, as follows.

(1) Maximum number of time steps since last output.

(2) Maximum time since last output.

WIPS-DATA requests both values. During the WIPS-ANAL analysis, results are saved whenever *either* the specified number of steps is exceeded *or* the specified time, whichever occurs first.

A9.1.3.9 Integration Scheme

The WIPS user must select one of two possible integration schemes, namely Newmark with viscous damping or Hilber-Hughes-Taylor (HHT) with numerical energy dissipation. The default is the Newmark scheme. Energy dissipation is essential to avoid instability in the numerical scheme. However, the amount of dissipation must not be so large that the accuracy of the results is affected.

A9.1.3.10 Energy Dissipation: Newmark Scheme

For the Newmark scheme, energy dissipation is obtained by introducing viscous damping. The amount of dissipation is controlled by a dimensionless "damping factor", with a default value of 0.1. A larger factor introduces greater dissipation. The default value is recommended.

The theory of the integration scheme is described in Section B1. The damping matrix for the scheme is $\beta_o \underline{K}_o$, in which $\underline{K}_o$ = initial (elastic) stiffness matrix and β_o is a coefficient with the dimension of time. The dimensionless damping factor defined in WIPS-DATA is $\beta_o/\Delta t$, where Δt is the initial time step.

A9.1.3.11 Energy Dissipation: HHT Scheme

For the HHT scheme, energy dissipation is an automatic part of the numerical scheme. The amount of dissipation is again controlled by a dimensionless damping factor, with a default value of 0.1. Again, a larger factor introduces greater dissipation, and the default value is recommended.

The theory of the integration scheme is described in Section B1. The damping factor is equal to minus the HHT "α" factor.

A9.1.4 RESTART MODE

A9.1.4.1 General

In the restart mode, an existing DATA file is modified so that WIPS-ANAL restarts from the point at which it previously stopped. This point is defined by the data on the PAUSE files. An analysis may be stopped and restarted as many times as desired. Before each restart, WIPS-DATA *must* be called to modify the DATA file.

A9.1.4.2 File Length Reduction

In restart mode, WIPS-DATA allows the ECHO, SLOG, and RSLT files to be "purged" by removing any data written on these files during preceding WIPS-ANAL runs. In general, the data on the ECHO and SLOG files will be used only to ensure that WIPS-ANAL has executed properly and need not be retained. Data storage costs can thus be saved by purging these files. It will be usual, however, to retain the data on the RSLT file so that this file contains the full time history of response, beginning at zero time. In some cases, however, the WIPS-DATA user may choose to purge the RSLT file also.

A9.1.4.3 Data Sequence

The following data is requested in restart mode.

(1) Sequence number of the DATA file set.

(2) Whether or not the ECHO, SLOG, and/or RSLT files are to be purged.

(3) New maximum and minimum time steps, as described in Section A9.1.3.5. The time step at the beginning of the restart is automatically the same as that at the end of the previous analysis run.

(4) New termination limits, as described in Section A9.1.3.6. The time at the end of the preceding analysis is displayed to provide guidance in specifying the new maximum time.

(5) New error tolerances, as described in Section A9.1.3.7.

(6) New output intervals, as described in Section A9.1.3.8.

The integration scheme *cannot* be changed.

A9.1.5 EXAMPLE

Table A9.1 shows typical session logs for WIPS-DATA sessions.

A9.1.6 FORCE AND DISPLACEMENT UNITS IN DATA FILE

In any DATA file, consistent units for forces and displacements must be used. DATA files set up by WIPS-DATA are always in kip and inch units, regardless of what units were used to specify the system geometry, component properties, etc.

In WIPS-DATA, certain force tolerances must be specified (see Section A9.1.3.7 and Table 9.1). These tolerances *must* be specified in kip units.

A9.2 WIPS-ANAL USER GUIDE

A9.2.1 VAX/UNIX SYSTEM

For the VAX/UNIX system at the University of California, Berkeley, WIPS-ANAL is executed from WIPS-EXEC using the "anal" command. The following data is then requested.

(1) Execution time limit (cpu seconds).

(2) Whether background or batch execution is required.

(3) For batch execution, whether day or night priority is required.

The status may be monitored using "ps" for background jobs and "batq" or "ps-x" for batch jobs. Progress can be monitored using "tail SLOGppss", where pp = problem number and ss = sequence number.

A9.2.2 OTHER SYSTEMS

A new section must be added to this chapter for each different system.

TABLE A9.1

WIPS-DATA EXAMPLE

USING MODL FILES FROM TABLES A8.2 AND A8.3

```
EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : data


DATA - SET UP WIPS-ANAL INPUT DATA

Sequence no. of MODL file : 1

Problem description (4 lines)
 Line 1 : Illustrative WIPS-DATA Example.                          12/28/81
 Line 2 :
 Line 3 : From MODL0101. Run 2, from b2 (break) to b3 (anchor), yz symmetry.
 Line 4 : Record "rec2", *0.5 for symmetry. U-bar at t4b. Small displacements

Set up PAUSE files at end of analysis ? :
Wait while files are processed

Specify data for each loaded point
LOAD NO.  1
  Force record name : rec2
  Name of loaded c.p. : b2
  Force direction (x,y,z or follower) : y
  Scale factor (+,- controls direction) : -0.5
  Time delay (sec) (dflt=0) :
LOAD NO.  2
  Force record name :
Last load? : y

Time steps : initial + max + min : .0005 .002 .00001
 Max steps + max total time : 200 .05
Error tolerances : upper + lower : 2 .4

Results output intervals
  Max. no. of steps : 1
  Max. time (secs.) : .001
Integration scheme (newm or hilb)(dflt=newm) :
Damping factor (dflt=0.1) :
Factor for time step increase (dflt=2.0) :
Factor for time step decrease (dflt=0.5) :
Max. unbal. to reduce time step (dflt=no limit) :
Max. unbal. to stop analysis (dflt=no limit) : 1000

DATA file set complete
Comment for file catalog : From MODL0101, rec2*0.5
```

```
WIPS-ANAL DATA FILES SET UP.  PARENT FILE NAME = DATA0101

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit

------------------------------------------------------------------------


EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : data


DATA - SET UP WIPS-ANAL INPUT DATA

Sequence no. of MODL file : 2

Problem description (4 lines)
 Line 1 : Illustrative WIPS-DATA Example.                          12/28/81
 Line 2 :
 Line 3 : From MODL0102. Runs 2,3 with break at a3, anchors at b1,b2.
 Line 4 : Record "rec1" *4.0. Unrestrained motion. With restart.

Set up PAUSE files at end of analysis ? : y
Wait while files are processed

Specify data for each loaded point
LOAD NO.   1
  Force record name : rec1
  Name of loaded c.p. : a3
  Force direction (x,y,z or follower) : foll
  Follower c.p. name : b3
  Scale factor (+,- controls direction) : 4
  Time delay (sec) (dflt=0) :
LOAD NO.   2
  Force record name :
Last load? : y

Time steps : initial + max + min : .0005 .002 .00003
Max steps + max total time : 20 .02
Error tolerances : upper + lower : 10 2

Results output intervals
  Max. no. of steps : 2
  Max. time (secs.) : .004
Integration scheme (newm or hilb)(dflt=newm) :
Damping factor (dflt=0.1) :
Factor for time step increase (dflt=2.0) :
Factor for time step decrease (dflt=0.5) :
Max. unbal. to reduce time step (dflt=no limit) :
Max. unbal. to stop analysis (dflt=no limit) : 1000

DATA file set complete
Comment for file catalog : From MODL0102, rec1*4, 20 steps.
```

```
WIPS-ANAL DATA FILES SET UP. PARENT FILE NAME = DATA0102

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit

-----------------------------------------------------------------------

AT THIS POINT WIPS-ANAL IS EXECUTED (2 ANALYSIS STEPS ONLY) AND PAUSE
FILES ARE PRODUCED. WIPS-DATA IS THEN CALLED TO MODIFY THE DATA FILE.

-----------------------------------------------------------------------

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : data


DATA - SET UP WIPS-ANAL INPUT DATA

Restart from existing PAUSE files ? : y
Sequence no. of DATA file set : 2
ECHO, SLOG and RSLT files may be purged if desired
  Purge ECHO file ? :
  Purge SLOG file ? :
  Purge RSLT file ? :
Initial time step will be step at end of last analysis
 Max. and min. time step sizes : .002 .00003
Time at end of last analysis =  0.11625e-01
Max steps + max total time : 40 .04
Error tolerances : upper + lower : 10 2

Results output intervals
  Max. no. of steps : 2
  Max. time (secs.) : .004
Integration scheme will be same as for last analysis
Factor for time step increase (dflt=2.0) :
Factor for time step decrease (dflt=0.5) :
Max. unbal. to reduce time step (dflt=no limit) :
Max. unbal  to stop analysis (dflt=no limit) : 1000

DATA file set modified for restart

DATA file set complete
Comment for file catalog : Restart, 40 steps or .04 sec.
WIPS-ANAL DATA FILES SET UP. PARENT FILE NAME = DATA0102

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit

-----------------------------------------------------------------------

WIPS-ANAL CAN NOW BE EXECUTED A SECOND TIME (40 MORE STEPS)

-----------------------------------------------------------------------
```

A10. WIPS-RSLT USER GUIDE

SUMMARY

This section describes the printing and plotting options available in the WIPS-RSLT post-processing module.

CONTENTS

A10.1 GENERAL FEATURES

A10.1.1 PURPOSE

WIPS-ANAL produces a RSLT file containing time-histories of results for node displacements, element stresses and strains (or actions and deformations), and impact forces. A set of results is saved at each of the output times, as specified in the WIPS-DATA phase. The purpose of WIPS-RSLT is to allow the WIPS user to extract results from the RSLT file and arrange them in tables and/or graphs.

A10.1.2 WIPS-RSLT OPTIONS

A10.1.2.1 General

WIPS-RSLT may be used to produce (a) time histories, (b) maximum values, and/or (c) deflected shapes.

A10.1.2.2 Time Histories

WIPS-RSLT obtains time histories by first setting up a table of output times and the corresponding values of any desired results quantities. The table may then be processed as follows.

(1) Display all or any part of the table on the terminal.

(2) Write the table to the WIPSLOG file.

(3) Graph any column of the table against time.

The procedure for obtaining time histories is explained in Section A10.3.

A10.1.2.3 Deflected Shapes

The RSLT file contains the original coordinates of all nodes in the analysis model plus the nodal displacements at all output times. This data can be used to draw the deflected shape of all or part of the analysis model at any desired output time. The procedure is explained in Section A10.4.

A10.1.2.4 WIPS-RSLT Commands

At the beginning of WIPS-RSLT, and after each post-processing operation is complete, WIPS-RSLT asks the user to specify the next WIPS-RSLT command. The commands are as follows.

TABLE: Produce a time-history table. This must be done before any MAXVAL, DISPLAY, PRINT, or GRAPH commands are specified.

MAXVAL: Determine maximum and minimum values, with times of occurrence, for all or selected columns in the current table. Display and write to the WIPSLOG file.

DISPLAY: Display all or part of the current table on the terminal. Any number of different "display" commands may be given for a single table to display different parts of the table. The displayed data is *not* written to the WIPSLOG file.

PRINT: Write the entire table to the WIPSLOG file. The table is *not* displayed on the terminal.

GRAPH: Graph any column of the table against time to obtain a time-history of response.

DRAW: Draw the deflected shape of all or part of the analysis model at any times.

QUIT: Return to WIPS-EXEC.

Any command may be abbreviated to its first 4 characters, if desired (i.e. TABL, MAXV, DISP, PRIN, GRAP, DRAW). Additional data is requested by WIPS-RSLT after each command, as described in the following sections.

A10.1.3 EXAMPLE

Table A10.2 shows a typical session log for a WIPS-RSLT session.

A10.2 TABLE COMMAND

A10.2.1 RESULTS QUANTITY DEFINITION

A10.2.1.1 General

Each column of a table contains the time-history of response for a particular *results quantity*. Results quantities are specified as follows.

A10.2.1.2 Node Identification in Main Structure

If the results quantity is a node displacement, the node and the displacement direction must be specified. A node in the main structure can be identified either by its control point (c.p.) name or by the number assigned to the node in the WIPS-GEOM phase (COOR data). When a WIPS user specifies a node identifier, WIPS-RSLT first checks to see if a node with that c.p. name exists. If not, WIPS-RSLT next checks to see if a node with that number exists. If a match is still not obtained, the identifier is flagged as incorrect.

