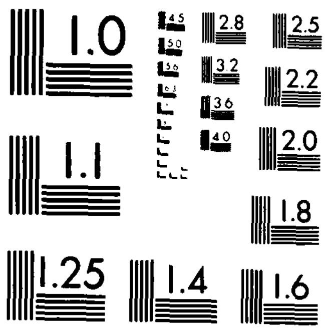
AD-A149 196 USER-ORIENTED COMPUTER-AIDED HYDRAULIC SYSTEM DESIGN 1/2
(U) OKLAHOMA STATE UNIV STILLWATER FLUID POWER RESEARCH
CENTER I T HONG JUN 83 OSU-FPRC-83-A-F1
UNCLASSIFIED DAAK70-81-C-0042 F/G 13/7 NL



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963 A

(2)

AD-A149 196

Report No. FPRC 83-A-F1

USER-ORIENTED COMPUTER-AIDED HYDRAULIC SYSTEM DESIGN

FINAL REPORT

JUNE, 1983

PREPARED FOR

U.S. ARMY MOBILITY EQUIPMENT RESEARCH
AND DEVELOPMENT COMMAND
FORT BELVOIR, VIRGINIA 22060

PREPARED BY

PERSONNEL OF THE
FLUID POWER RESEARCH CENTER
OKLAHOMA STATE UNIVERSITY
STILLWATER, OKLAHOMA

CONTRACT NUMBER
DAAK70-81-C-0042

The views, opinions, and/or findings contained in the report are those of the author(s) and should not be construed as an official Department of the Army position, policy, or decision, unless so designated by other documentation.

Report No. FPRC 83-A-F1

USER-ORIENTED COMPUTER-AIDED HYDRAULIC SYSTEM DESIGN

FINAL REPORT

JUNE, 1983

PREPARED FOR

U.S. ARMY MOBILITY EQUIPMENT RESEARCH
AND DEVELOPMENT COMMAND
FORT BELVOIR, VIRGINIA 22060

PREPARED BY

PERSONNEL OF THE
FLUID POWER RESEARCH CENTER
OKLAHOMA STATE UNIVERSITY
STILLWATER, OKLAHOMA

CONTRACT NUMBER
DAAK70-81-C-0042

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

1. REPORT NUMBER: FPRC 83-A-F1
2. GOVT ACCESSION NO.:
3. RECIPIENT'S CATALOG NUMBER:
4. TITLE (and Subtitle): USER ORIENTED COMPUTER AIDED HYDRAULIC SYSTEM DESIGN
5. TYPE OF REPORT & PERIOD COVERED: Final Technical Report April 1981 to ~~June 1983~~ MARCH 1983
6. PERFORMING ORG. REPORT NUMBER:

7. AUTHOR(s): Hong, I. T.

8. CONTRACT OR GRANT NUMBER(s): DAAK70-81-C-0042

9. PERFORMING ORGANIZATION NAME AND ADDRESS:
Fluid Power Research Center
Oklahoma State University
Stillwater, OK 74078

10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS:

11. CONTROLLING OFFICE NAME AND ADDRESS:
U.S. Army Mobility Equipment
Research & Development Command Procurement &
Production Office / Ft. Belvoir, VA 22060

12. REPORT DATE: June 1983
13. NUMBER OF PAGES: 174

14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office):
ONRRP, Rm. 582, Federal Bldg.
300 E. 8th St.
Austin, TX 78701

15. SECURITY CLASS (of this report): Unclassified
15a. DECLASSIFICATION DOWNGRADING SCHEDULE:

16. DISTRIBUTION STATEMENT (of this Report)

Approved for Public Release: Distribution Unlimited

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

hydraulic system	computer aided design	user oriented
system simulation	power flow modeling	problem oriented language
transient state	steady state	valves
FORTRAN	PL/I	pumps
	motors	cylinders

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

This report presents the results of the first two phases of the development of a user oriented computer aided hydraulic system design package. The basic system analysis and simulation methodologies, a problem oriented language for use with the developed program, and the models of commonly used hydraulic valves, pumps, motors, and cylinders are presented. The results of the empirical verification of the valve models are presented and discussed. A program user's manual is included.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

SUMMARY

This report presents the results of two years of work under U.S. Army MERADCOM Contract No. DAAK70-81-C-0042. The purpose of this effort was to develop a user oriented computer aided analysis and simulation package for hydraulic systems. The project covered two years and consisted of two discrete units of work termed Phase I and Phase II.

The specific objectives of Phase I were to:

1. Develop the basic system analysis program with its associated numerical analysis package.
2. Develop a Problem Oriented Language.
3. Develop models for commonly used hydraulic valves.

The specific objectives of Phase II were to:

1. Verify the valve models developed in Phase I by testing selected valves.
2. Develop models for commonly used hydraulic pumps, motors, and cylinders.
3. Extend the Problem Oriented Language to include hydraulic pumps, motors, and cylinders.

The results of Phase I were reported initially in Fluid Power

Research Center Report No. 82-A-I1, User-Oriented Computer-Aided System Design, Interim Report dated May, 1982.

All of the objectives of the project have been met successfully.

The work completed to-date represents only a portion of the total proposed project. While the individual models can be used to analyze individual components, the capability to analyze an entire hydraulic system still does not exist. The development of this full system analysis capability is the ultimate objective of the project and requires additional work.

It is highly recommended that this additional work be funded immediately in order to preserve the corporate experience of the project team and to ensure maximum benefits from any future efforts.

PER CALL JC

A-1

PREFACE

This report was prepared by the staff of the Fluid Power Research Center (FPRC), Oklahoma State University under the direction of Dr. E. C. Fitch. The work reported here was authorized by U.S. Army MERADCOM Contract No. DAAK70-81-C-0042. The report documents the work completed under Phases I and II of the subject contract covering the period 1 April 1981 to 31 March 1983.

The principal investigator for this effort was Dr. I. T. Hong, Research Engineer at the FPRC. Project personnel were:

G. Ball

K. Izawa

D. Ong

Z. He

W. Hensley

The Contract Officer Technical Representative for this contract was Mr. Delmar Craft.

The effort reported here represents the first two years of a multi-year project. While individual components can be analyzed using the models developed in these two years, the ultimate objective of a full system analysis capability cannot be met until the total proposed package is completed.

LIST OF FIGURES

Figure		Page

TABLE OF CONTENTS

CHAPTER I

INTRODUCTION

The Computer-Aided Analysis and Simulation (CAAS) package is the result of a multi-phased program to develop a user-oriented computer aided techniques for the analysis and design of hydraulic power transmission systems. The entire program provides a computer package complete with system analysis, optimization capabilities, component catalogue, Problem Oriented Language (POL), and operator prompting as well as the ability to consider contamination sensitivity and changing fluid parameters. The package essentially eliminates the requirement for any detailed knowledge of computer programming on the part of the system designer and allows effective use of the technique by anyone proficient with the POL.

The first phase of the CAAS program was completed during the period of April 1981 to March 1982. It led to the development of the general system analysis program, the adaptation of an existing numerical simulation package to this purpose, the development of the Problem Oriented Language, and the modelling of the family of commonly used hydraulic valves. These activities provided the theoretical basis to achieve the objectives of the second phase.

The activity of the second phase included testing a selected group of valves to verify the models developed during Phase I, developing models for commonly used pumps, hydraulic motors, and cylinders, and extending the Problem Oriented Language to include those three types of components.

To validate the developed valve models, a series of tests have been performed which include both the static and dynamic performance tests. Nine hydraulic control valves were selected for this purpose. Along with the experimental activities, each of the selected valve simulation procedures used in the CAAS system have been verified. This assures the designer that a model which has been simulated will indeed perform as predicted.

In general, a hydraulically powered machine consists of three basic elements. These are the power control element, the power conversion element, and the external element.

The action of the power control elements modulates the rate, direction, and level of energy transmitted by the power output elements. The performance of these components was comprehensively investigated during Phase I. The developed component models were modified and verified experimentally during this phase.

Pumps, motors, and cylinders are the typical power conversion elements. They convert energy for the external elements in order to achieve the desired task. The energy transfer and conversion are controlled by the control elements. The characteristics of the power conversion elements influence the operation of a hydraulically powered machine. Thus, the overall characteristics of a power conversion element are extremely important to the satisfactory operation of the machine.

One of the important aspects of the CAAS program is that it simplifies the communications between the user and the computer system. In order to develop a user oriented computer-aided design package, normally requires a highly conversational procedure for communication between the user and the computer, and an effective numerical analysis procedure to simulate the mathematical models.

During the first phase of this program, efforts were made to select proper program languages that could meet both requirements. It was found that the PL/I language has rich functions for manipulating character commands and the FORTRAN language has functions for analyzing mathematical problems. Both languages are comprehensively used around the world. Therefore, the PL/I language was selected to form the POL and the FORTRAN language was used to do mathematical calculations in Phase I.

The package which combined both the PL/I and the FORTRAN together has been proved very powerful to meet the specification of the CAAS program on the IBM 370 computer at Oklahoma State University. Although this approach was successful, it was recognized that the combination of two different program languages may decrease the program portability and increase the effort of maintenance if the package is implemented on different computers. With this in mind, the entire CAAS program was completely rewritten in the FORTRAN language in the second phase. Many innovative subprograms were developed which greatly improve the character manipulation capability of the FORTRAN language; thus it maintains the merits that were provided by using the PL/I.

This report delineates the results of the project to date. It presents the experimental verification of the developed hydraulic valve models, the theoretical background of the development of hydraulic pumps, motors, and cylinders, and the description of the updated CAAS package. In addition, a user's manual is presented in Appendix A. To assist the system programmers in maintaining or updating the CAAS package, a CAAS maintainer's manual is furnished and published separately from this report.

CHAPTER II

THEORETICAL DEVELOPMENT OF COMPONENT MODELS

The theoretical basis of analyzing a hydraulic component was studied in Phase I. It concluded that the static and the dynamic characteristics of a hydraulic component/system are governed by the principles of energy conservation and momentum conservation. In other words, it states that the net mass flow rates through a control volume equal the difference of mass flow rates between the inlet and outlet of the system (energy conservation) and the forces act in equilibrium on the system (momentum conservation).

Pumps, motors, and cylinders are hydraulic components. Consequently, the theories developed in the first phase of this program can be applied to analyze the performance characteristics of pumps, motors, and cylinders; however, because the function of any specific component is different from others, the performance variables normally are different. Thus, the performance model of each specific component should be addressed individually.