The use of the c.p. name will be most common. However, if the displacement at an unnamed node (i.e. interpolated between c.p.'s) is required, it is possible that the node number could be confused with a c.p. name. For example, the node number might be 36, and a c.p. with the name 36 might also exist. In this case, specify the number 036 or 0036 to distinguish the number from the name. Note, however, that the number of characters must *not exceed 4*. Thus, 00036 would be interpreted as 0003, and the fifth character would be ignored.

A10.2.1.3 Node Identification in Substructures

A node inside a straight pipe, elbow, or slab substructure is defined by its two grid locations. Note that the terms "grid" and "mesh" have different meanings. A "grid" location refers to the intersection of two grid lines and defines a node. A "mesh" location refers to the intersection of two rows of elements and defines a shell element.

A10.2.1.4 Displacement Specification in Main Structure

At a node in the main structure, the results quantity may be any one of the following:

(1) Global X displacement.

(2) Global Y displacement.

(3) Global Z displacement.

(4) Displacement in any inclined direction.

If an inclined displacement is required, the WIPS user must specify the direction cosines of the displacement direction (not necessarily normalized).

A10.2.1.5 Displacement Specification in Substructures

At a node in a substructure, the results quantity may be any one of the following:

(a) Global X, Y, or Z.

(b) Local X, Y, or Z.

The local displacements in this context for straight pipe and elbow substructures are in the *longitudinal* (X), *circumferential* (Y), and *radially outwards* (Z) directions.

A10.2.1.6 Element Identification in Main Structure

If the results quantity is an element *output item* (stress, strain, etc.), the element and the output item must be specified.

An element in the main structure is identified by the following:

(1) The element type (PIPE, UBAR, etc.).

(2) The c.p. name or node number, at the element *J end* for two-node elements (PIPE, BEAM) or at the element location for one-node elements (UBAR, GAPF). The use of c.p. names and node numbers is as described in Section A10.2.1.2.

In some cases it is possible for the specification to be non-unique (for example, if two PIPE elements end at a branch point between two pipe runs). In such a case, WIPS-RSLT displays a message stating that multiple elements exist and asks the user to specify the element to be used. The element sequence in this case is the sequence in which the elements were defined in the WIPS-MODL phase.

A10.2.1.7 Element Identification in Substructures

An element inside a straight pipe, elbow, or slab substructure is identified by its mesh location. It is not necessary to specify the element type, because only shell elements are used.

A10.2.1.8 Element Output Item

For an element, the results quantity extracted may be any one of a number of element *output items*. The output items vary with the element type, as shown in Tables A10.1. The number of the required output item must be specified.

A10.2.1.9 Automatic Enveloping of Element Results

If the element location (c.p. or mesh) is left blank, the element output item is automatically the largest numerical value for all elements of that type in the specified substructure (or main structure). For example, if the largest value of a particular strain were required in a STRP substructure, the mesh location would be left blank, and the output item corresponding to that strain would be specified. The tabulated values would then be the largest strain (absolute value) occurring anywhere in the substructure.

A10.2.1.10 Impact Results

If the results quantity concerns impact, the impact surface and the required impact force component must be specified.

Each impact surface pair was assigned a name in the WIPS-MODL phase. This name is used to identify the surface. The results quantity may then be any one of the following:

(1) Magnitude of total force (normal plus friction).

(2) Magnitude of normal force.

(3) Magnitude of friction force.

(4) Global X, Y, or Z component of total force.

If several points are in contact at any surface, normal and friction force resultants for the surface are obtained by vector summation of the individual normal and friction forces. The force magnitudes are then the lengths of these resultant vectors. The X,Y,Z components are the components of the force exerted *by* the secondary nodes *on* the primary surface.

A10.2.2 REQUIRED INPUT DATA

For each column in the table, data is requested as follows.

(1) Substructure name (as assigned in the WIPS-MODL phase). Default is the main structure.

(2) Type of quantity. The types are as follows.

    (a) NODE (or just N): node displacement.

(b) ELEM (or just E): element output item.

(c) IMPACT (or just I): impact forces.

(d) COMBINE (or just C): combine two or more earlier columns.

(3) For NODE in main structure:

(a) Node identification (see Section A10.2.1.2).

(b) Displacement direction (X, Y, Z, or INCL).

(c) If INCL, direction cosines of direction (as the projections of any vector along the direction, not necessarily a unit vector).

(4) For NODE in a substructure:

(a) Node identification (see Section A10.2.1.3).

(b) Displacement direction (X, Y, or Z plus GLOBAL or LOCAL; see Section A10.2.1.5).

(5) For ELEM in main structure:

(a) Element identification (see Section A10.2.1.6).

(b) Output item number (see Section A10.2.1.8).

(6) For ELEM in a substructure:

(a) Element location (see Section A10.2.1.7).

(b) Output item number (see Section A10.2.1.8).

(7) For IMPACT:

(a) The name of the impact surface pair (see Section A10.2.1.10).

(b) The type of force required (X,Y,Z plus TOTAL, NORMAL, or FRICTION; see Section A10.2.1.10).

(8) For COMBINE:

(a) The numbers of the columns to be combined.

(b) If two columns are to be combined, whether the *sum or difference* is required. See Section A10.2.3.

A10.2.3 COMBINE OPTION

A10.2.3.1 Column Sum

The combine option can be used to add together two or more earlier columns in the table. A possible application is when impact with a plane surface is modeled using gap-friction elements at a number of different nodes. The total impact force at any time is the sum of the forces for the separate elements. This total force can be obtained by setting up a column for each element and then combining the columns.

A10.2.3.2 Column Difference

When impact occurs on an STRP or ELBO substructure, an important results quantity may be the change of pipe diameter. This change might be obtained as follows.

(1) Set up a column containing the vertical displacement at the top of the pipe.

(2) Set up a second column containing the vertical displacement at the bottom of the pipe.

(3) Use the COMBINE command and specify the *difference* option. This will give the change in diameter.

The change of horizontal diameter could similarly be obtained considering horizontal displacements.

A10.2.3.3 Alternative Procedure for Diameter Change

A more general procedure for obtaining the diameter change in an STRP or ELBO substructure is as follows.

(1) Set up a column containing the local Z displacement (radially outwards) at one grid location.

(2) Set up a second column containing the local Z displacement at the grid location at the opposite end of the diameter.

(3) Use the COMBINE command with the *sum* option.

Note that deformed cross sections can also be studied using the DRAW command.

A10.3 MAXVAL, PRINT, AND DISPLAY COMMANDS

A10.3.1 MAXVAL COMMAND

For each MAXVAL command, WIPS-RSLT first asks whether the column headings are to be displayed. A list of column numbers is then requested to identify the columns for which maximum values are required. Default is all columns. The maximum and minimum values and their times of occurrence are displayed and written to the WIPSLOG file.

A10.3.2 PRINT COMMAND

The PRINT command causes the complete current table to be written to the WIPSLOG file. The table is preceded by a heading which identifies the results quantity for each column of the table. At the end of each column the maximum and minimum values for the column are written. The table is not displayed on the terminal.

A10.3.3 DISPLAY COMMAND

A10.3.3.1 Purpose

The DISPLAY command may be used to display all or part of the current table on the terminal. The time column plus up to five other columns may be displayed at one time. The columns may be displayed for the full range of times in the table or for only a specified range of times. Maximum values are not displayed. The display is not copied to the WIPSLOG file.

A10.3.3.2 Required Input Data

For each DISPLAY command, WIPS-RSLT requests the following:

(1) Column numbers to be displayed.

(2) Starting time (default = first time in table).

(3) Ending time (default = last time in table).

A10.4 GRAPH AND DRAW COMMANDS

A10.4.1 PLOT DEVICE

Potting is performed interactively, using either a graphics terminal or a pen plotter connected to a regular CRT terminal. Any of the following plot devices may be specified:

(a) Tektronix 4662 pen plotter.

(b) Tektronix 4013 or 4015 graphics terminal.

(c) Tektronix 4027 color graphics terminal.

A10.4.2 GRAPH COMMAND

A10.4.2.1 Columns to be Graphed

The graph X axis is always time. The full range, or any part of the range, of times in the current table may be considered.

Up to 5 columns may be plotted against time in a single graph. The required columns must be specified, in response to a request from WIPS-RSLT.

A10.4.2.2 Grid Limits

WIPS-RSLT displays the minimum and maximum times and the minimum and maximum values for each of the columns to be graphed. The WIPS-RSLT user is then requested to specify lower and upper limits for the X and Y axes. These will typically be round numbers.

An option is also provided to multiply the values in any column by a scale factor before plotting. This allows, for example, negative values to be converted to positive.

A10.4.2.3 Plot Size

For a pen plotter, the paper size and border widths must be specified in inch units. The plot grid then occupies the remaining width and height. The borders should allow sufficient space for adding scales and titles.

For a graphics terminal, the "paper" size and border widths must be specified as proportions of the screen width. Maximum allowable values are displayed for guidance.

A10.4.2.4 Plot Scales

The scales for the X and Y axes are displayed. If these are not suitable, the user may return to specify a different plot size.

A10.4.2.5 Tick Spacing

Lines are drawn for the borders and, if necessary, for the X and Y axes. Tick marks at user-specified spacings may be added to the axes, or a complete graph grid may be drawn.

Scales may be added alongside the axes. The numerical values at the tick or grid locations are assigned automatically. Labels for the axes (e.g. Time (sec)) may be specified, if desired, for the 4662 pen plotter only.

A10.4.2.6 Text

Text (titles, etc.) can be added to plots produced by the Tektronix 4662 pen plotter only. The text is added interactively at positions which the WIPS-RSLT user selects by positioning the pen manually. The required character height, text, and text angle (0 = horizontal, 90 = vertical) are entered in response to requests from WIPS-RSLT. The user then positions the pen and hits RETURN to plot the text. Any desired amount of text may be added.

A10.4.3 DRAW COMMAND

A10.4.3.1 Parts of Model to be Drawn

The drawing may consist of one or more *parts* of the analysis model. A *part* can consist of all or part of an STRP, ELBO, or SLAB structure, or all or part of the main structure.

For the main structure, *parts* are specified by the c.p. names at the beginning and end of the part. Each part must lie within a single pipe run.

For STRP and ELBO substructures, a *part* consists of one or more circumferential grid lines (i.e. pipe cross sections). The full circumference (or, if there is transverse symmetry, the half circumference) is always drawn for each cross section. The number of cross sections to be drawn is controlled by specifying the first and last sections. If the first and last values are the same, a single cross section is drawn.

For SLAB substructures, a *part* is defined by specifying the first and last grid lines to be drawn along edges OA and OB of the substructure.

A10.4.3.2 Times

Deflected shapes at up to 5 different times may be plotted on a single drawing. Time zero corresponds to the undeformed shape.

WIPS-RSLT first requests the times. These need not exactly match output times in the RSLT file. If a specified time lies between two output times, the nearest output time is used.

A10.4.3.3 Viewing Direction

The viewing direction may be along any of the global X, Y, or Z axes, or along any inclined axis. For inclined viewing, the direction cosines (not necessarily normalized) of the view direction must be specified. For example, a standard isometric is produced with direction cosines -1,-1,-1.

A10.4.3.4 Vertical Axis

The vertical axis in the drawing may be the projection on the drawing plane of the global X, Y, or Z axes (provided that the chosen axis is not parallel to the view direction). Default is global Y.

A10.4.3.5 Displacement Scaling

Displacements may be exaggerated by scaling if desired. Displacements will typically be scaled for restrained piping systems (small displacements) but not for unrestrained systems (large displacements).

A10.4.3.6 Drawing Size

The drawing size and border widths may be specified as for the GRAPH command. The drawing scale is calculated and displayed. If this scale is not satisfactory, it can be adjusted by changing the drawing size.

A10.4.3.7 Text

Text may be added as for the GRAPH command.

TABLE A10.1a - OUTPUT ITEMS FOR PIPE ELEMENTS

Item	Description
1	Axial force at end I.
2	In-plane (M_z) moment at end I.
3	Out-of-plane (M_{yy}) moment at end I.
4	Torsional moment at end I.
5-8	As for 1-4, but at end J.
9	In-plane (V_y) shear force at slice 1.
10	Out-of-plane (V_z) shear force at slice 1.
11-12	As for 9-10, but at slice 2.
13	In-plane ovalling at 1 (change in radius).
14	Out-of-plane ovalling at slice 1.
15-16	As for 13-14, but at slice 2.
17	Strain at pipe axis at slice 1.
18	In-plane curvature at slice 1.
19	Out-of-plane curvature at slice 1.
20	Maximum hoop strain (combined membrane strain and bending strain due to ovalling) at slice 1.
21	Torsional shear strain at slice 1.
22-26	As for 17-21, but at slice 2.
27	Maximum axial strain.
28	Maximum hoop strain.
29	Maximum torsional shear strain.
30	Maximum effective strain.*

*Maximum value of

$$(4(\epsilon_{pa}^2 + \epsilon_{ph}^2 + \epsilon_{pa}\epsilon_{ph} + 0.25\gamma_p^2)/3)^{0.5} + \sigma_e/E$$

where

ϵ_{pa} = plastic axial strain (membrane strain only);

ϵ_{ph} = plastic hoop strain (including bending due to ovalling);

γ_p = plastic torsional shear strain;

σ_e = von Mises effective stress;

E = Young's modulus;

and

plastic strain = total strain minus elastic strain.