The following is a description of the function and theoretical development of the models for hydraulic pumps, motors, and cylinders. It is intended to develop general performance models for pumps, motors, and cylinders individually in this report. The detail analysis of each individual component is illustrated in the sub-programs of components (see the CAAS Maintainer's Manual).

HYDRAULIC PUMPS

Hydraulic pumps are used to convert mechanical energy to hydraulic energy. In general, there are two types of pumping mechanisms for producing fluid flow in a hydraulic system: non-positive and positive.

Turbine, centrifugal, and jet pumps are typical examples of non-positive displacement pumps. Because they are not commonly used in the hydraulic control applications, these types of machines are not discussed in this study.

The positive displacement pumps are the major power transfer devices in a fluid power control system because they have better performance and higher efficiency than the hydrodynamic devices. Except in rare application, rotary rather than the linear pumps are used in hydraulic systems.

Rotary hydraulic pumps can be classified according to the functional design. Gear, vane, and piston pumps are the most commonly used units. Furthermore, pumps can also be classified according to the fluid delivery characteristic; either a fixed-displacement or a variable-displacement unit. Based on the above discussion, the types of pumps employed in this study are outlined as shown in Fig. 2.1.

Pumps
- (1) Gear
- (2) Vane
  - (1) Fixed
    - (1) Balanced
    - (2) Unbalanced
  - (2) Variable
    - (1) Pressure feedback
    - (2) Servo Piston
- (3) Axial Piston
  - (1) Fixed
    - (1) Swashplate
    - (2) Bent Axis
  - (2) Variable
    - (1) Swashplate
    - (2) Bent Axis
- (4) Radial Piston
  - (1) Fixed
  - (2) Variable

Figure 2.1. Classification of Positive Displacement Hydraulic Pumps

FIXED DISPLACEMENT PUMP MODEL

The fixed-displacement pumps provide a constant delivery to the system. The pump is driven by a prime mover. In general, the mass of the pump rotating mechanisms is negligibly small as compared to the mass of the prime mover. Consequently, it is a reasonable approach that the dynamic characteristics of this type of pumps are not taken into consideration.

In his static pump performance model, Wilson indicated that the pump delivery is a resultant of the ideal delivery due to geometrical features, the loss due to cavitation at inlet. And the torque required to drive the pump is the total effect of the ideal torque due to the pressure differential and physical size, the resisting torque due to viscous drag of the fluid and the resisting torque of mechanical friction. These relationships are represented as:

$$Qp = Dp \cdot Np - Cs - \frac{Dp \cdot \Delta Pp}{2\pi\mu} - Qr \qquad (2.1)$$

$$Tp = \frac{Dp \cdot \Delta Pp}{2\pi} + Cd \cdot Dp \cdot \mu \cdot Np + Cf - \frac{Dp \cdot \Delta Pp}{2\pi} + Tc \qquad (2.2)$$

where Qp pump delivery flow rate

Dp pump displacement

Np rotation speed

Cs slip coefficient

ΔPp pressure differential

μ	fluid viscocity
Qr	delivery loss due to cavitation
Tp	torque required to drive pump
Cd	viscous drag coefficient
Cf	dry friction coefficient
Tc	resisting torque of mechanical friction

The slip coefficient is proportional to the value of the clearance ratio while the viscous drag coefficient is inversely proportional to the clearance. The delivery loss due to the viscous drag effect is caused by the journal operating concentrically in a bearing with a full, laminar flow oil film. The frictional loss is caused by direct contact of two moving surfaces, for instance, the metal to metal contact or the oil-seal contact around the pump shaft.

VARIABLE DISPLACEMENT PUMPS MODEL

A variable displacement pump generates a fluid delivery rate from zero to the maximum designed output. Although the speed of the pump remains fixed, the variable delivery rate can be obtained with the aid of a feedback compensation mechanism. A common displacement control is accomplished with the pressure compensating mechanism which produces just enough flow to meet the demand of the load.

The pressure compensating mechanism is schematically drawn in Fig. 2.2. It consists of a three way pressure compensator and an unequal area piston which controls the displacement of the pump.

When load pressure exceeds the preset spring force, the piston head side chamber is opened to the drain. This causes the piston to move upward, thus, it decreases the displacement of the pump. When load pressure decreases, the compensator displaces right. The piston chamber is opened to the high pressure side. As a result, the piston moves downward to increase the displacement of the pump. The process continues till an equilibrium condition is obtained.

The mathematical representation of the pressure compensated pump is represented as follows:

Flow equations for the compensator are:

$$Q_s = C_d W_T X_c \sqrt{\frac{2}{\rho}(P - P_s)} \tag{2.3}$$

$$Q_T = -C_d W_T X_c \sqrt{\frac{2}{\rho}(P_s - P_T)} \tag{2.4}$$

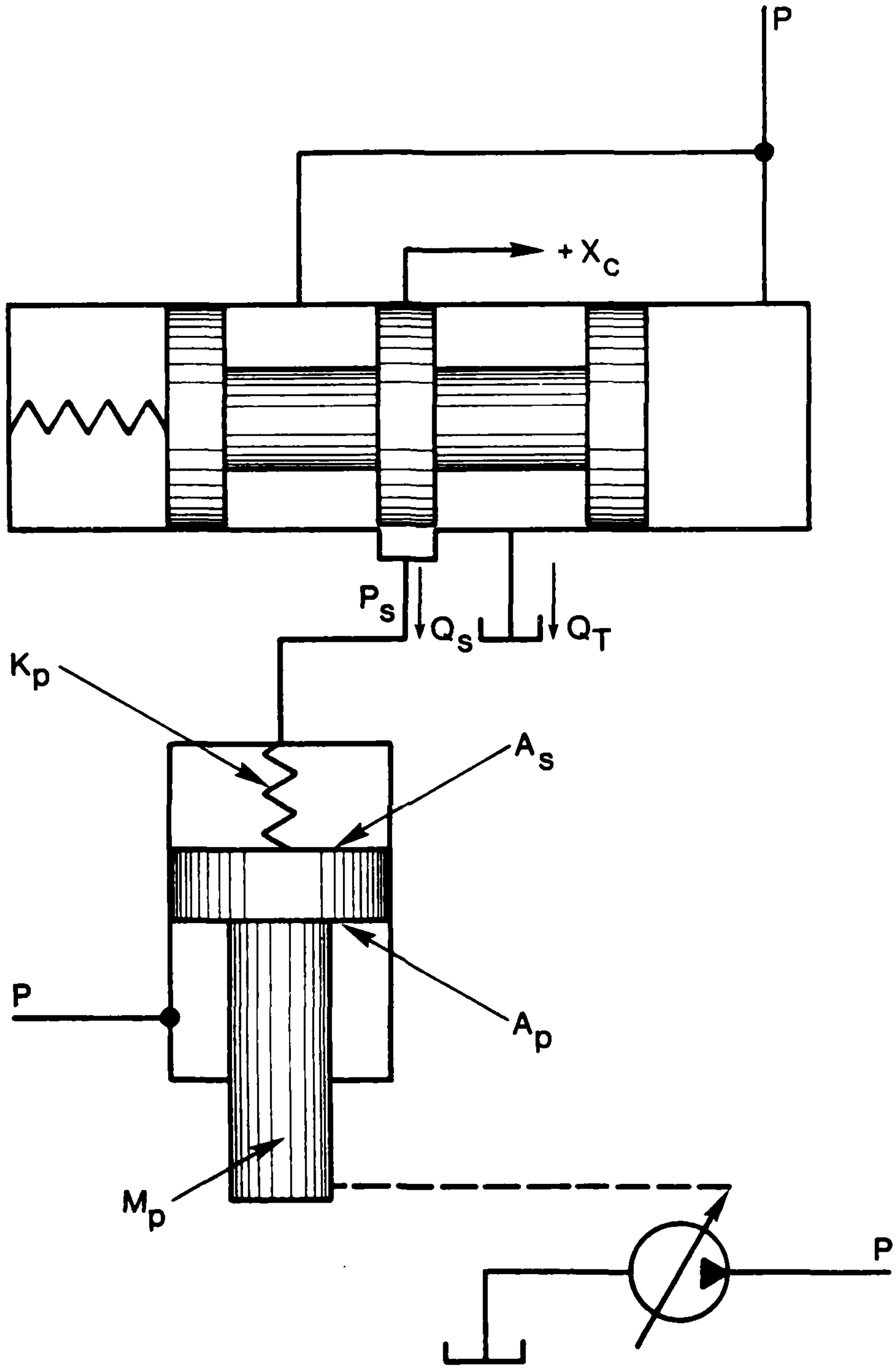


Fig. 2.2. Pressure Compensated Mechanism of Variable Displacement Pump

where	Q_s	flow to the piston chamber
	Q_T	flow to the drain
	C_d	discharge coefficient of the orifice
	W_s	area gradient of the high pressure side orifice
	W_T	area gradient of the drain side orifice
	X_c	the displacement of the spool
	ρ	fluid density
	P	load pressure
	P_s	pressure in the piston chamber
	P_T	pressure at the drain port

Force balance equations for the spool are:

$$PA_c = M_c \ddot{X}_c + B_c \dot{X}_c + K_c (X_o + X_c) + 0.43\, W_s X_c (P - P_s) \quad \text{When } X_c \geq 0$$

(2.5)

$$PA_c = M_c \ddot{X}_c + B_c \dot{X}_c + K_c (X_o + X_c) + 0.43\, W_T X_c (P_s - P_T) \quad \text{When } X_c < 0$$

where Ac the reaction area of the spool

Bc viscous damping of thc spool

Mc mass of the spool

Kc *spring constant*

Xc spring preset

Assuming that the flow due to the compensator displacement is negligible and that the compressibility of fluid is absent, thus, the continuity equation of the piston chamber becomes:

$$Q_s = A_s \dot{X}_p + C_l (P_s - P_T) \tag{2.6}$$

where As head side of piston reaction area

Cl leakage coefficient of the piston

The force balance equation for the piston is:

$$P_s A_s + K_p (X_o - X_p) = M_p \ddot{X}_p + B_p \dot{X}_p + P A_p \tag{2.7}$$

where A_p reaction area of rod side of the piston

K_p spring constant

X_o the preset displacement of spring

M_p mass of the piston with total pump displacement mechanism

B_p viscous damping coefficient of piston

Eqs. (2.3) to (2.7) describe the pressure compensation mechanism completely. Since Eqs. (2.3), (2.4), and (2.5) are nonlinear, a computer-aided analysis is needed to obtain the solution.