TABLE A10.1b - OUTPUT ITEMS FOR BEAM ELEMENTS

Item	Description
1	Moment M_{yy} at end I.
2	Moment M_{zz} at end I.
3-4	As for 1-2, but at end J.
5	Torsional moment.
6	Axial force.
7	Shear force V_y.
8	Shear force V_z.
9	Curvature ψ_{yy} at end I.
10	Curvature ψ_{zz} at end I.
11-12	As for 9-10, but at end J.
13	Rate of torsional twist.
14	Axial strain.

TABLE A10.1c - OUTPUT ITEMS FOR UBAR ELEMENTS

Item	Description
1	Total axial force.
2	Axial extension.
3	Extension rate.

TABLE A10.1d - OUTPUT ITEMS FOR GAPF ELEMENTS

Item	Description
1	Bearing force.
2	Deformation of normal spring.
3	Local x force.
4	Local x deformation.
5	Local y force.
6	Local y deformation.
7	Resultant friction force.
8	Resultant tangent deformation.

TABLE A10.1e - OUTPUT ITEMS FOR SHELL ELEMENTS

Item	Description
1-3	Total strains (ϵ_{11}, ϵ_{22}, ϵ_{12}) at output point 1.
4-6	Total strains at output point 2.
7-9	Total stresses (σ_{11}, σ_{22}, σ_{12}) at output point 1.
10-12	Total stresses at output point 2.
13-15	Stresses due to rate dependence at output point 1.
16-18	Stresses due to rate dependence at output point 2.
21-22	Effective strains at output points 1,2.*
23-24	Mises effective stresses at output points 1,2.

*As for PIPE elements.

TABLE A10.2

WIPS-RSLT EXAMPLE

```
EXEC - WIPS EXECUTIVE

NEXT WIPS-EXEC COMMAND : rslt

RSLT - POSTPROCESSING OF WIPS-ANAL RESULTS

Define units
  Length (ft, in, m, mm) : in
  Force (k, lb, kgf, kN) : k

Sequence no. of RSLT file : 1     RESTRAINED SYSTEM, FROM DATA0101
Wait while data is set up

PROBLEM TITLE
     Illustrative WIPS-DATA Example.                          12/28

     From MODL0101. Run 2, from b2 (break) to b3 (anchor), yz sym
     Record "rac2", *0.5 for symmetry. U-bar at t4b. Small displa

NEXT WIPS-RSLT COMMAND : tabl

Starting time of results = 0.00050
Ending time              = 0.05175
No. of output times      =      46
Include all times in table ? : y
No. of columns in table (dflt=10) : 4
Define column contents

COLUMN  1
   Data type (node, elem, impa or comb) : n
   C.p. name or node no. : b2
   Displ, veloc or accel (d, v or a) : d
   Direction (x, y, z or incl) : y          Y DISPL AT POINT B2

COLUMN  2
   Data type (node, elem, impa or comb) : e
   Element type (dflt=no change) : ubar
   C.p. name or node no. (dflt=all elem) : t4b
   Output item no. : 1                               FORCE IN U-BAR

COLUMN  3
   Data type (node, elem, impa or comb) : e
   Element type (dflt=no change) : pipe
   C.p. name or node no. (dflt=all elem) : t4b
   Output item no. : 6                          MOMENT AT T4B IN ELBOW

COLUMN  4
   Data type (node, elem, impa or comb) : e
   Element type (dflt=no change) :
```

```
   C.p. name or node no. (dflt=all elem) :
   Output item no. : 6                              MAX. MOMENT, ALL ELEMENTS

Wait while table is set up
Table complete

NEXT WIPS-RSLT COMMAND : maxval

Display column headings? : y

Column numbers (max.10)(dflt=all) :

MAXIMUM VALUES

 Col.    Max. Pos.         Time      Max. Neg.          Time
   1   0.       e+00     0.         -0.9671e+00       0.00775
   2   0.1632e+02        0.00475    0.       e+00     0.
   3   0.7174e+02        0.00675   -0.1037e+01        0.03175
   4   0.1033e+03        0.00350    0.       e+00     0.
END OF DATA

NEXT WIPS RSLT COMMAND : prin            DO NOT DISPLAY ON SCREEN



TABLE NO. 1 OF THIS RSLT SESSION


Col  Rslt  Subs  Subs  C.P.  Node  Elem  Elem  Rslt  Col Col Col Col
No.  Type  Name  Type  or X  or Y  Type  Item  Dirn   1   2   3   4

  1  disp  main        b2     10                y-g
  2  elem  main        t4b    16   ubar     1
  3  elem  main        t4b    16   pipe     6
  4  elem  main        ALL   ALL   pipe     6

Time(sec)   Column  1   Column  2   Column  3   Column  4

 0.00050 -0.1365e-01  0.      e+00  0.6024e+01  0.6024e+01
 0.00100 -0.5691e-01  0.      e+00  0.1342e+02  0.1602e+02
 0.00150 -0.1299e+00  0.      e+00  0.8311e+01  0.3803e+02
 0.00200 -0.2343e+00  0.      e+00  0.3090e+01  0.7180e+02
 0.00250 -0.3695e+00  0.      e+00  0.6740e+01  0.8638e+02
 0.00300 -0.5234e+00  0.4558e+01  0.1289e+02  0.9422e+02
 0.00325 -0.6012e+00  0.1607e+02  0.2093e+02  0.9984e+02
 0.00350 -0.6723e+00  0.1615e+02  0.3417e+02  0.1033e+03
 0.00375 -0.7321e+00  0.1620e+02  0.4409e+02  0.1032e+03
 0.00400 -0.7803e+00  0.1625e+02  0.4874e+02  0.1002e+03
 0.00425 -0.8194e+00  0.1629e+02  0.5055e+02  0.9739e+02
 0.00475 -0.8743e+00  0.1632e+02  0.5413e+02  0.9264e+02
 0.00500 -0.8908e+00  0.1363e+02  0.5824e+02  0.8673e+02
 0.00525 -0.9019e+00  0.6320e+01  0.6017e+02  0.7615e+02
 0.00550 -0.9117e+00  0.3160e+01  0.6041e+02  0.7181e+02
 0.00575 -0.9239e+00  0.6612e+01  0.6193e+02  0.7222e+02
 0.00600 -0.9385e+00  0.1145e+02  0.6478e+02  0.7338e+02
```

0.00625	-0.9517e+00	0.1265e+02	0.6841e+02	0.7421e+02
0.00650	-0.9603e+00	0.1017e+02	0.7118e+02	0.7118e+02
0.00675	-0.9641e+00	0.7339e+01	0.7174e+02	0.7174e+02
0.00725	-0.9665e+00	0.6869e+01	0.6970e+02	0.7079e+02
0.00775	-0.9671e+00	0.8462e+01	0.6893e+02	0.7151e+02
0.00825	-0.9637e+00	0.6786e+01	0.6636e+02	0.6886e+02
0.00875	-0.9542e+00	0.4962e+01	0.5789e+02	0.6903e+02
0.00975	-0.9271e+00	0.3990e+01	0.3393e+02	0.8406e+02
0.01175	-0.8796e+00	0.4336e-01	0.1883e+01	0.9442e+02
0.01375	-0.8806e+00	0.1263e+01	-0.1930e+00	0.9528e+02
0.01575	-0.9234e+00	0.3514e+01	0.3870e+02	0.7932e+02
0.01775	-0.9422e+00	0.4654e+01	0.5501e+02	0.7168e+02
0.01975	-0.9120e+00	0.2996e+01	0.2854e+02	0.8486e+02
0.02175	-0.8768e+00	0.3437e+00	-0.6238e+00	0.9356e+02
0.02375	-0.8864e+00	0.1840e+01	0.7365e+01	0.9403e+02
0.02575	-0.9250e+00	0.3408e+01	0.4165e+02	0.7591e+02
0.02775	-0.9347e+00	0.4790e+01	0.5037e+02	0.7615e+02
0.02975	-0.9035e+00	0.2020e+01	0.2270e+02	0.8445e+02
0.03175	-0.8771e+00	0.1024e+01	-0.1037e+01	0.9547e+02
0.03375	-0.8936e+00	0.1765e+01	0.1390e+02	0.8925e+02
0.03575	-0.9278e+00	0.3992e+01	0.4407e+02	0.7605e+02
0.03775	-0.9295e+00	0.4210e+01	0.4590e+02	0.7736e+02
0.03975	-0.8978e+00	0.1742e+01	0.1746e+02	0.8602e+02
0.04175	-0.8792e+00	0.1300e+01	0.1016e+01	0.9564e+02
0.04375	-0.9001e+00	0.1897e+01	0.1954e+02	0.8510e+02
0.04575	-0.9292e+00	0.4465e+01	0.4542e+02	0.7733e+02
0.04775	-0.9238e+00	0.3406e+01	0.4085e+02	0.7727e+02
0.04975	-0.8935e+00	0.1936e+01	0.1370e+02	0.8917e+02
0.05175	-0.8623e+00	0.1140e+01	0.3814e+01	0.9324e+02
Max. Pos.	0. e+00	0.1632e+02	0.7174e+02	0.1033e+03
Time	0.	0.00475	0.00675	0.00350
Max. Neg.	-0.9671e+00	0. e+00	-0.1037e+01	0. e+00
Time	0.00775	0.	0.03175	0.

```
NEXT WIPS RSLT COMMAND : graph

Plot device (dflt=4662) :
Baud rate (dflt=1200) :

Starting time of table = 0.00050
Ending time            = 0.05175
No. of times in table  =      46
Include all times in graph ? : y

No. of columns to be graphed (max=5) : 1
  Col. no. for line 1 : 2                    GRAPH U-BAR FORCE
Display column headings ? :

Any changes ? :

MAX. AND MIN. VALUES
   LINE   COL.      MIN. VALUE      MAX. VALUE
 X-axis time        0.5000e-03      0.5175e-01
```

```
    1      2      0.    e+00    0.1632e+02
Multiply any columns by scale factors? :

Specify lower and upper limits of grid
  Lower limit for time axis : 0
  Upper limit for time axis : .06
  Lower limit for Y axis : 0
  Upper limit for Y axis : 20
Specify drawing size and border widths
  Max. allowable width and height =  15.000,   10.000
  Width and height (incl. borders) : 11 8.5
  Left and right border widths : 2.5 2
  Top and bottom border widths : 2 2.5
Scales (graph units per drawing unit)
  Horizontal =  0.9231e-02
  Vertical   =  0.5000e+01
Any changes ? :
Omit grid (i.e. plot line on existing grid)? : n
Tick or grid spacing (graph units)(dflt=none)
  Horizontal axis : .005
  Vertical axis   : 4
Specify tick or grid (t or g)(dflt=t) :
Add scales along axes? : y
Enter labels for axes (max 40 char)
 Horiz. axis : Time (sec)
 Vert. axis  : U-bar Force (k)
Use default character size? : y
Hit RETURN to begin plotting                  PLOTS GRAPH FOR FIG. A10.1

Add text to drawing? : y
Enter text (max.60 characters)
 : FIG. A10.1  FORCE IN U-BAR
Symbol height (inches) : .125
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : n

NEXT WIPS-RSLT COMMAND : graph

Plot device (dflt=no change) :

Starting time of table = 0.00050
Ending time            = 0.05175
No. of times in table  =      46
Include all times in graph ? : y

No. of columns to be graphed (max=5) : 2
  Col. no. for line 1 : 3
  Col. no. for line 2 : 4
Display column headings ? : y

Col  Rslt  Subs  Subs  C.P.  Node  Elem  Elem  Rslt  Col  Col  Col  Col
No.  Type  Name  Type  or X  or Y  Type  Item  Dirn   1    2    3    4

  3  elem  main        t4b    16   pipe    6
```

```
  4  elem  main          ALL   ALL  pipe     6

Any changes ? :

MAX.  AND MIN.  VALUES
   LINE  COL.        MIN. VALUE       MAX. VALUE
 X-axis time       0.5000e-03     0.5175e-01
    1      3      -0.1037e+01     0.7174e+02
    2      4       0.6024e+01     0.1033e+03
Multiply any columns by scale factors? : y
  Col. no. and scale factor (blank to end) : 3 2.
  Col. no. and scale factor (blank to end) : 4 2.
  Col. no. and scale factor (blank to end) :

MAX.  AND MIN.  VALUES
   LINE  COL.        MIN. VALUE       MAX. VALUE
 X-axis time       0.5000e-03     0.5175e-01
    1      3      -0.2074e+01     0.1435e+03
    2      4       0.1205e+02     0.2066e+03
Multiply any columns by scale factors? :

Specify lower and upper limits of grid
  Lower limit for time axis : 0
  Upper limit for time axis : .06
  Lower limit for Y axis : -50
  Upper limit for Y axis : 250
Change drawing size or borders? :
Scales (graph units per drawing unit)
  Horizontal =  0.9231e-02
  Vertical   =  0.7500e+02
Any changes ? :
Omit grid (i.e. plot line on existing grid)? :
Tick or grid spacing (graph units)(dflt=none)
  Horizontal axis : .01
  Vertical axis   : 50
Specify tick or grid (t or g)(dflt=t) : g
Add scales along axes? : y
Enter labels for axes (max 40 char)
 Horiz. axis : Time (sec)
 Vert. axis  : Pipe Mzz (k.in)
Use default character size? : y
Hit RETURN to begin plotting                  PLOTS FIG. A10.2

Add text to drawing? : y
Enter text (max.60 characters)
 : FIG. A10.2  MOMENTS Mzz IN PIPE
Symbol height (inches) : .125
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : Max.
Symbol height (dflt=no change) : .1
Horizontal lettering? : y
Position pen, then hit RETURN
```

```
More text? : y
Enter text (max.60 characters)
 : At T4B
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? :

NEXT WIPS RSLT COMMAND : quit
End of this RSLT session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : rslt

RSLT - POSTPROCESSING OF WIPS-ANAL RESULTS

Define units
  Length (ft,in,m,mm) : in
  Force (k,lb,kgf,kN) : k

Sequence no. of RSLT file : 2    UNRESTRAINED SYSTEM, FROM DATA0102
Wait while data is set up

PROBLEM TITLE
     Illustrative WIPS-DATA Example.                              12/28/

     From MODL0102. Runs 2,3 with break at a3, anchors at b1,b2.
     Record "rec1" #4.0. Unrestrained motion. With restart.

NEXT WIPS RSLT COMMAND : draw

Plot device (dflt=4662) :
Baud rate (dflt=1200) :

SPECIFY PARTS OF MODEL TO BE DRAWN
PART  1
  Beginning c.p.  : b2
  Ending c.p.     : a3
More parts? : y
PART  2
  Beginning c.p.  : b1
  Ending c.p.     : b3
More parts? :

First time in results = 0.00025
Last time             = 0.04163
No. of times for this drawing (max.5) : 3
Specify times. Zero = undeformed shape
  Time 1 :
  Time 2 : .02
  Time 3 : .04
Closest times are as follows
  Time 1   Undeformed
```

```
  Time 2  0.19625e-01
  Time 3  0.41625e-01
Any changes? :

Viewing direction (+x,-x,+y,-y,+z,-z or incl) : incl
Direction cosines (3 values) : -1 -1 -1
Vertical axis in drawing (dflt=+y) :

Scale factor for displs (dflt=1.) :
Max. drawing dimensions (current length units)
  Width  =  0.3233e+02
  Height =  0.4556e+02
Specify drawing size and border widths
  Max. allowable width and height =  15.000,  10.000
  Width and height (incl. borders) : 11 8.5
  Left and right border widths : 2 2
  Top and bottom border widths : 2 2
Scale (length units per drawing unit) =  0.1012e+02
Any changes ? :
Hit RETURN to begin plotting              DRAWS FIG. A10.3

Add text to drawing? : y
Enter text (max.60 characters)
 : FIG. A10.3  DISPLACED SHAPES, T = 0,0.02,0.04
Symbol height (inches) : .125
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : B1
Symbol height (dflt=no change) : .11
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : B2
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : A3
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : B3
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN
```

```
More text? : y
Enter text (max.60 characters)
 : NOTE
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : Element subdivision is
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : very coarse. More elements
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? : y
Enter text (max.60 characters)
 : would be needed for accuracy.
Symbol height (dflt=no change) :
Horizontal lettering? : y
Position pen, then hit RETURN

More text? :
Add X,Y,Z axis symbol? : y
Axis length (dflt=0.25in) :
Symbol location (bl,tl,br or tr) : tr

NEXT WIPS RSLT COMMAND : quit
End of this RSLT session

EXEC - WIPS EXECUTIVE

NEXT WIPS EXEC COMMAND : quit
```