Furthermore, to attain flow response, the pump displacement is expressed in a linear function in terms of the piston displacement as shown in the following equation:

$$D_p = C X_p \tag{2.8}$$

where C is the displacement gradient of pump stroke control.

Pump flow rate is finally, calculated by using the Wilson model as illustrated in the fixed displacement pump model, Eq. (2.1).

HYDRAULIC MOTORS

A hydraulic motor is designed such that it receives the hydraulic power and converts it into mechanical power. Hydraulic motors, like pumps, can be classified as three major types: gear, vane, and piston. Fig. 2.3 illustrates the classification of motors used in the CAAS program.

Motor (1) Gear - fixed

(2) Vane - fixed

(3) Axial Piston - fixed

(1) Swash Plate

(2) Bent Axis

(4) Radial Piston - fixed

Figure 2.3. Classification of Hydraulic Motors

The characteristics of hydraulic motors are expressed based on the Wilson model in this study. In general, the mass of the motor is small compared to the load; therefore, it is practical to include it in the load mass. Thus, the force equations and flow equations are expressed as follows:

The theoretical torque generated:

$$T_{TH} = \frac{D_m \Delta P_m}{2\pi} \tag{2.9}$$

The loss due to viscous drag of fluid and dry friction is:

$$T_{Loss} = C_d D_m \mu N_m + C_f \frac{D_m \Delta P_m}{2\pi} \tag{2.10}$$

Thus, the force equilibrium equation is:

$$J \frac{d}{dt}(N) = T_{TH} - T_{Loss} - T_L \tag{2.11}$$

where TL is external load torque

J is polar moment of inertia of load

The flow equations are:

$$\frac{V}{2\beta} \frac{dP_a}{dt} = Q_a - D_m N_m - C_{sm} D_m \frac{\Delta P_m}{\mu} \tag{2.12}$$

$$\frac{V}{2\beta} \frac{dP_b}{dt} = Q_b + D_m N_m + C_{sm} D_m \frac{\Delta P_m}{\mu} \tag{2.13}$$

where V is total entrained volume in the motor

β is fluid bulk modulus

Pa,b is pressure in each motor chamber

Qa,b is flow rate into or out from the chamber.

Solving Eqs. (2.4) through (2.13) gives a dynamic response of the motor and load combination.

HYDRAULIC CYLINDERS

Hydraulic cylinders are widely used to convert the hydraulic energy into a linear motion mechanical energy. Basically, a cylinder consists of a movable element that travels in a cylinder bore. There are many different designs of cylinders to meet the requirement of specific functions. It is very difficult and impractical to classify cylinders in great detail. In order to develop a general model for hydraulic cylinders in this study, cylinders are classified based on their operational performance. Fig. 2.4. shows the various types of cylinders used in the CAAS program. As can be seen, the study was limited to linear and the rotary type actuators. It is noted that the rotary type mentioned here is focused on the cylinder that produces a finite angular displacement; that is, a positional control unit. This is different from the function of most rotary motors which are used to control rotating speed. The selection of this type of cylinder is merely for user's convenience so that he may directly employ the model to obtain the mechanism of generating an angular displacement.

CYLINDERS

- (1) Linear type
  - (1) Single acting
    - (1) One side rod
      - (1) Piston
      - (2) Plunger
      - (3) Spring return
    - (2) Telescopic
  - (2) Double acting
    - (1) One side rod
      - (1) Piston
      - (2) Plunger
      - (3) Differential
    - (2) Double rod
    - (3) Telescopic
- (2) Rotary type
  - (1) Single Vane
  - (2) Double Vane

Figure 2.4. Classification of Hydraulic Cylinders

The linear type cylinders can further be categorized into single acting and double acting cylinders. A single acting cylinder has only one port connected with system, and the cylinder or plunger is forced back by gravitational or spring force. Generally, it is used in an auxilliary situation. The double acting cylinder has two ports connected with the system and it actuates forward and backward by the action of hydraulic pressure.

Cylinders that have one piston rod extending from the cylinder bore are called the one-side rod (or single-rod) cylinders. Obviously, the two-side rod cylinders are cylinders have a piston rod extending from both sides of the cylinder body. Telescoping cylinders are actually multiple single-side rod cylinders.

There are two rotary type cylinders included in the CAAS program: the single vane and double vane types. The functions of single vane and double vane type devices are similar to those of single acting and double acting type cylinders except that the rotary type devices produce angular displacement instead of linear displacement.

Although the control functions and control mechanisms of cylinders are different from case to case, the fundamental principles used to model a cylinder are identical. The following paragraph illustrates the approach to formulate the performance equations of cylinders.

Figure 2.5 depicts the schematic diagram of a hydraulic cylinder. Assume that the movable assembly, the piston and rod, are motionless at the start. The upstream pressure increases whenever fluid flows into the upstream chamber of the cylinder. This is due to the effect of the compressibility property of the fluid. The drive force (upstream pressure times the effective area) is balanced by the external load imposed on the rod, and the total resistance force. The movable assembly begins to accelerate and moves toward the downstream direction since the drive force is greater than the external load. Simultaneously, at the opposite end of the piston, except for a single-acting cylinder, fluid is exhausted back to the reservoir through some other cylinder components.

In practice, it is necessary to provide hydraulic resistance in the return line. Therefore, a certain amount of pressure, called the "basic pressure" will be generated in the downstream chamber. These forces (back pressure times the downstream effective area) act to resist motion in the moving direction. This further results in an increase of the upstream chamber pressure. Normally, the upstream pressure should not exceed the nominal pressure setting.

Friction resistance is also included in the model. It is generated from the viscous damping effects and the mechanical friction which is always in the opposite direction to the velocity.

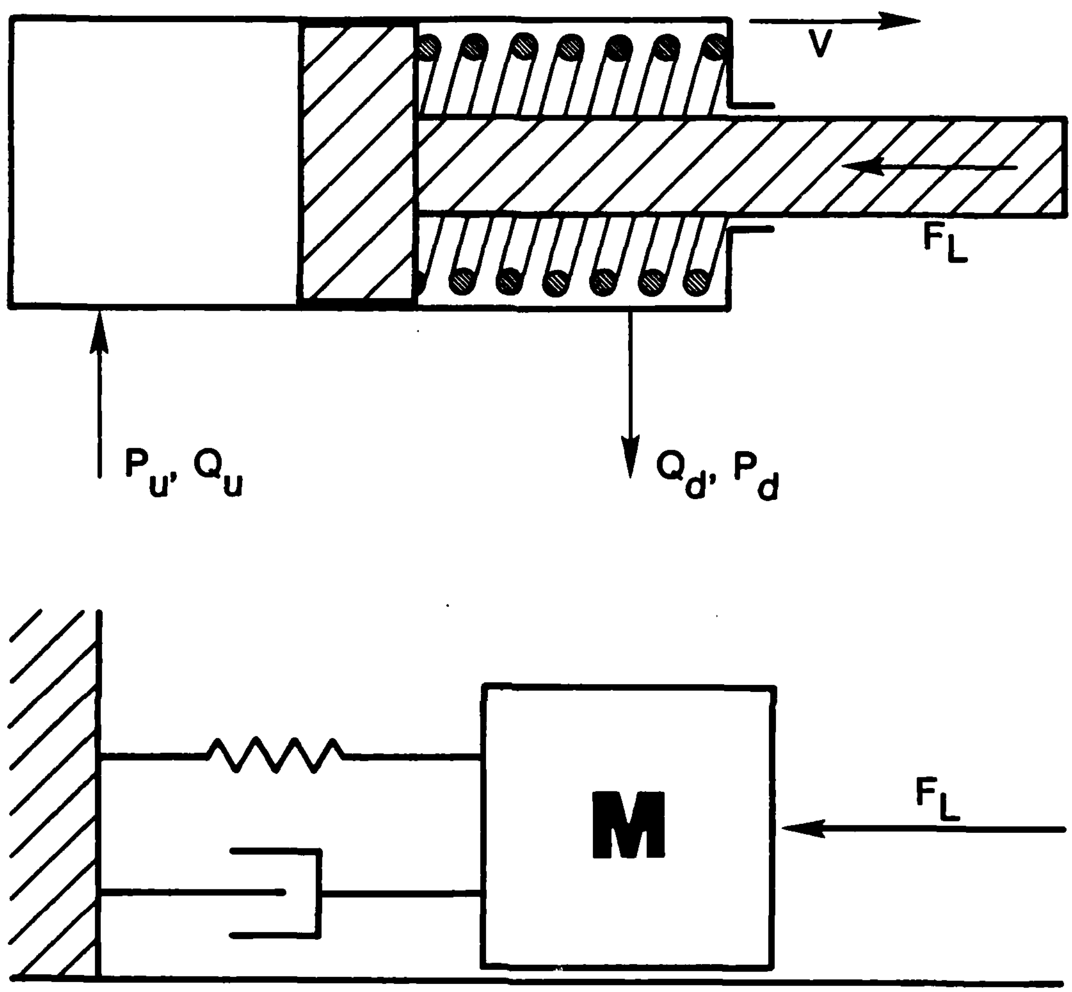


Fig. 2.5. Schematic Diagram of a Hydraulic Cylinder

The flow entering the cylinder depends on the velocity of the movable assembly. In addition, due to the pressure difference existing between the upstream and downstream chambers of a cylinder, the leakage flow effect is considered in the model.

Consequently, the mathematical model of a hydraulic cylinder is described as:

$$Q_u = A_u v + K_\ell (P_u - P_d) + \frac{V_u}{\beta} \frac{dP_u}{dt} \tag{2.14}$$

$$P_u A_u - P_d A_d = F_\ell + F_f + F_s + F_v + m \frac{dv}{dt} \tag{2.15}$$

$$\frac{dx}{dt} = v \tag{2.16}$$

and

$$F_s = F_{so} + K_s X \tag{2.17}$$

$$F_f = R_K + P_K (P_u - P_d)^{P_{KC}} \tag{2.18}$$

$$F_v = B_\ell v \tag{2.19}$$

$$B_\ell = \frac{\pi D \ell \mu}{h} \tag{2.20}$$

where	Au	upstream effective area
	Ad	downstream effective area
	Be	viscous damping coefficient
	D	diameter of piston
	Ff	seal friction
	Fs	spring force
	Fv	viscous damping force
	Fso	initial push force of spring
	Fe	load
	Ke	leakage coefficient
	Qu	flow rate entering cylinder
	Qd	flow rate draining from cylinder
	Pk	coefficient
	Pkc	exponent
	Rk	constant of seal friction
	Vu	volume of upstream chamber
	X	piston displacement
	h	radial clearance
	e	piston length
	m	mass of moving element
	Pu	upstream chamber pressure
	Pd	downstream chamber pressure
	β	fluid bulk modulus
	v	piston velocity

μ absolute viscosity

Eqs. (2.14) to (2.20) describes the dynamic performance of a cylinder. The static performance of a cylinder can be directly obtained by simply eliminating the differential terms.