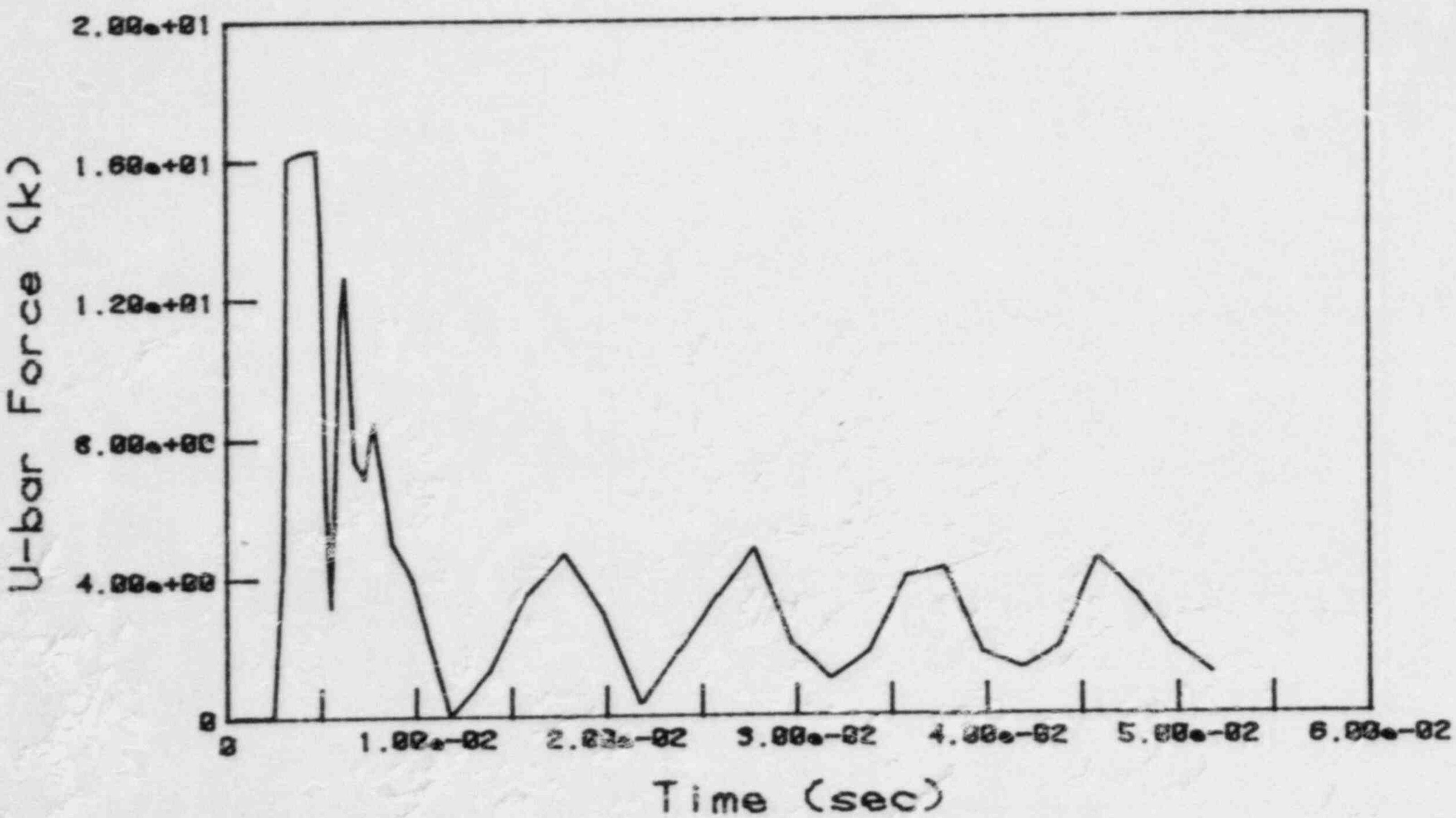


FIG. A10.1 FORCE IN U-BAR

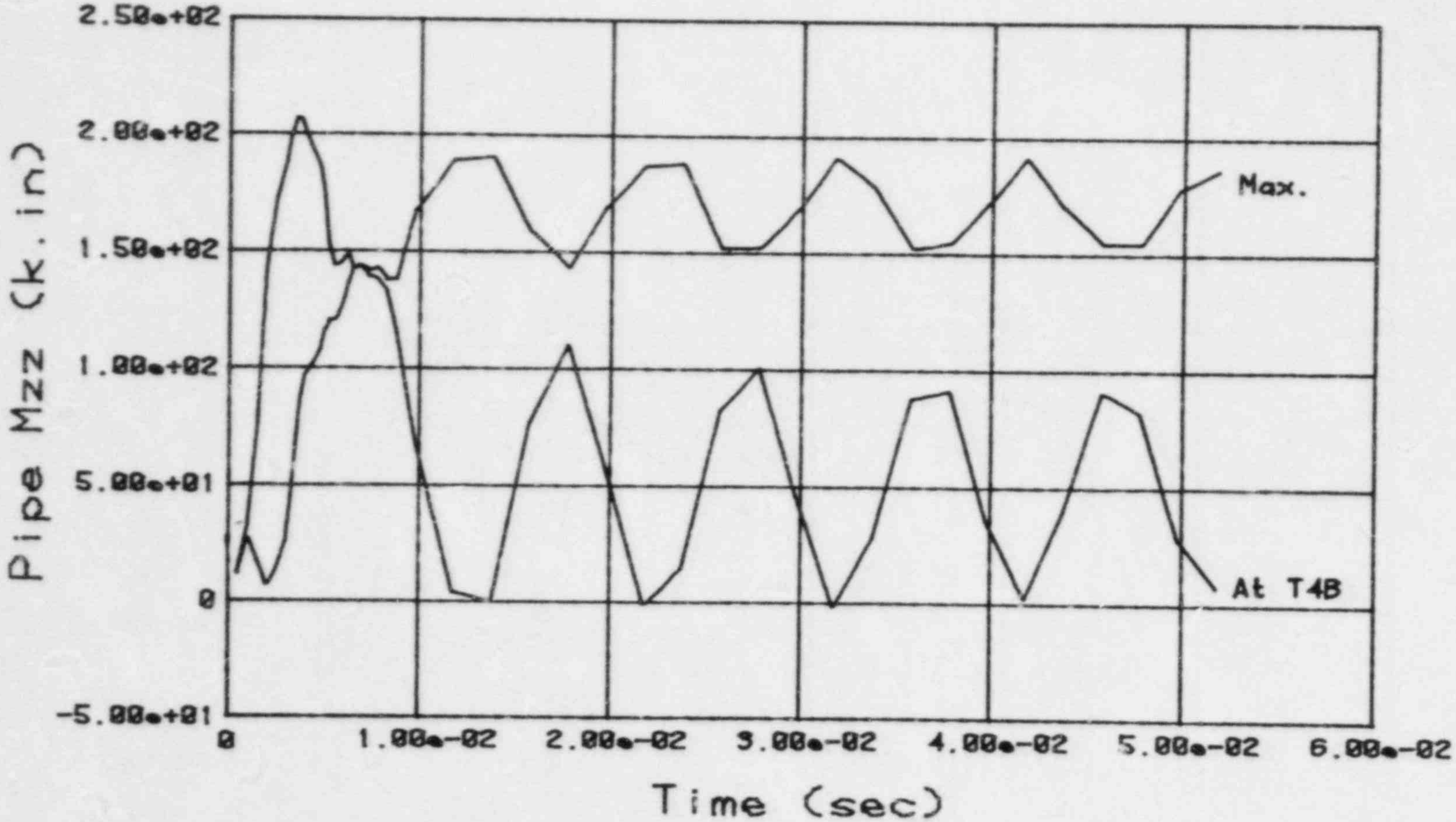


FIG. A10.2 MOMENTS Mzz IN PIPE

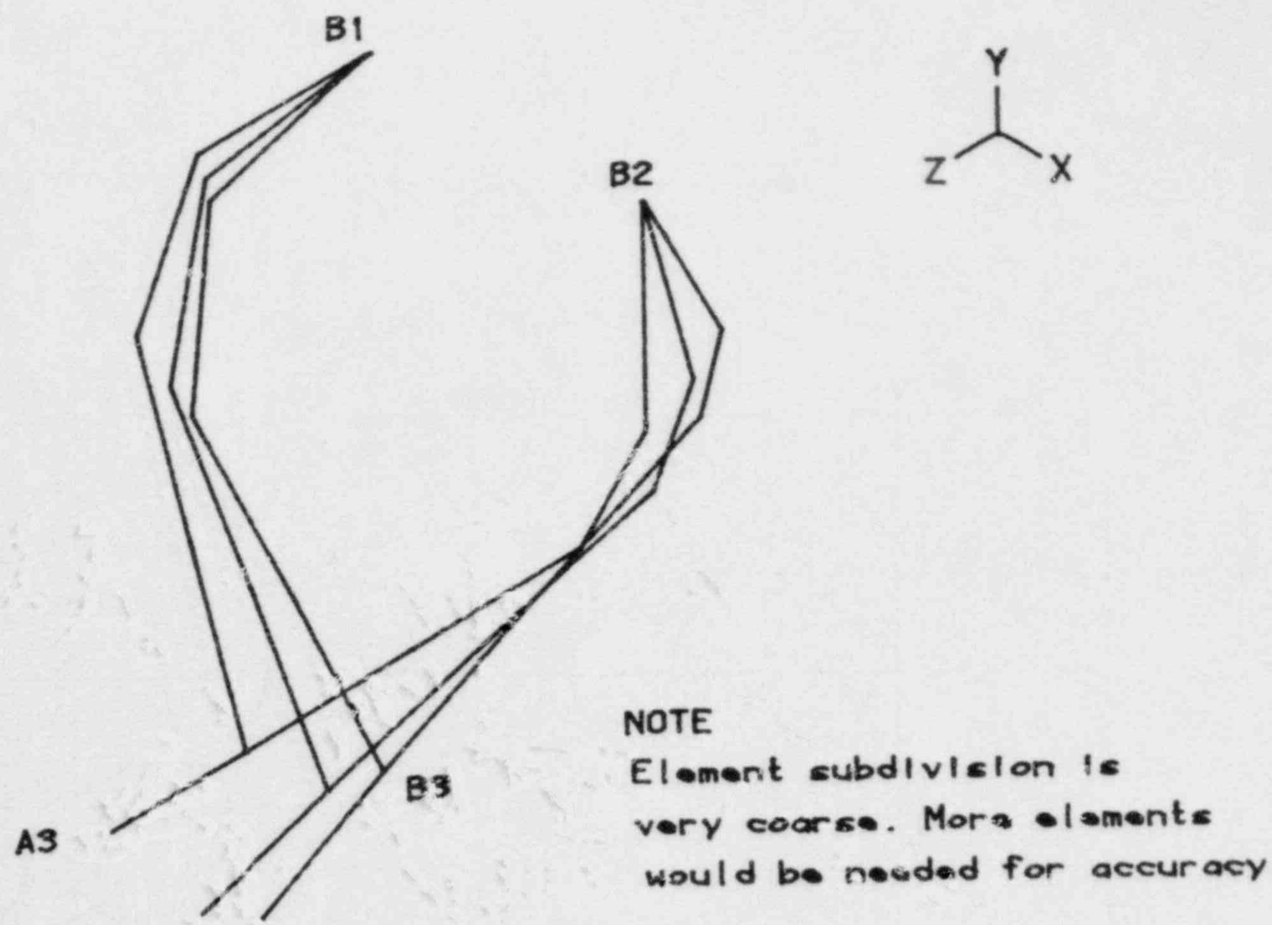


FIG. A10.3 DISPLACED SHAPES, T = 0,0.02,0.04

A11. WIPS-ANAL INPUT DATA

SUMMARY

For most applications, input data files for WIPS-ANAL will be prepared using the WIPS-DATA module. However, input data files for WIPS-ANAL may be prepared by other means if desired. This section describes the structure of the WIPS-ANAL input data. In effect, this section is a user guide for WIPS-ANAL.

CONTENTS

A11.1 WIPS-ANAL INPUT FILE

A11.1.1 GENERAL

A formatted DATA file is created by WIPS-DATA. This file is the input file for WIPS-ANAL. Part of the data is obtained from a MODL file produced by WIPS-MODL. This part contains input data defining the geometry and element properties for the analysis model. The remaining part of the data is added by WIPS-DATA. This remaining data includes the loading specifications and control information for the step-by-step analysis.

DATA files will typically not be seen by the WIPS user. However, because DATA files are formatted, they may be displayed, listed, and/or edited (using options available in the computer operating system, not WIPS modules). This permits files to be corrected (if WIPS-MODL or WIPS-DATA produces files which contain errors), extended (if modeling features are required which are not available in WIPS), or constructed completely from scratch.

This section describes the data formats for the DATA file. During execution from the file, the computation is controlled by a series of commands. Each command consists of an alpha command word, plus alpha, integer, and/or real command data. Only the first four characters of the command word need to be specified. Certain commands may be followed by "cards" (lines) of formatted input data (e.g. node coordinates).

A11.2 OVERALL CONTROL

A11.2.1 GENERAL

The first data card defines the names of the ECHO, SLOG, RSLT, PAUS, and PAUZ files. The input data proper then begins with either a START or RESTART command and ends with a STOP or PAUSE command.

A11.2.2 FIRST DATA CARD

In 5(2X,A8) format, enter filenames for the following files:

ECHO: echo of input data

SLOG: solution log

RSLT: results time-history

PAUS: database for restart

PAUZ: database for restart

Filenames for PAUS and PAUZ need not be provided if the input data proper begins with START and ends with STOP.

A11.2.3 START - BEGIN NEW ANALYSIS

Command Format:

START

A11.2.4 RESTART - CONTINUE EXISTING ANALYSIS

Command Format:

RESTART

A11.2.5 STOP - END ANALYSIS, NO RESTART OPTION

Command Format:

STOP

A11.2.6 PAUSE - END ANALYSIS, WITH RESTART OPTION

Command Format

PAUSE

A11.3 STRUCTURAL DESCRIPTION

A11.3.1 GENERAL

The structure is built by defining and connecting substructures. Substructures are first defined as SUBSTRUCTURE TYPES. Once a SUBSTRUCTURE TYPE has been defined, it can be used to define SUBSTRUCTURE TYPES of progressively higher levels. The ANALYSIS MODEL is typically at the highest level and defines the structure to be analyzed.

The building process starts with a GBUILD command. The data defining each SUBSTRUCTURE TYPE is initiated with a NEW command and terminated with an END command. The end of the building process is terminated with a FINISH command.

A11.3.2 GBUILD - BEGIN BUILDING PROCESS

Command Format:

GBUILD

A11.3.3 FINISH - END BUILDING PROCESS

Command Format:

FINISH

A11.4 DEFINE SUBSTRUCTURE TYPES

A11.4.1 NEW - BEGIN NEW SUBSTRUCTURE TYPE

Command Format:

NEW,NATY,NUN,NSYS,NINT,MAXN,MAXE

Variable	Format	Data
NATY	A	Substructure type name (max. 4 characters).
NUN	A	Linear or nonlinear substructure type. NONL = nonlinear LINE = linear
NSYS	A	Coordinate system for this substructure type. CART = cartesian CYLN = cylindrical SPHR = spherical
NINT	I	Node format code. 0 = integer (number each node, in any sequence). 1 = alphanumeric (name each node, max. 4 characters).
MAXN	I	Upper limit on number of nodes for this substructure (optional; default = no limit).
MAXE	I	Upper limit on number of elements for this substructure (optional; default = no limit).

FOLLOWING DATA

Follow by one card containing optional title (80A1).

A11.4.2 END - END THIS SUBSTRUCTURE TYPE

Command Format:

END

A11.4.3 REFNOD - NODE COORDINATES

Command Format:

REFNOD,MREF

Variable	Format	Data
MREF	I	Number of nodes for which coordinates are input.

FOLLOWING DATA - MREF cards as follows.

Columns	Variable	Format	Data
2 - 5	NDEX	I/A	Node number or name.
6 - 20	N1	F	Coordinate 1*
21-35	N2	F	Coordinate 2*
36-50	N3	F	Coordinate 3*

*The following convention must be followed for coordinate input, depending on NSYS.

COORDINATE	NSYS		
	CART	CYLN	SHPR
1	X	R	R
2	Y	θ	θ
3	Z	Z	ϕ

A11.4.4 CONSUB - CONTRIBUTING SUBSTRUCTURES

Command Format:

CONSUB,NCSB

Variable	Format	Data
NCSB	I	Number of lower level substructure types contributing to this substructure.

FOLLOWING DATA - NCSB sets of cards as follows.

(A) SUBSTRUCTURE IDENTIFICATION - 1 card

Columns	Variable	Format	Data
2 - 5	KTYP	A	Type name of lower level substructure.
6 - 10	NKNC	I	Number of connection nodes.
15	IFLG	I	Coordinate code: 1 = different system from current substructure 0 = same system as current substructure

(B) SUBSTRUCTURE LOCATION - 3 cards. Omit if IFLG = 0.

Columns	Variable	Format	Data
2 - 5		I/A	Name or number of any connection node in substructure type KTYP.
6 - 20		F	Coordinate 1 in current substructure.
21-35		F	Coordinate 2 in current substructure.
36-50		F	Coordinate 3 in current substructure.

(C) CONNECTION NODES - NKNC Cards

Columns	Variable	Format	Data
2 - 5	NK	I/A	Connection node name or number in substructure type KTYP.
7 - 10	NF	I/A	Node name or number in current substructure.

A11.4.5 BOUND - BOUNDARY CONDITIONS

Command Format:

BOUND,NBCM

Variable	Format	Data
NBCM	I	Number of boundary condition commands.

FOLLOWING DATA - NBCM cards as follows.

Columns	Variable	Format	Data
2 - 5	NDEX	I/A	Node name or number.
6 - 29	Blank		
30	KOD(1)	I	
31	KOD(2)	I	
32	KOD(3)	I	Boundary condition code for each global d.o.f. (0=free; 1=fixed).
33	KOD(4)	I	
34	KOD(5)	I	
35	KOD(6)	I	

A11.4.6 EQUAL - EQUAL DISPLACEMENT CONSTRAINTS

Command Format:

EQUAL,NEQC

Variable	Format	Data
NEQC	I	Number of constrained nodes.

FOLLOWING DATA - NEQC cards as follows.

Columns	Variable	Format	Data
2 - 5	NDEX	I/A	Name or number of slave (constrained) node.
7 - 10	NEQN	I/A	Name or number of master node.

A11.4.7 SLAVE - SLAVED DISPLACEMENT CONSTRAINTS

Command Format:

SLAVE,NMSC

Variable	Format	Data
NMSC	I	Number of slave nodes.

FOLLOWING DATA - NMSC cards as follows.

Columns	Variable	Format	Data
2 - 5	NDEX	I/A	Name or number of slave node.
7 - 10	NSLM	I/A	Name or number of master node.
11 - 29	Blank		
30 - 35	KOD(6)	6I1	Slaving Code: 0=not slaved; 1=slaved. See Fig. A11.1 for definition of the slaving constraint.

A11.4.8 CNOD - CONNECTION NODES

Command Format:

CNOD,NCND

Variable	Format	Data
NCND	I	Number of cards defining nodes which connect to higher level substructures.

FOLLOWING DATA - NCND cards as follows.

Columns	Variable	Format	Data
2 - 5		I/A	Node name or number.
6 - 10			Blank
12 - 16			
17 - 20		I/A	Additional node names or numbers. Up to 14 nodes.
etc.			

A11.4.9 ENOD - END NODE SPECIFICATION

Command Format:

ENOD,IPRT

Variable	Format	Data
IPRT	I	Print indicator (0 = no print; 1 = print summary of node data).

A11.5 MATERIAL TABLES

A11.5.1 GENERAL

The MATERIAL tables contain material or element strength and stiffness data. The required data for each element type is shown in Tables A11.5. A column in the tables must be associated with each element (see ELEMENTS command). The information contained in this column then defines the element properties. If more than one table is defined, the column number defined in any ELEMENT command must be the cumulative column number. In WIPS-DATA only a single table is defined. This combines the EMAT arrays for all element types used in the current model.

A11.5.2 MATL - MATERIAL DATA

Command Format:

MATL,NMAT,NITM

Variable	Format	Data
NMAT	I	Number of material sets to be specified.
NITM	I	Number of items in each material set (max. = 64; default = 8). In WIPS-DATA this is set to 64. If the length of the EMAT array for any element type is less than 64, it is padded with zeros.