CHAPTER III

AN OVERVIEW OF THE CAAS PROGRAM VERSION 2

GENERAL CONSIDERATION

The CAAS program is based on the development of a problem-Oriented Language (POL) which simplifies communications between the computer system and the system's users. This simplification of communication allows users with no prior programming experience to utilize the power of the computer for many applications, thus, increasing productivity. Obviously, such a package requires both a highly interactive procedure to communicate between the user and the computer and an effective numerical analysis procedure to manipulate system simulation. With this in mind, the function of the most commonly used high level languages were investigated at the first stage of Phase I of this program.

It was found that the PL/I language has its unique merit in manipulating character command which inherently provides an attractive procedure to meet the requirement of developing a user's oriented program. On the other hand, PL/I doesn't have as many mathematical functions available as FORTRAN. Therefore, in order to take the advantage of both languages, the PL/I and the FORTRAN, the CAAS program was designed to use PL/I in the POL and FORTRAN in the simulation sections. These two procedures were successfully linked together and proved very powerful in meeting the specification of the CAAS program. The PL/I-FORTRAN version CAAS program was termed "CAAS Program" Version 1.

The effectiveness of Version 1 has been recognized by implementing it on the computer that has both PL/I and FORTRAN supported facilities; for example, the IBM-370 system. Although PL/I is a very commonly used language, the portability and the maintainability of the program may somehow degrade if the package is implemented on different computers. Therefore, under a recommendation from the MERADCOM, the CAAS program was completely reconstructed and rewritten in the FORTRAN IV language during Phase II. The consideration of portability and maintainability is therefore reduced to the minimum. Furthermore, the updated FORTRAN version, the CAAS Version 2, not only retains all the character manipulation functions of PL/I, but also improves the entire program structure in a way that provides a more effective approach in data manipulation and system simulation.

Like version 1, version 2 uses one main (driver) routine and many sublevel routines. The communication with the user is governed by the main routine which calls the subprograms requested by the user. The calling program displays the requesting information and accepts the related input from the user. In all cases, the routine scans each response by the user for feasibility, and, if an incorrect response is detected, the user is told what error he has committed and is asked again for a response.

Another important aspect of the CAAS program is its "HELP" module. Anytime a user is asked for an input of some kind, he may enter the word "HELP" or an "H". Regardless of where he is in the program at the time, a tutorial explanation relevant to his requested input is presented in order to help him decide what is needed. This eliminates many unnecessary trips to a reference manual which may not contain the desired information.

THE PROGRAM STRUCTURE

The entirety of the CAAS program version 2 is subdivided into several logical units, Fig. 3.1:

1. Driver (MAIN)
2. Simulation (SIMULT)
3. Optimization (OPTIMZ)
4. Degradation (DEGRDN)
5. File Processing (GETCOM, GETINF)
6. Question/Answer (INPDAT)
7. Help (HELP01, HELP02)
8. Output Interpretation (OUTPPT)

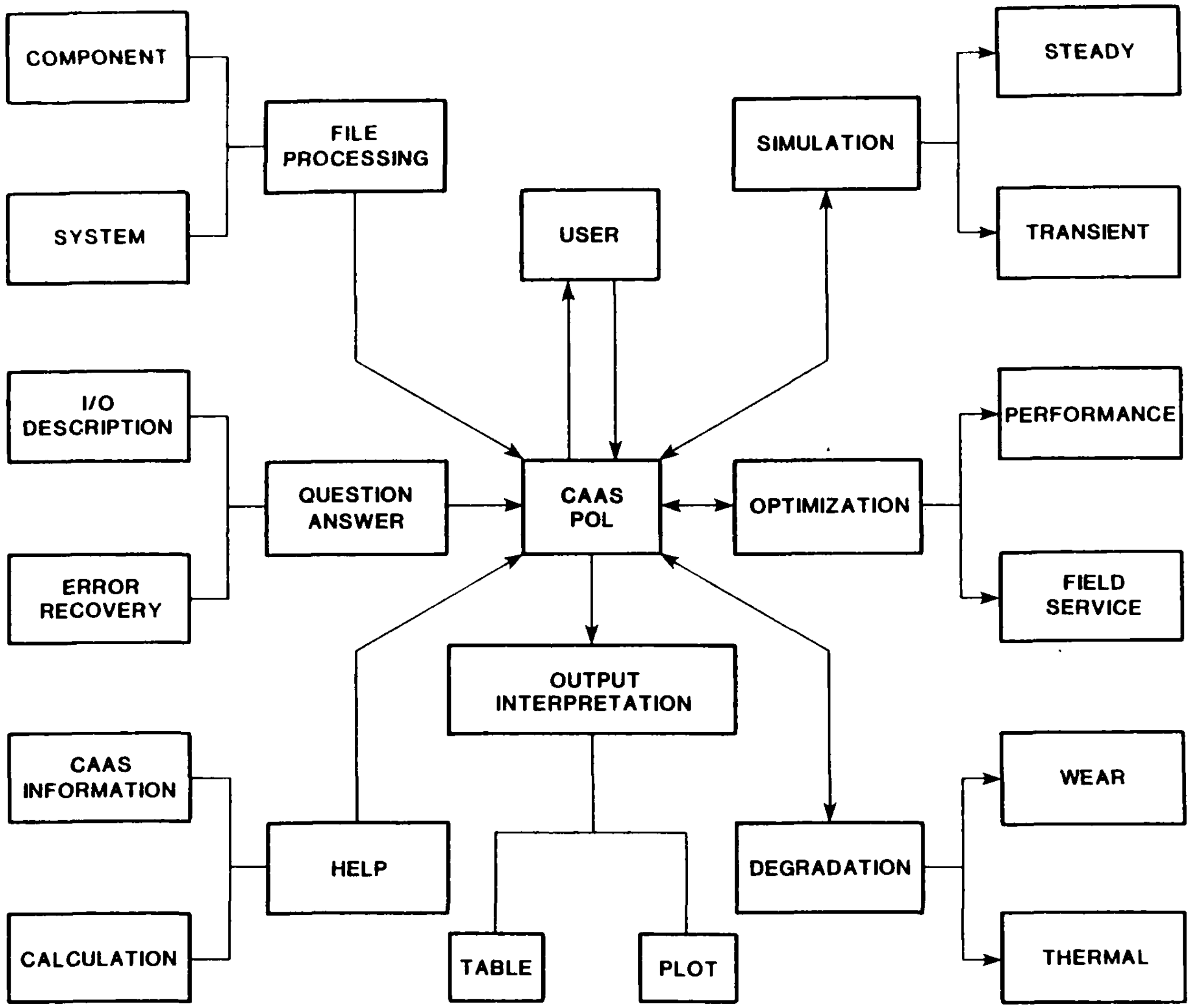


Fig. 3.1. Basic Structure of CAAS Program

The following is a description of all sections, the modules, and their interconnections.

Driver

This is the main drive modular of the CAAS program. It links all the logic units together to achieve the entire simulation. The main drive program along with the File Processing, Question/Answer, HELP, and Output Interpretation forms the Problem Oriented Language (POL) to communicate between the user and the computer.

Simulation

Essentially, this procedure is governed by a main drive subroutine which receives input data from GETINF through the POL and links sub-level subroutines together. This simulation procedure is used to perform the system simulation. Figure 3.2 depicts the basic structure of the SIMULT procedure. Basically, it consists of three major parts: component models, numerical analysis packages, and auxiliary engineering routines.

In the component models section, there is a component model bank which includes many preprogrammed simulation models of the most commonly used hydraulic components (in Phase I and II of this program, the models of hydraulic control valves, pumps, motors, and cylinders were generated). Each model subroutine is designed in