FOLLOWING DATA

NMAT sets of cards, each consisting of as many cards as needed to define NITM items, 5 per card.

Columns	Variable	Format	Data
1 - 75	EMAT(K)	5E15.7	Material data, 5 items per card.

TABLE A11.5(a) - EMAT ARRAY FOR PIPE ELEMENT

1:	Number of yield surfaces for material (min. 1, max. 2).
2:	Poisson's ratio.
3-5:	Material moduli for up to 3 segments.
6-7:	Yield stresses.
8:	Not used.
9:	Radial error tolerance for material.
10:	Angle tolerance for material.
11:	Number of subelements in cross section.
12:	Number of Gauss slices. Usually 2.
13:	Ovalling code (0=small; 1=large).
14:	Number of ovalling hardening ratios (max. 2).
15:	Scale factor for ovalling stiffness. Typically 1.0.
16:	Scale factor for ovalling strength. Typically 1.0.
17:	Outside diameter.
18:	Wall thickness.
19:	Self-weight per unit length.
20:	Not used.
21-22:	Ovalling hardening ratios.
23:	Number of dashpot stiffnesses for material rate dependence (max. 3).
24-26:	Dashpot stiffnesses.
27-29:	Corresponding strain rate limits (last very large).
30:	**Factor controlling ovalling geometric stiffness. Enter 0.64.**

TABLE A11.5(b) - EMAT ARRAY FOR BEAM ELEMENT

1:	Yield surface type. Typically 1.
2:	Angle tolerance for stiffness reformulation.
3-6:	Yield surface exponents α_1 - α_4. Typically unused.
7:	Overshoot tolerance for yield event (proportion of Y_1).
8:	Reversal tolerance for unloading event (proportion of Y_1).
9-16:	K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3, GA' for M_{yy}.
17-24:	K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3, GA' for M_{zz}.
25-31:	K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3 for M_{xx}.
32:	Not used.
33-39:	K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3 for F_x.
40:	Strain rate tolerance.
41:	Number of dashpot stiffnesses for rate dependence (max. 3).
42-44:	Dimensionless rates for dashpot stiffness changes.
43-47:	Dimensionless strength increases.
48:	Weight per unit length.

TABLE A11.5(c) - EMAT ARRAY FOR UBAR ELEMENT

1:	Number of segments in static force-extension curve (NSEG). Min. 2; Max. 6.
2:	Number of segments in curve of force increase versus extension rate (NSEGV). Min. 0; Max. 3.
3:	Default gap clearance.
4:	Angle tolerance for stiffness reformulation.
5:	Static stiffnesses (NSEG values).
5+NSEG:	Static yield forces (NSEG values, last very large).
5+2*NSEG:	Dashpot stiffnesses (NSEGV values).
5+2*NSEG+NSEGV:	Dashpot "yield" forces (NSEGV values, last very large).
5+2*NSEG+2*NSEGV:	Overshoot tolerance for gap closure and yield events (force units).
6+2*NSEG+2*NSEGV:	Reversal tolerance for unloading event (force units).
7+2*NSEG+2*NSEGV:	Separation tolerance for separation event (force units).

TABLE A11.5(b) - EMAT ARRAY FOR BEAM ELEMENT

1: Yield surface type. Typically 1.

2: Angle tolerance for stiffness reformulation.

3-6: Yield surface exponents α_1 - α_4. Typically unused.

7: Overshoot tolerance for yield event (proportion of Y_1).

8: Reversal tolerance for unloading event (proportion of Y_1).

9-16: K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3, GA' for M_{yy}.

17-24: K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3, GA' for M_{zz}.

25-31: K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3 for M_{xx}.

32: Not used.

33-39: K_1, K_2, K_3, K_4, Y_1, Y_2, Y_3 for F_x.

40: Strain rate tolerance.

41: Number of dashpot stiffnesses for rate dependence (max. 3).

42-44: Dimensionless rates for dashpot stiffness changes.

43-47: Dimensionless strength increases.

48: Weight per unit length.

TABLE A11.5(c) - EMAT ARRAY FCʀ UBAR ELEMENT

1:	Number of segments in static force-extension curve (NSEG). Min. 2; Max. 6.
2:	Number of segments in curve of force increase versus extension rate (NSEGV). Min. 0; Max. 3.
3:	Default gap clearance.
4:	Angle tolerance for stiffness reformulation.
5:	Static stiffnesses (NSEG values).
5+NSEG:	Static yield forces (NSEG values, last very large).
5+2*NSEG:	Dashpot stiffnesses (NSEGV values).
5+2*NSEG+NSEGV:	Dashpot "yield" forces (NSEGV values, last very large).
5+2*NSEG+2*NSEGV:	Overshoot tolerance for gap closure and yield events (force units).
6+2*NSEG+2*NSEGV:	Reversal tolerance for unloading event (force units).
7+2*NSEG+2*NSEGV:	Separation tolerance for separation event (force units).

TABLE A11.5(e) - EMAT ARRAY FOR SHELL ELEMENT

1:	Number of yield surfaces.
2:	Number of strain rate segments.
3:	Radial error tolerance (multiple of yield stress).
4:	Angle tolerance for stiffness reformulation.
5:	Poisson ratio.
6:	Weight density.
7:	Strain rate tolerance (multiple of strain rate).
8:	Yield reversal tolerance (multiple of yield stress).
9:	Not used.
10-15:	Static moduli.
16-20:	Yield stresses.
21:	Not used.
22-24:	Strain rate stiffnesses.
25-27:	Stress limits for strain rate (last very large).
28-30:	Local r,s,t coordinates of output point 1.
31-33:	Local r,s,t coordinates of output point 2.

A11.6 PROPERTY TABLES

A11.6.1 GENERAL

The PROPERTY tables contain any desired element property data not contained in MATERIAL tables. Element stiffness and strength data will normally be placed in MATERIAL tables but may be placed in PROPERTY tables if preferred. PROPERTY tables are not set up by WIPS-DATA.

A11.6.2 PROPERTY - PROPERTY DATA

Command Format:

PROP,NPRP,NITM

Variable	Format	Data
NPRP	I	Number of property sets to be specified.
NITM	I	Number of items in each property set (max. = 16; default = 8).

FOLLOWING DATA

NPRP sets of cards, each consisting of as many cards as needed to define NITM items, 5 per card.

Columns	Variable	Format	Data
1 - 75	PROP(K)	5E15.7	Property data, 5 items per card.

A11.7 ELEMENT SPECIFICATION

A11.7.1 GENERAL

Elements must be arranged in *groups*. All elements in any group must be of the same type. All elements of a given type need not necessarily be in a single group, although this is done by WIPS-DATA. The groups may be specified in any convenient sequence.

A11.7.2 ELEMENTS - ELEMENT DATA

Command Format:

ELEMENTS,NUMG

Variable	Format	Data
NUMG	I	Number of element groups.

FOLLOWING DATA

NUMG sets of cards. Each set of data consists of one group control card plus one card per element.

(A) GROUP CONTROL CARD

Columns	Variable	Format	Data
1 - 5	IGRP	I	Element type number, as follows. 1 = beam 2 = U-bar 3 = pipe 4 = gap-friction 20 = shell
6 - 10	NELS	I	Number of elements in this group.
11 - 15	NFST	I	Number of first element in this group. Default = 1.
20	NKON	I	Element stiffness code: 0 = elements have different stiffnesses 1 = all elements have identical stiffnesses (in local element coordinates) Must be zero for nonlinear elements.
41 - 50	DAMP	R	Factor for initial stiffness dependent viscous damping. See INTEGRATE command for method of use. For most cases: 1.0 = viscous damping required (pipe, beam shell elements). 0.0 = not required (U-bar, gap-friction elements).

(B) ELEMENT CARDS

As many cards as needed to specify all elements in the group. The data varies with the element type. See the following sections.

(C) TERMINATION CARD

One blank card following the element cards.

A11.7.3 ELEMENT CARDS - BEAM ELEMENT

Three cards per element, unless generation is used.

(A) CARD 1

Columns	Variable	Format	Data
1 - 5	JFEL	I	Element number. NELS elements, starting with NFST, must be specified.
6 - 10	JLEL	I	Last element number in generated set. Default = JFEL (no generation).
11 - 15	JDIF	I	Element number increment for generated set. Default = 1.
16 - 20	JNOD	I	Node number increment for generated set. Default = 1.
22 - 25	JN(1)	I/A	Node number or name at element end i.
27 - 30	JN(2)	I/A	Node number or name at element end j.
31 - 35	IEIN(1)	I	Material number (column number in EMAT array).
40	IEIN(2)	I	Large displacements code (0 = no; 1 = yes).
45	IEIN(3)	I	Time history code, for saving on RSLT file (0 = no; 1 = yes).
50	IEIN(4)	I	Node K code (0 = default; 1 = specified).
55	IEIN(5)	I	Curved element code (0 = straight; 1 = curved). Not currently used.
60	IEIN(6)	I	Symmetry code (0 = 3D; 1 = local xy plane only, half element).

(B) CARD 2

Columns	Variable	Format	Data
1 - 15	REIN(1)	F	X coordinate of node K (if used).
16 - 30	REIN(2)	F	Y
31 - 45	REIN(3)	F	Z
46 - 60	REIN(4)	F	X coordinate of bend center, if a bend. Not currently used.

61 - 75	REIN(5)	F	Y

(C) CARD 3

Columns	Variable	Format	Data
1 - 15	REIN(6)	F	Z

A11.7.4 ELEMENT CARDS - PIPE ELEMENT

Two cards per element, unless generation is used.

(A) CARD 1

Columns	Variable	Format	Data
1 - 5	JFEL	I	Element number. NELS elements, starting with NFST, must be specified.
6 - 10	JLEL	I	Last element number in generated set. Default = JFEL (no generation).
11 - 15	JDIF	I	Element number increment for generated set. Default = 1.
16 - 20	JNOD	I	Node number increment for generated set. Default = 1.
22 - 25	JN(1)	I/A	Node number or name at element end i.
27 - 30	JN(2)	I/A	Node number or name at element end j.
31 - 35	IEIN(1)	I	Material number (column number in EMAT array).
40	IEIN(2)	I	Large displacements code (0 = no; 1 = yes).
45	IEIN(3)	I	Time history code, for saving on RSLT file (0 = no; 1 = yes).
50	IEIN(4)	I	Node K code (0 = default; 1 = specified).
55	IEIN(5)	I	Curved element code (0 = straight; 1 = curved).
60	IEIN(6)	I	Symmetry code (0 = 3D; 1 = local xy plane only, half element).

(B) CARD 2

Columns	Variable	Format	Data
1 - 15	REIN(1)	F	X coordinate of node K (if used) or bend center (if on a bend).
16 - 30	REIN(2)	F	Y
31 - 45	REIN(3)	F	Z
46 - 60	REIN(4)	F	Internal pressure.

A11.7.5 ELEMENT CARDS - UBAR ELEMENT

Three cards per element, unless generation is used.

(A) CARD 1

Columns	Variable	Format	Data
1 - 5	JFEL	I	Element number. NELS elements, starting with NFST, must be specified.
6 - 10	JLEL	I	Last element number in generated set. Default = JFEL (no generation).
11 - 15	JDIF	I	Element number increment for generated Default = 1.
16 - 20	JNOD	I	Node number increment for generated set. Default = 1.
22 - 25	JN(1)	I/A	Node number or name at element end i.
26 - 30	IEIN(1)	I	Material number (column number in EMAT array).
35	IEIN(2)	I	Large displacements code (0 = no; 1 = yes).
40	IEIN(3)	I	Time history code, for saving on RSLT file (0 = no; 1 = yes).
45	IEIN(4)	I	Node K code (0 = default; 1 = specified).
50	IEIN(5)	I	Symmetry code (0 = 3D; 1 = use half element only).

(B) CARD 2

Columns	Variable	Format	Data
1 - 15	REIN(1)	F	X coordinate of node J.
16 - 30	REIN(2)	F	Y
31 - 45	REIN(3)	F	Z
46 - 60	REIN(4)	F	X coordinate of node K (if used).
61 - 75	REIN(5)	F	Y

(C) CARD 3

Columns	Variable	Format	Data
1 - 15	REIN(6)	F	Z

A11.7.6 ELEMENT CARDS - GAPF ELEMENT

Three cards per element, unless generation is used.