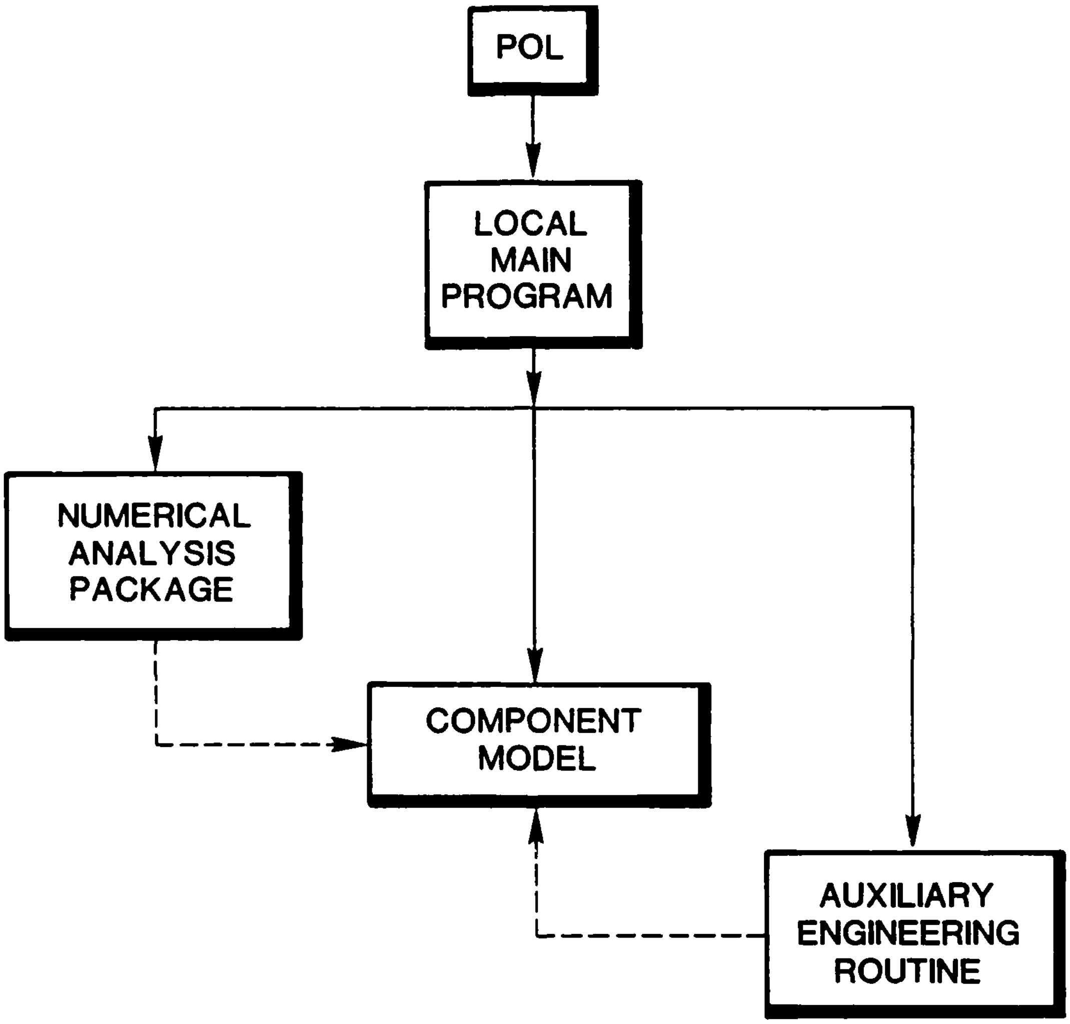


Fig. 3.2. Basic Structure of Simulation Program

such a way that the designer can select the emperical, static or dynamic model to meet the specific simulation problem. Flexibility in the model selection is one of the major advantages of the CAAS program. The emperical model uses the curve-fitting method with the polynomial interpolation technique to interpolate the performance variable by the specific input data. This technique is very useful if the information of the model can only be obtained experimentally. There is a procedure in "HELP" developed to fit the data in a polynominal form. The static and dynamic models are based on the mathematical model derived through the power flow method. They are used to perform the steady-state and the transient state simulations, respectively.

In addition to the mathematical model developed for each individual component, there is a design data request subprogram which delineates all the input information required for performing simulation of that specific component.

The numerical analysis package provides the necessary numerical differential equation analysis procedure and the non-linear equation solution package whenever the simulation program demands. The differential equation analysis procedures involve the Euler's method and the Runge-Kutta 4th order method. In addition to these two methods, the program is allowed to include any user supplied procedures. The non-linear equation solver is based on the algorithm of iteration searching and it is normally involved in the static model selection directly. The simulation algorithm will be extended to analyze an actual system performance in Phase III of this project.

The auxiliary engineering routines are developed for special engineering functions which may be repeatedly used in the simulation program, for example, the subprogram DAMP is used to calculate the damping ratio.

Optimization

OPTIMZ provides the required procedures to achieve system parametric optimization. Several options of optimization criteria are preset in the CAAS. This procedure will be developed in Phase IV of this project.

Degradation

DEGRDN deals with the system performance degradation due to the variation of fluid temperature and/or contamination level. This procedure will be developed in Phase IV of this project.

File Processing

The File Processing section consists of GETCOM, GETINF, CIRCKT, NATSIM, FLUPRO, and RERUN, Fig. 3.3. It receives the required information from user to achieve the desired simulation. This

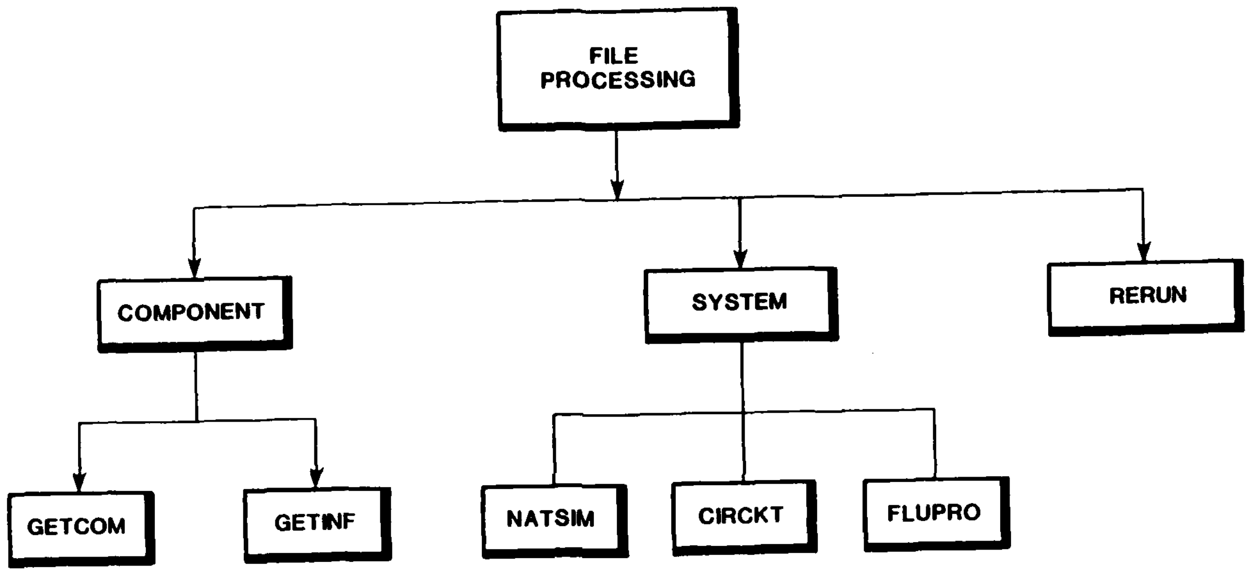


Fig. 3.3. Basic Structure of File Processing Programs

procedure makes the user's oriented design possible.

GETCOM interacts with the user to choose a single component from the CAAS component ID bank which is stored in an on-line data file. There are two ways for the user to select the component. First, if the user already knows the component ID number that he needs, then he is allowed to enter it directly, and the program checks the validity of the entered number. If the chosen component is indicated to be correct, control is passed back to the selection manual for next selection; otherwise, the user is requested to enter a proper ID number. Second, if the user does not know the component ID number when asked, he is given the master component menu and asked to choose one. Each component type has a menu which subdivides into several levels so that the user can specify every detail of any component with very little effort.

GETINF carries on a dialog with the user for each component selected. The dialog is specified by the component database sub-programs. The name of the database subprograms are usually in a form of "DBXXXX" for component XXXX. In addition, to get the data of each specific component, GETINF also has a function to allow the user to change data if such a change is necessary.

CIRCKT guides the user to construct the hydraulic control circuit in a way which is acceptable to CAAS. Because the CAAS program is based on the algorithm of the power-flow method, before a series of components can be of any use in a simulation, they must be connected together in some fashion which is compatible with the power-flow algorithm. CIRCKT gets the component port number and connecting line numbers labeled by the user, thus forming the power-flow representation of the hydraulic control circuit. With the aid of CIRCKT, the computer "knows" the sequence of simulation and the process of manipulating data.

NATSIM receives system simulation data from the user. The input data required are the nature of system simulation (static or dynamic), simulation time parameters, and the integration method used to perform dynamic simulation.

FLPROP obtains the values of working fluid properties. It includes the values of fluid bulk modulus, density, and viscosity. A table of default values for five kinds of commonly used fluids are preset. This allows the user to use the default values for a specific fluid directly or he can change any of the default values if he needs to do so.

RERUN gives the user the opportunity to alter any of the parameters that he has already specified before he conducts the simulation or reruns the simulation with only minor modifications of the previously entered parameters.

Question/Answer

The Question/Answer section provides an efficient means of I/O data manipulation. It also offers the ability of error recovery during the program processing. Basically this section is governed by the subprogram INPDAT along with sublevel programs that are required to support it.

INPDAT is a general purpose routing which is used any time user input is needed. It recognizes user input errors when a wrong type or class of input string is given. The subprograms get an entire record from the user's terminal and break it down according to the specifications requested by the calling program.

INPDAT detects the following user input errors and reprompts the user for a correct response, for instance:

1. An INTEGER number entered when a REAL number is needed.
2. A REAL number entered when an INTEGER number is needed.
3. An alphanumeric string entered when a numerical value is needed.
4. A numerical value entered when an alphabetic string is needed.

5. A number or character string when a "yes" or "no" answer is needed.

HELP

The HELP module is fully accessible to the user while INPDAT (see previous section) is executing and aids the user when he is unsure of what is required in the way of a response. The CAAS program provides three different on-line help facilities to assist the user in successfully performing the simulation he desired, Fig. 3.4.

HELP 01 provides the first level help of written clarification. It calls from the data file short one-or two-line sentences to help the user decide what the system needs from him.

HELP 02 is called when the user needs more detailed information in addition to the message displayed in HELP 01. HELP 02 is a series of full-page explanations that detail exactly what the system is asking for. There are many commands available for the user to "turn" pages of the on-line user's manual to find any information he needs.

The third on-line aid is a series of subsystems that do calculations for the user, for example calculating the integration step size, the viscous damping ratio, etc.

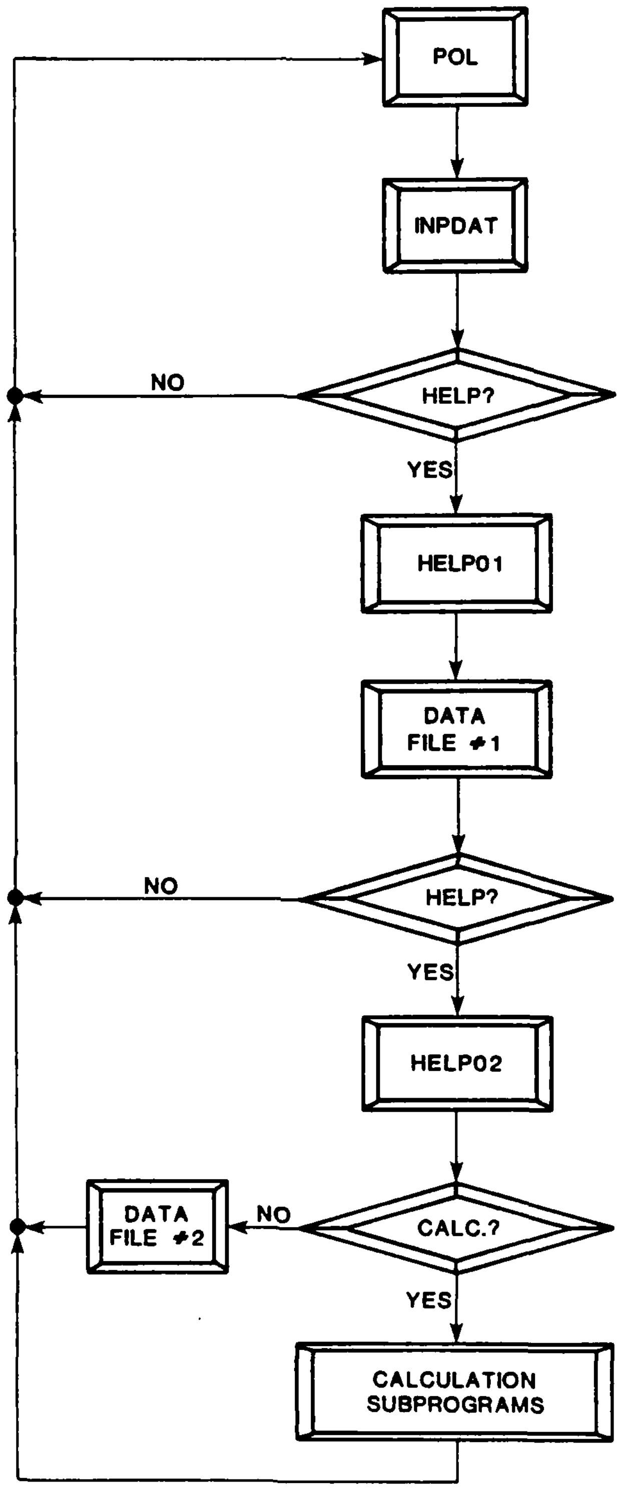


Fig. 3.4. Basic Structure of HELP Module

Output Interpretation

OUTPPT allows the user to select the simulation results of any effort/flow pair he desires to be displayed on either a tabular format (TABLE) or a graphic representation (PLOT). Furthermore, subprogram TEKPLT generates a two-dimensional plot on a TEKTRONIX 4000-series graphics terminal.

CHAPTER IV

PRESENTATION AND EVALUATION OF EXPERIMENTAL RESULTS

The purpose of this chapter is to present the experimental work carried out at the FPRC to validate the models developed for hydraulic control valves during Phase I. There were 9 valves selected for this purpose:

1. Directional Control Valve: 2-way, normally open type.
2. Directional Control Valve: 3-way, 2-position, normally open type.
3. Directional Control Valve: 4-way, 3-position, normally closed type.
4. Flow Control Valve: Restrictive, pressure compensated type.
5. Flow Control Valve: By-pass, 2-way, pressure compensated type.
6. Flow Control Valve: By-pass, 3-way, pressure compensated type.
7. Pressure Control Valve: Direct acting relief valve.
8. Pressure Control Valve: Pilot operated relief valve.
9. Pressure Control Valve: Pilot operated reducing valve.

The presentation of test results is constructed with a simple format as shown below:

Test Component - Identifies the component tested. It includes the description of component function, component I.D. number, and component schematic diagram.

Experimental Verification - Illustrates the layout of test system and describes all necessary procedures to conduct the test.

Computer-Aided Simulation - Uses the CAAS package to simulate the test system by employing the actual measured data.

Results Representation - Presents and discusses both the simulation results and the actual test results.

In addition to the above presentations, the following test conditions were followed throughout the tests.

1. Temperature of working fluid: $100^{o}F$
2. Working fluid: MIL-H-5606
3. Measurement accuracy: Flow $\pm$ 2%

   Pressure $\pm$ 2%

   Temperature $\pm$ $5^{o}F$

TEST I

A. Test Component

1. Name: 2-way, normally open, solenoid actuated directional control valve.

2. I. D. No.: 121131

3. Schematic Diagram: See Figure 4.1(a).

B. Experimental Verification

1. Set-up: Figure 4.1(b) illustrates the test system used for the dynamic and static performance test of a 2-way, normally open direction control valve. The circuit includes:

   - a fixed displacement pump which delivers flow at 20 in^3/sec.
   - a relief valve to accomplish a constant supply pressure for the test valve.
   - an accumulator and an orifice to filter out the excessive hydraulic pulsations.
   - a D.C. signal to control the open/close function of the test valve.

2. Test Procedure

   <u>Static Test</u>

   - Install the test valve and achieve test temperature.
   - Adjust system pressure relief valve to vary the inlet (upstream) pressure level.

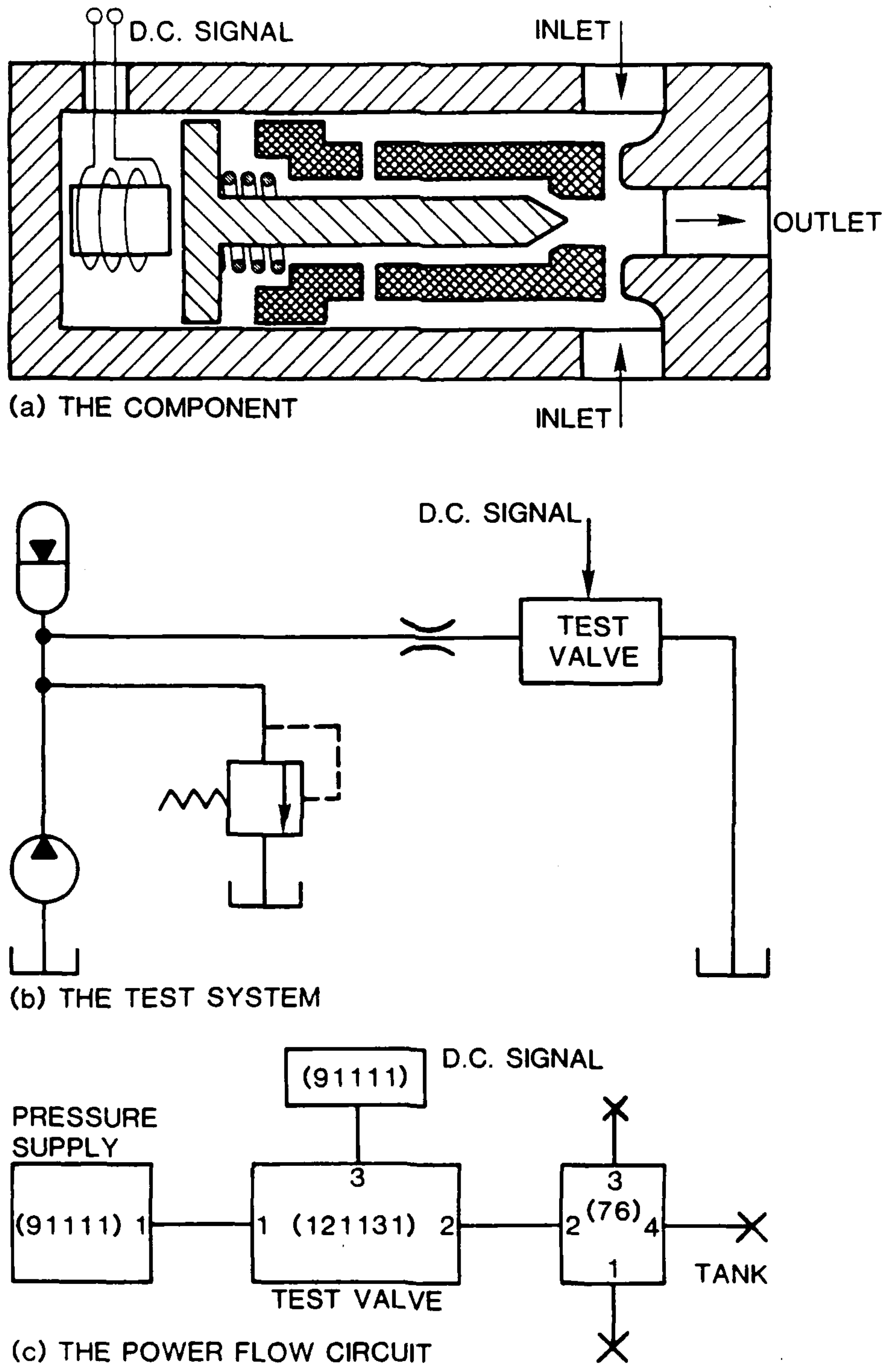


Fig. 4.1. The Test System of 2-way Directional Control Valve

- Measure and record the outlet (downstream) flow rate in terms of the pressure difference across the inlet and outlet.

Dynamic Test

- Install the test valve and achieve test temperature.
- Adjust the inlet pressure to produce an outlet flow of 20 in^3/sec.
- Close the test valve.
- Measure and record the response of the outlet flow.

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.1(c) component 1 (91111) provides a constant pressure to component 3 (the test valve). Component 2 (91111) provides the required force to close the valve at the specific time.
2. Input Data:

   Orifice area: 0.009 in^2

   Discharge coefficient: 0.61

   Area gradient of discharge orifice: 0.09 in^2/in

   Overlap (+) or Underlaps (-): 0.0 in.

   Flow jet angle: 90 degrees.

   Spring constant: 40.0 lbf/in.

   Spool clearance: 0.00001 in.

Mass of spool: 0.00012 $lbf\text{-}sec^2/in.$

Viscous damping length: 0.5 in.

Unsteady flow force coefficient: 0

Spool diameter: 0.5 in.

Initial Conditions:

Spool displacement: 0. in.

Spool velocity: 0. in/sec.

D. Results Presentation

1. Static performance: See Fig. 4.2.
2. Dynamic performance: See Fig. 4.3.

TEST 2

A. Test Component

1. Name: 3-way, 2-position, normally open solenoid, actuated directional control valve.
2. I. D. No: 131131
3. Schematic Diagram: See Fig. 4.4(a).

B. Experimental Verification

(Same as the Experimental Verification section of Test I).

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.4(c). Component 1 (91111) provides a constant pressure to component 3 (the test valve).