(A) CARD 1

Columns	Variable	Format	Data
1 - 5	JFEL	I	Element number. NELS elements, starting with NFST, must be specified.
6 - 10	JLEL	I	Last element number in generated set. Default = JFEL (no generation).
11 - 15	JDIF	I	Element number increment for generated set. Default = 1.
16 - 20	JNOD	I	Node number increment for generated set. Default = 1.
22 - 25	JN(1)	I/A	Node number or name.
26 - 30	IEIN(1)	I	Material number (column number in EMAT array).
31 - 35	IEIN(2)	I	Time history code, for saving on RSLT file (0 = no; 1 = yes).
36 - 40	IEIN(3)	I	Symmetry code (0 = 3D; 1 = use half element).

(B) CARD 2

Columns	Variable	Format	Data
1 - 15	REIN(1)	F	X direction cosine of element z axis.
16 - 30	REIN(2)	F	Y direction cosine.
31 - 45	REIN(3)	F	Z direction cosine.
46 - 60	REIN(4)	F	X direction cosine of element x axis.
61 - 75	REIN(5)	F	Y direction cosine.

(C) CARD 3

Columns	Variable	Format	Data
1 - 15	REIN(6)	F	Z direction cosine.
16 - 30	REIN(7)	F	Initial gap.

A11.7.7 ELEMENT CARDS - SHELL ELEMENT

Two cards per element, unless generation is used.

(A) CARD 1

Columns	Variable	Format	Data
1 - 5	JFEL	I	Element number. NELS elements, starting with NFST, must be specified.
6 - 10	JLEL	I	Last element number in generated set. Default = JFEL (no generation).
11 - 15	JDIF	I	Element number increment for generated set. Default = 1.
16 - 20	JNOD	I	Node number increment for generated set. Default = 1.
22 - 25	JN(1)	I	Node name or number of element node 1. See Section C5 for node sequence.
27 - 30	JN(2)	I	Node 2.
	etc.		
57 - 60	JN(8)	I	Node 8
65	IEIN(1)	I	Material number (column number in EMAT array).
70	IEIN(2)	I	Large displacements code (0 = no; 1 = yes).
75	IEIN(3)	I	Time history code, for saving on RSLT file (0 = no; 1 = yes).
80	IEIN(4)	I	Number of integration points through thickness (min. 2, max. 7). Default = 2.

(B) CARD 2

Columns	Variable	Format	Data
1 - 15	REIN(1)	F	Initial axial stress (due to internal pressure). See Section C5 for procedure.
16 - 30	REIN(2)	F	Initial hoop stress.
31 - 45	REIN(3)	F	Internal pressure.

A11.8 NODE ORDERING

A11.8.1 NORDER - ORDER NODES AND NUMBER EQUATIONS

Command Format:

NORD,IORD,NSTR.

Variable	Format	Data
IORD	A	Node ordering indicator: Blank = no reordering; OPTO = reordering using a modified Reverse Cuthill-McKee algorithm (RC-M).
NSTR	I	Number of different nodes at which RC-M scheme is to be initiated. Default = every node.

FOLLOWING DATA

As many cards as necessary to define NSTR starting nodes. Omit if NSTR = 0.

Columns	Variable	Format	Data
1 - 80		16I5 or 16(1X,A4)	Names or number of starting nodes for RC-M ordering.

A11.8.2 PROFILE - FORM ACTIVE COLUMN PROFILE OF SUBSTRUCTURE STIFFNESS

Command Format:

PROFILE

A11.8.3 END OF SUBSTRUCTURE TYPE DATA

Insert END card (Section A11.4.2) at this point.

A11.8.4 END OF BUILDING PROCESS

If this is the last substructure, insert FINISH card (Section A11.3.3) at this point.

A11.9 ANALYSIS MODEL

A11.9.1 GENERAL

Before performing an analysis, an ANALYSIS MODEL must be specified. The ANALYSIS MODEL may be any one of the SUBSTRUCTURE TYPES, not necessarily the highest level substructure. Once the ANALYSIS MODEL has been specified, the program automatically forms a structural connectivity tree relating all lower level SUBSTRUCTURE TYPES to the ANALYSIS MODEL. The individual components in the structure connectivity tree are identified as ACTUAL SUBSTRUCTURES.

A11.9.2 MODEL - BEGIN MODEL SPECIFICATION

Command Format:

MODEL

A11.9.3 TREE - NAME OF ANALYSIS MODEL

Command Format: TREE,NTYP

Variable	Format	Data
NTYP	A	Name of substructure type which is to be the ANALYSIS MODEL. For WIPS this is always "MAIN".

A11.9.4 FINISH - END MODEL SPECIFICATION

Command Format:

FINISH

A11.10 MASS SPECIFICATION

A11.10.1 GENERAL

Masses may be defined for each SUBSTRUCTURE TYPE. The mass matrices for the ACTUAL SUBSTRUCTURES and the ANALYSIS MODEL are then constructed automatically.

The mass specification begins with an MBUILD command and ends with a FINISH command. For each SUBSTRUCTURE TYPE, the specification begins with a NEW command and ends with an END command.

A11.10.2 MBUILD - BEGIN MASS SPECIFICATION

Command Format:

MBUILD

A11.10.3 FINISH - END MASS SPECIFICATION

Command Format:

FINISH

A11.10.4 NEW - BEGIN SUBSTRUCTURE MASS DEFINITION

Command Format:

NEW,NMST

Variable	Format	Data
NMST	A	Name of SUBSTRUCTURE TYPE.

A11.10.5 END - END SUBSTRUCTURE MASS DEFINITION

Command Format:

END

A11.10.6 PMASS - NODAL MASSES

Command Format:

PMASS,NMC

Variable	Format	Data
NMC	I	Number of mass points.

FOLLOWING DATA - NMC cards as follows.

Columns	Variable	Format	Data
1 - 5	NODI	A/I	Name or number of node.
7 - 18	RX	F	Mass in X direction.
19 - 30	RY	F	Mass in Y direction.

31 - 42	RZ	F	Mass in Z direction.
43 - 54	RXX	F	Rotational mass about X axis.
55 - 66	RYY	F	Rotational mass about Y axis.
67 - 78	RZZ	F	Rotational mass about Z axis.

A11.11 FORCE RECORDS

A11.11.1 GENERAL

Any number of force records may be specified. The specification begins with a PTREC command, and ends with a FINISH command. Each individual record begins with a NEWREC command.

A11.11.2 PTREC - BEGIN FORCE RECORD SPECIFICATION

Command Format:

PTREC

A11.11.3 FINISH - END FORCE RECORD SPECIFICATION

Command Format:

FINISH

A11.11.4 NEWREC - SPECIFY FORCE RECORD

Command Format:

NEWREC,NAME,MODE,NVAL,NPRB,DT,PFAC

Variable	Format	Data
NAME	A	Name of this force record.
MODE	A	Data code. SING = force values at constant time intervals. PAIR = force-time pairs
NVAL	I	Number of values or pairs of values to be input.
NPRB	I	Blocking parameter = number of values in each block for this record. Default = 200.
DT	F	Time interval (if MODE = SING; otherwise blank).
PFAC	F	Scale factor by which all force values are multiplied. Default = 1.0.

FOLLOWING DATA

One format card, followed by as many cards as needed to define record.

(A) FORMAT

Columns	Variable	Format	Data
1 - 80	IFORM(20)	20A4	Input format. Default = (8F10.0).

(B) FORCE RECORD

As many cards as needed to define the force record.

Note: If MODE = PAIR, the force value must *precede* the corresponding time value.

A11.12 IMPACT DATA

A11.12.1 GENERAL

Include these cards only if an impact analysis is to be performed.

A11.12.2 IMPACT - IMPACT SURFACE CONTROL

Command Format:

IMPACT,NSURF

Variable	Format	Data
NSURF	I	Number of impact surface pairs.

FOLLOWING DATA

NSURF sets of cards. Each set consists of (1) an impact surface control card; (2) a primary surface control card; (3) two or more cards defining the primary surface node list; (4) a secondary surface control card; and (5) one or more cards defining the secondary surface node list.

(A) IMPACT SURFACE CONTROL CARD

Columns	Variable	Format	Data
2 - 5		A	Impact surface name.
10		I	Substructure naming code: 1: specify substructure type name. Names must be unique for this option. 0: specify actual substructure number (in tree for analysis model).
11 - 25		F	Friction coefficient.
26 - 40		F	Velocity tolerance for full friction coefficient.
41 - 55		F	Half-thickness of primary surface.
56 - 70		F	Half-thickness of secondary surface.

(B) PRIMARY SURFACE CONTROL CARD

Columns	Variable	Format	Data
2 - 5		A/I	Name or number of primary surface substructure.
10	ICOL	I	Number of mesh columns.
11 - 15	IROW	I	Number of mesh rows.

(C) PRIMARY SURFACE NODE LIST

ICOL+1 cards (or sets of cards), each defining IROW+1 node names or numbers, in 16(1X,A4) or 16I5 format.

(D) SECONDARY SURFACE CONTROL CARD

Columns	Variable	Format	Data
2 - 5		A/I	Name or number of secondary surface substructure.
6 - 10	NPTS	I	Number of secondary surface nodes.

(E) SECONDARY SURFACE NODE LIST

As many cards as needed to define NPTS node names or numbers, in 16(1X,A4) or 16I5 format.

A11.13 TIME HISTORY OUTPUT CONTROL

A11.13.1 OUTPUT RECORD

At each output time (sampling interval), an output buffer containing node, element, and impact results is set up and written as a single record on the RSLT file. The THOUT command contains first word address (FWA) information for storage of the results in the output buffer.

A11.13.2 THOUT - BEGIN OUTPUT CONTROL DATA

Command Format:

THOUT,LBUFO

Variable	Format	Data
LBUFO	I	Length of output buffer (single precision words). All output items are single precision.

FOLLOWING DATA

One card for main structure plus one card for each substructure (if any) plus one card for each impact surface pair (if any).

(A) MAIN STRUCTURE CARD

Columns	Variable	Format	Data
1 - 10		I	FWA in output buffer of results for first node.
11 - 20		I	FWA for first element.
21 - 25	I	Displ/veloc/accn code for node results:	1: node displacements only 2: displacements and velocities 3: displacements, velocities and accelerations
26 - 30		I	Rotation code for node results: 0: include rotations 1: omit rotations

(B) SUBSTRUCTURE CARDS

Substructures are numbered in the sequence that they appear in the tree for the analysis model (actual substructure number minus 1). For WIPS single-level substructuring, this is the order in which the substructures are defined in the WIPS-MODL phase.

One card for each substructure, containing the same data as CARD (A).

(C) IMPACT CARDS

One card for each impact surface pair.

Columns	Variable	Format	Data
1 - 10		I	FWA of data in output buffer.
11 - 15		I	Results type code: 1: 6 items per surface pair

A11.13.3 FWA CALCULATION

The LBUFO length and FWA values must be specified to allow sufficient storage for results, without overlapping. Node results for any substructure (including the main structure) are stored sequentially beginning at the node results FWA for that substructure. Element results are similarly stored sequentially.

A11.14 INITIATE NONLINEAR DYNAMIC ANALYSIS

A11.14.1 NDYN - BEGIN ANALYSIS

This command must be included.

Command Format:

NDYN

A11.14.2 VELO - OPTIONAL INITIAL VELOCITY SPECIFICATION

Command Format:

VELO,NSUBV,NAMTYP

Variable	Format	Data
NSUBV	I	Number of substructures (including main structure) with initial velocities.
NAMTYP	I	Substructure naming code: 0: Specify actual substructure number (in tree for analysis model). 1: Specify substructure type name. Names must be unique for this option.

FOLLOWING DATA

NSUBV sets of cards, each consisting of a control card plus one or more velocity cards.

(A) CONTROL CARD

Columns	Variable	Format	Data
2 - 5		I/A	Substructure number or name.
6 - 10	NVEL	I	Number of nodes with initial velocities.

(B) VELOCITIES - NVBL cards as follows.

Columns	Variable	Format	Data
1 - 5		A/I	Name or number of node.
7 - 18		F	X velocity
19 - 30		F	Y velocity
31 - 42		F	Z velocity
43 - 54		F	XX rotational velocity

55 - 66	F	YY velocity
67 - 78	F	ZZ velocity

A11.14.3 MASS - MASS ASSEMBLY COMMAND

This command must be included.

Command Format:

MASS

A11.14.4 DYLOAD - DYNAMIC LOADS

Dynamic loads may be applied on main structure.

Command Format:

DYLOAD,NAMTYP,NLOAD

Variable	Format	Data
NAMTYP	I	Actual substructure number (main structure = 1).
NLOAD	I	Number of loaded points.

FOLLOWING DATA

NLOAD sets of cards. Each set consists of one control card plus one or more cards defining a list of nodes.