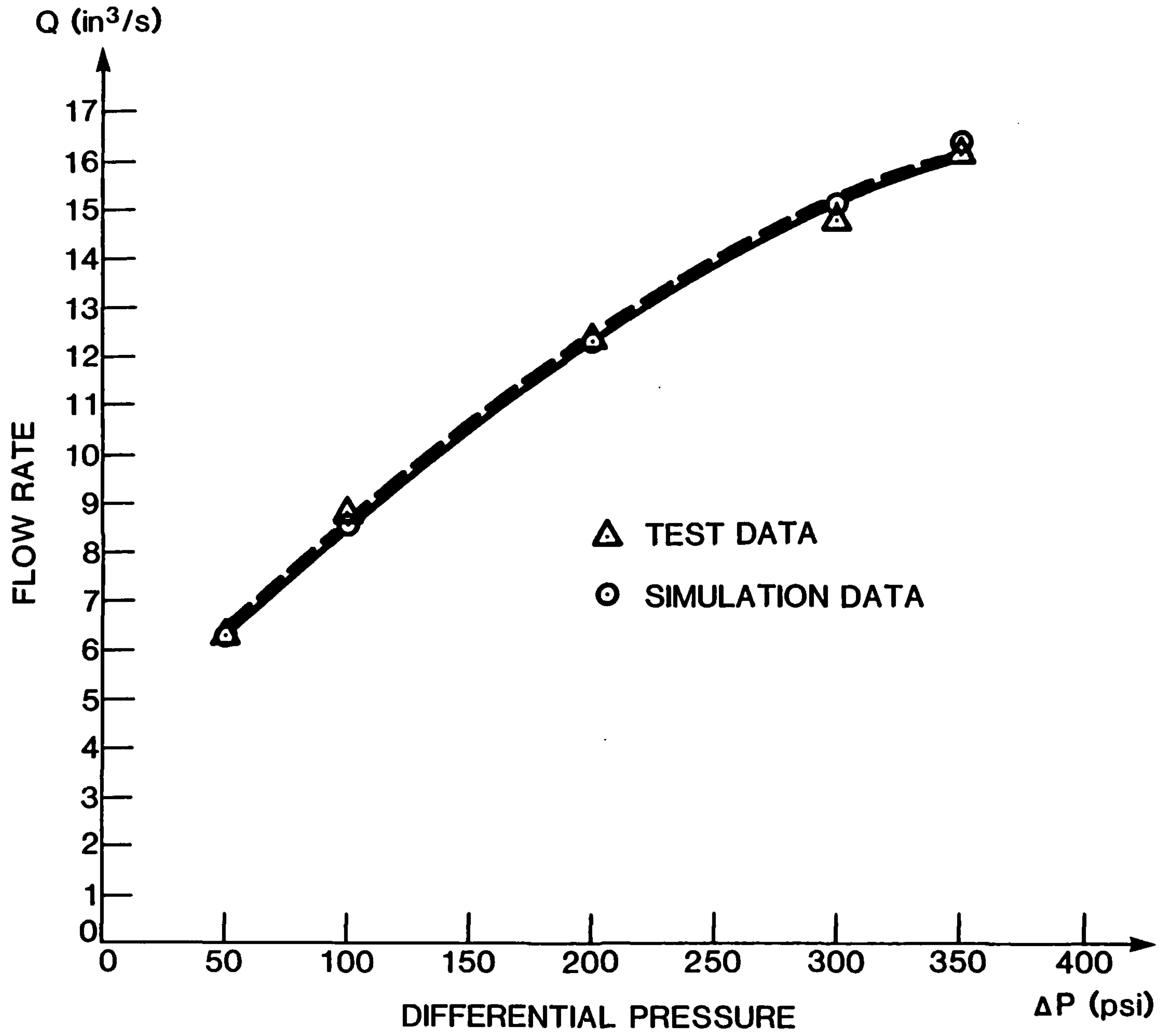


Fig. 4.2. Static Characteristic of 2-way Directional Control Valve

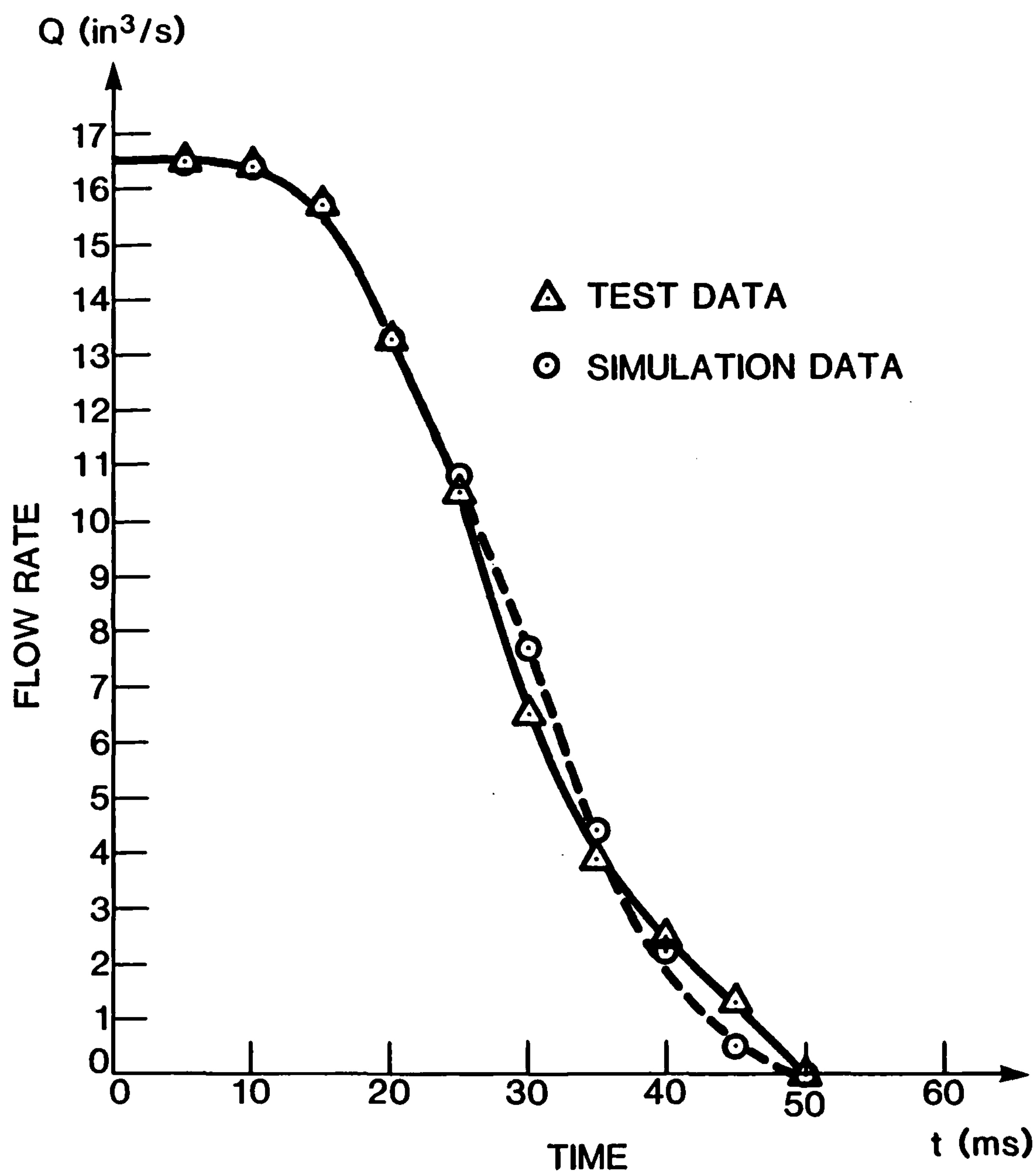


Fig. 4.3. Dynamic Characteristic of 2-way Directional Control Valve

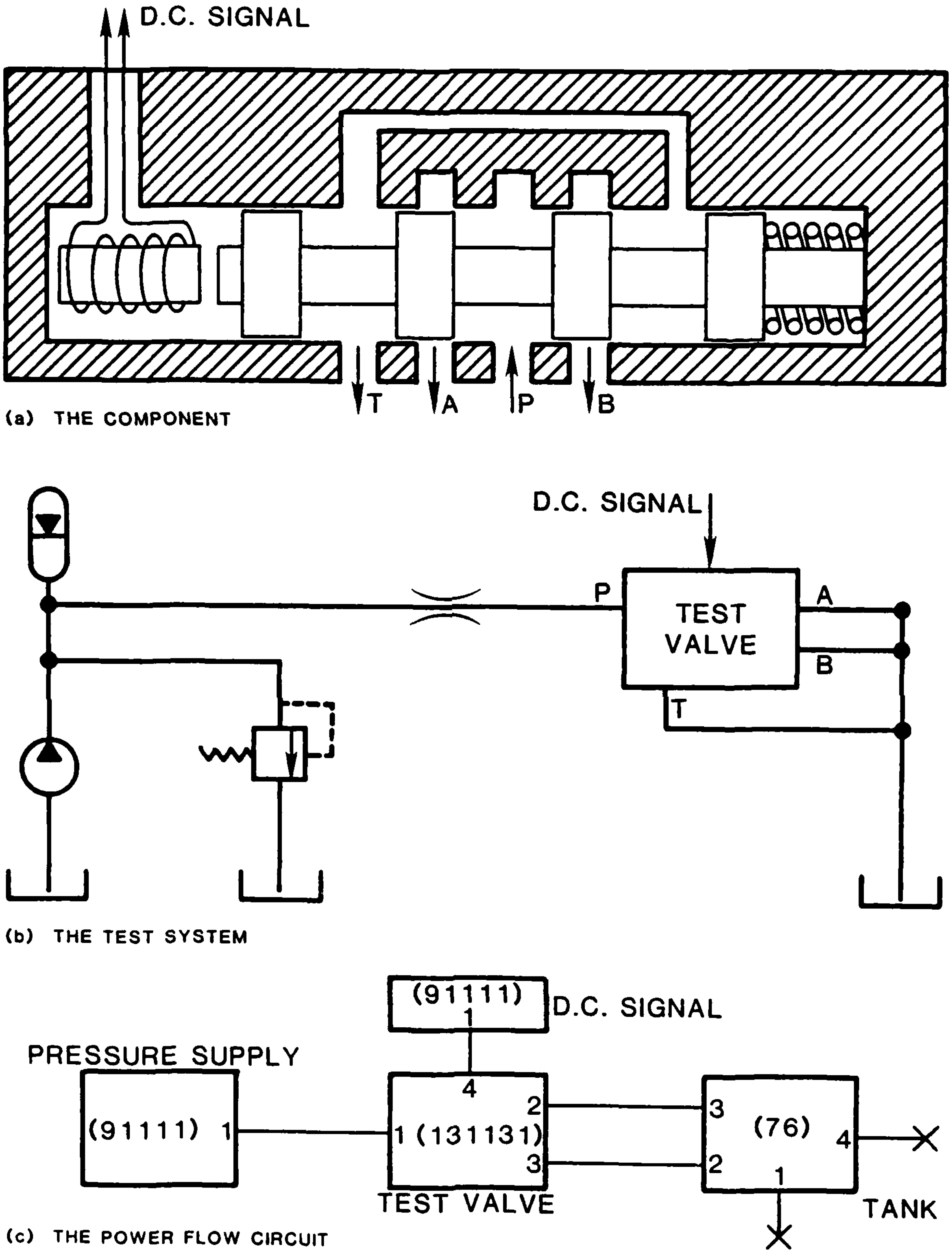


Fig. 4.4. The Test System of 3-way Directional Control Valve

Component 2 (91111) provides the required force to close the valve at the specific time.