(A) CONTROL CARD

Columns	Variable	Format	Data
1 - 4		A	Name of force record.
6 - 10	IDIR	I	Direction code: 0: follows force 1: global X 2: Y 3: Z
11 - 20		F	Time delay before loading arrives at node.
21 - 30		F	Scale factor by which force values are multiplied.
31 - 35	NNODS	I	Number of nodes in following list.

(B) NODE LIST

Enough cards to define NNODS node, names or numbers, 16 per card in 16(1X,A4) or (16I5) format. If IDIR=0, nodes are considered in *pairs.* The first node of each pair is the loaded node, and the force is directed towards the second node. For IDIR>0, all nodes in the list are loaded nodes.

A11.14.5 TIMO - TIME HISTORY OUTPUT CONTROL

Command Format:

TIMO,IFIL,INT,TIM

Variable	Format	Data
IFIL	A	Output file identifier: RSLT: output to RSLT file PRIN: output to ECHO file BOTH: output to RSLT and ECHO files
INT	I	Output step interval.
TIM	F	Output time interval.

Note: Results are output every INT time steps or TIM seconds, whichever occurs first.

A11.15 COMMANDS TO CONTROL ANALYSIS

A11.15.1 GENERAL

The analysis is controlled by a series of commands of the type described in this section. For the actual sequence used, see the commands generated in a typical DATA file.

A11.15.2 ZERO - ZERO AN EXISTING ARRAY

Command Format:

ZERO,LNAM

Variable	Format	Data
LNAM	A	Name of array to be set to zero.

A11.15.3 STEP - FORM CURRENT LOAD VECTOR

Command Format:

STEP

A11.15.4 UNBALANCE - FORM UNBALANCED LOAD VECTOR

Command Format:

UNBA,CHEC,TOL1,TOL2

Variable	Format	Data
CHEC	A	If CHEC is present, the unbalance norm is checked against TOL1 and TOL2.
TOL1	F	Tolerance for time step decrease. If exceeded, the time step is reduced and the step repeated.
TOL2	F	Tolerance for termination. If exceeded, the execution stops.

A11.15.5 AUTO - AUTOMATIC STEPPING

A11.15.5.1 AUTO,DEFINE - Initialize Control Files

Command Format:

AUTO,DEFI

A11.15.5.2 AUTO,INIT - Initialize Control Parameters

Command Format:

AUTO,INIT

A11.15.5.3 AUTO,LOAD

Multiply load vector by remaining step factor.

Command Format:

AUTO,LOAD

A11.15.5.4 AUTO,MERR - Midstep Error Check

Command Format:

AUTO,MERR,TOLL,TOLU

Variable	Format	Data
TOLL	F	Lower tolerance. If the midstep error norm is below this tolerance for two consecutive steps, the time step is increased.
TOLU	F	Upper tolerance. If the midstep error norm exceeds this value, the time step is reduced and the step repeated.

A11.15.5.5 AUTO,FACTOR

Multiply displacement increment by event factor.

Command Format:

AUTO,FACT

A11.15.5.6 AUTO,STEP - Automatic Step Size Determination

Command Format:

AUTO,STEP,SLW,SUP,SCL,SCU

Variable	Format	Data
SLW	F	Minimum step size.
SUP	F	Maximum step size.
SCL	F	Factor for step reduction (default = 0.5).
SCU	F	Factor for step increase (default = 2.0).

A11.15.5.7 AUTO,EXIT - Check Analysis Completion

Command Format:

AUTO,EXIT,ETIM

Variable	Format	Data
ETIM	F	Exit time. If analysis time exceeds this value, execution ceases.

A11.15.6 STIFFNESS - STIFFNESS FORMATION

Command Format:

STIF,ALPH

Variable	Format	Data
ALPH	A	Stiffness formation code: Blank: modify and decompose structure stiffness. UPDA: save current structure stiffness in back-up storage. INIT: restore stiffness to that in back-up storage.

A11.15.7 LOAD - FORWARD REDUCTION LOAD VECTOR

Command Format:

LOAD

A11.15.8 DISPLACEMENT

Backward Substitution for Displacement Increment.

Command Format:

DISP

A11.15.9 KINRES - KINEMATIC RESPONSE

Command Format:

KINR,I1,I2,I3,I4,I5,I6

Variable	Format	Data
I1-I6	I	Codes indicating tasks to be performed (in the order of input): 1: update displacements, velocities and accelerations (the kinematic vectors). 3*: save current vectors. 4: print current vectors. 5: set up initial step value for velocity and acceleration. 9: update slaving constraints. 10: calculate velocity increment for midstep error computation. 11: output to RSLT file. 12*: initialize temporary kinematic vectors.

*can only occur singly or a the end of a command string

A11.15.10 ELMRES - ELEMENT TASK EVALUATION

Command Format:

ELMR,I1,I2,I3,I4,I5,I6

Variable	Format	Data
I1-I6	I	Codes indicating tasks to be performed (in the order of input): 1: element state determination. 2: internal resisting force calculation. 3: save current element state. 4: print current element state. 5: restore element state to that in back-up storage. 6: calculate dynamic step load due to damping. 9: event factor calculation. 10: midstep error calculation. 11: output to RSLT file.

A11.15.11 INTEGRATE - INTEGRATION SCHEME SPECIFICATION

Command Format:

INTE,ISCHM,AUTO,DT,BETA,DELTA,ALFA

Variable	Format	Data
ISCHM	A	Integration scheme: NEWM: Newmark scheme HILB: Hilber-Hughes-Taylor scheme
AUTO	A	Optional time step code. If absent, time step = DT. The remaining command parameters are used to initialize the integration scheme. If present, time step as determined by the AUTO,STEP command is used. The remaining command parameters, if any, are ignored.
DT	F	Time step.
BETA	F	Beta value for Newmark scheme. Default = 0.25.
DELTA	F	Delta value for Newmark scheme.
ALFA	F	For HILB option: dissipation factor alpha (a negative value). For NEWM option: value of BETA-O/DT, where BETA-O = initial stiffness damping factor (positive).

A11.15.12 LDSWAP

Copy load vector from back-up storage.

Command Format:

LDSW

This is necessary when repeating a time step using the HILB integration option.

A11.15.13 OUTPUT TO RSLT FILE

A11.15.13.1 CHOUT - Check if Output Interval

Command Format:

CHOU

A11.15.13.2 WROUT - Write Output Buffer to RSLT File

Command Format:

WROU

A11.15.14 SUMMARY - ANALYSIS SUMMARY

Command Format:

SUMM

A11.15.15 BRANCHING OPERATIONS

A11.15.15.1 ENTRY - ENTRY POINT

Command Format:

ENTR,I1

Variable	Format	Data
I1	I	Entry Number

A11.15.15.2 GOTO - GOTO Command

Command Format:

GOTO,I1

Variable	Format	Data
I1	I	Entry number to go to.

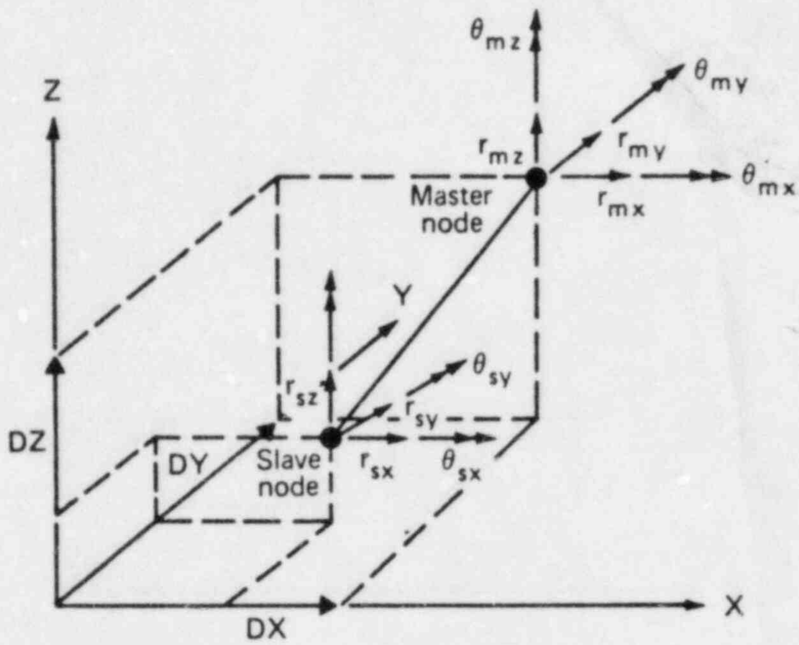
A11.15.15.3 IF - Multiple Branching

Command Format:

IF,IFLG,I1,I2,I3

Variable	Format	Data
IFLG	I	Flag number to be checked.

I1	I	If flag is negative, control is transferred to this ENTRY number.
I2	I	If flag is zero, control is transferred to this ENTRY number.
I3	I	If flag is positive, control is transferred to this ENTRY number.



SLAVING CODE						DISPLACEMENT CONSTRAINT
r_{sx}	r_{sy}	r_{sz}	θ_{sx}	θ_{sy}	θ_{sz}	
1	0	0	0	0	0	$r_{sx} = r_{mx} + \theta_{mz} \times DY - \theta_{my} \times DZ$
0	1	0	0	0	0	$r_{sy} = r_{my} + \theta_{mx} \times DZ - \theta_{mz} \times DX$
0	0	1	0	0	0	$r_{sz} = r_{mz} + \theta_{my} \times DX - \theta_{mx} \times DY$
0	0	0	1	0	0	$\theta_{sx} = \theta_{mx}$
0	0	0	0	1	0	$\theta_{sy} = \theta_{my}$
0	0	0	0	0	1	$\theta_{sz} = \theta_{mz}$

FIG. A11.1 - SLAVING CONSTRAINT

NRC FORM 335 (7-77)

U.S. NUCLEAR REGULATORY COMMISSION

BIBLIOGRAPHIC DATA SHEET

1. REPORT NUMBER (Assigned by DDC)	NUREG/CR-3686, Vol. 1 UCRL-15597
4. TITLE AND SUBTITLE (Add Volume No., if appropriate)	WIPS--Computer Code for Whip and Impact Analysis of Piping Systems -- Part A - User's Manual
2. (Leave blank)	
3. RECIPIENT'S ACCESSION NO.	
7. AUTHOR(S)	Graham H. Powell et al*
5. DATE REPORT COMPLETED	MONTH: March; YEAR: 1983
9. PERFORMING ORGANIZATION NAME AND MAILING ADDRESS (Include Zip Code)	Lawrence Livermore National Laboratory Post Office Box 808, L-46 Livermore, California 94550 *Subcontractor: University of California Berkeley, CA
DATE REPORT ISSUED	MONTH: June; YEAR: 1984
6. (Leave blank)	
8. (Leave blank)	
12. SPONSORING ORGANIZATION NAME AND MAILING ADDRESS (Include Zip Code)	Division of Engineering Technology Office of Nuclear Regulatory Research U.S. Nuclear Regulatory Commission Washington, D.C. 20555
10. PROJECT/TASK/WORK UNIT NO.	
11. CONTRACT NO.	A0383-3
13. TYPE OF REPORT	Technical
PERIOD COVERED (Inclusive dates)	
15. SUPPLEMENTARY NOTES	
14. (Leave blank)	

16. ABSTRACT (200 words or less)

WIPS (Whip and Impact of Piping Systems) is a special purpose computer code for the structural analysis of pipe whip dynamic effects following a postulated pipe rupture. WIPS has been developed primarily to provide support for the pipe whip analysis procedures described in Section 3.6.2 of the U.S. Nuclear Regulatory Commission Standard Review Plan.

This report summarizes the purpose and scope of the WIPS development effort, identifying those clauses in the standard Review Plan which refer to pipe whip analysis, and indicating how the WIPS code can be used to provide supporting data. Detailed information on use of the code is contained in accompanying reports which cover (1) use instructions, (2) theory, (3) programming procedures, and (4) verification examples.

17. KEY WORDS AND DOCUMENT ANALYSIS	pipe whip analysis structural analysis
17a. DESCRIPTORS	WIPS Code
17b. IDENTIFIERS/OPEN-ENDED TERMS	
18. AVAILABILITY STATEMENT	Unlimited
19. SECURITY CLASS (This report)	Unclassified
20. SECURITY CLASS (This page)	
21. NO. OF PAGES	
22. PRICE	$

NRC FORM 335 (7-77)

UNITED STATES
NUCLEAR REGULATORY COMMISSION
WASHINGTON, D.C. 20555

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, $300

FOURTH CLASS MAIL
POSTAGE & FEES PAID
USNRC
WASH. D. C.
PERMIT No. G 67

120555078877 1 IAN1R4
US NRC
ADM-DIV OF TIDC
POLICY & PUB MGT BR-PDR NUREG
W-501
WASHINGTON DC 20555