2. Input Data: Orifice area: 0.0255 in^2

   Discharge coefficient: 0.6

   Area gradient at discharge orifice: 0.425 $in^2/in.$

   Overlap (+) or Underlap (-): 0.0 in.

   Spring constant: 20 lbf/in.

   Flow jet angle: 90 degrees

   Leakage flow coefficient: 0

   Spool clearance: 0.0000/25 in.

   Mass of spool: 0.0034 $lbf\text{-}sec^2/in.$

   Viscous damping length: 1 in.

   Unsteady flow force coefficient: 0

   Spool diameter: 0.625 in.

   Initial conditions:

   Spool displacement: 0 in.

   Spool velocity: 0 in/sec.

D. Results Presentation

1. Static performance: See Fig. 4.5.
2. Dynamic performance: See Fig. 4.6.

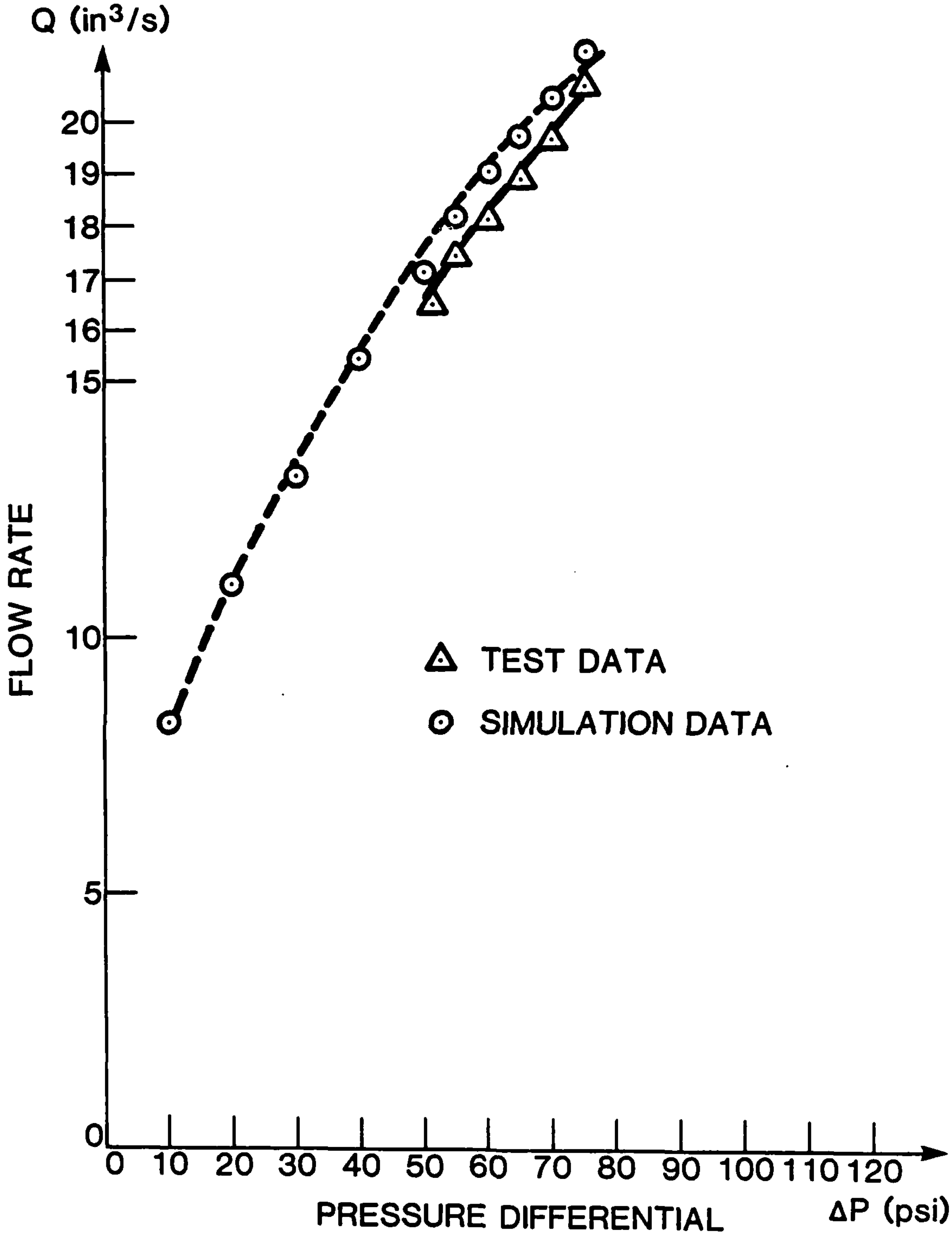


Fig. 4.5. The Static Characteristic of 3-way Directional Control Valve

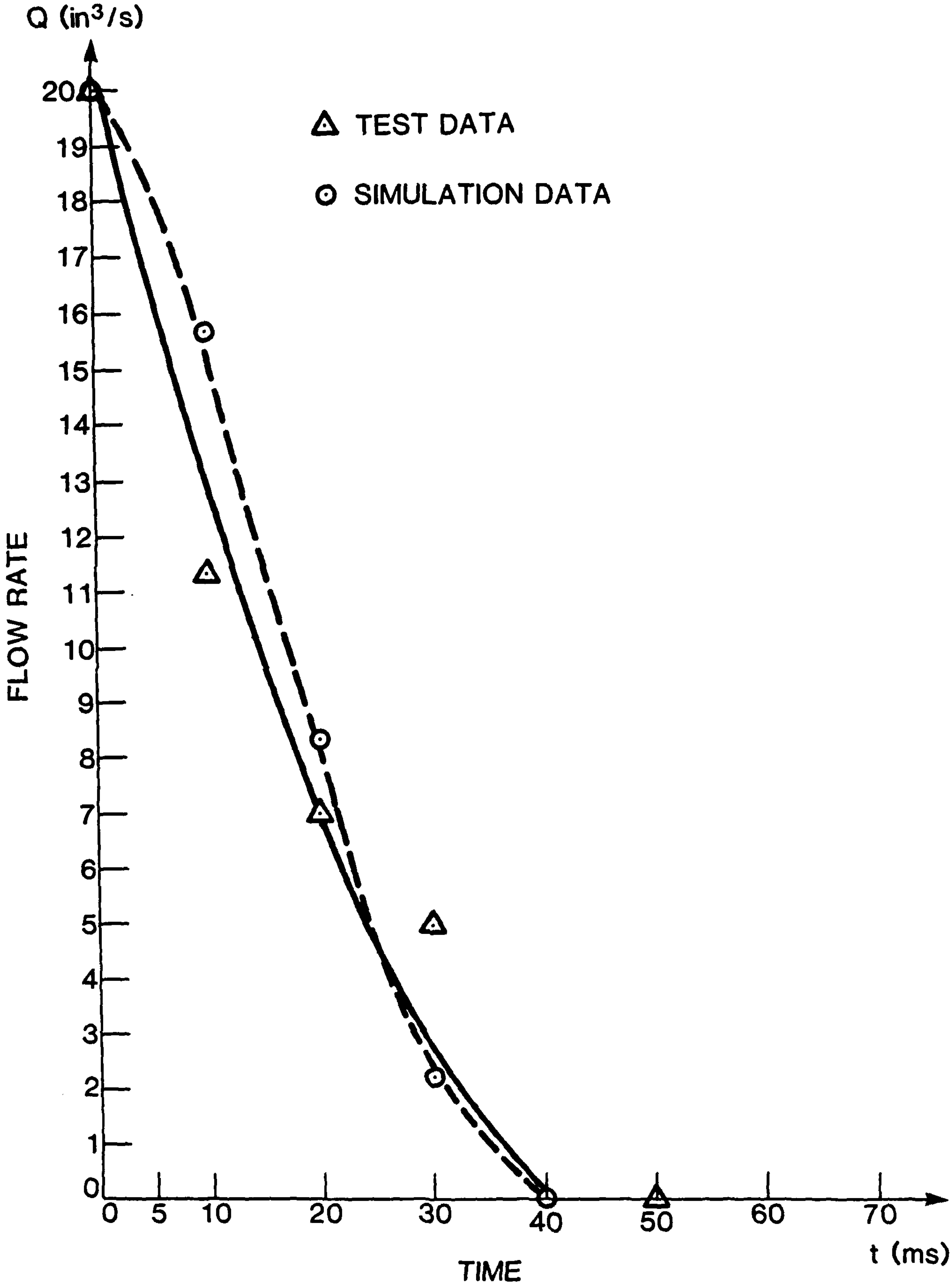


Fig. 4.6. The Dynamic Characteristic of 3-way Directional Control Valve

TEST 3

A. Test Component

1. Name: 4-way, 3-position, normally closed, solenoid actuated, directional control valve.

2. I. D. No: 142231

3. Schematic Diagram: See Fig. 4.7(a).

B. Experimental Verification

(Same as the Experimental Verification Section of Test I except change 'close the test valve' to 'open the test valve' in Step 3 of the Dynamic Test, and open the valve during the Static Test).

C. Computer-Aided Simulation

1. Power-flow circuit: See Fig. 4.7(c). Component: 1 (91111) provides a constant pressure to component 3 (the test valve). Component 2 (91111) provides the required force to open the valve at the specific time.

2. Input Data: Orifice area: 0.0255 in^2

   Discharge Coefficient: 0.6

   Area gradient at discharge orifice: 0.425 in^2/in

   Overlap (+) or Underlap (-): 0.0 in.

   Spring constant: 20 lbf/in.

   Flow jet angle: 90 degrees.

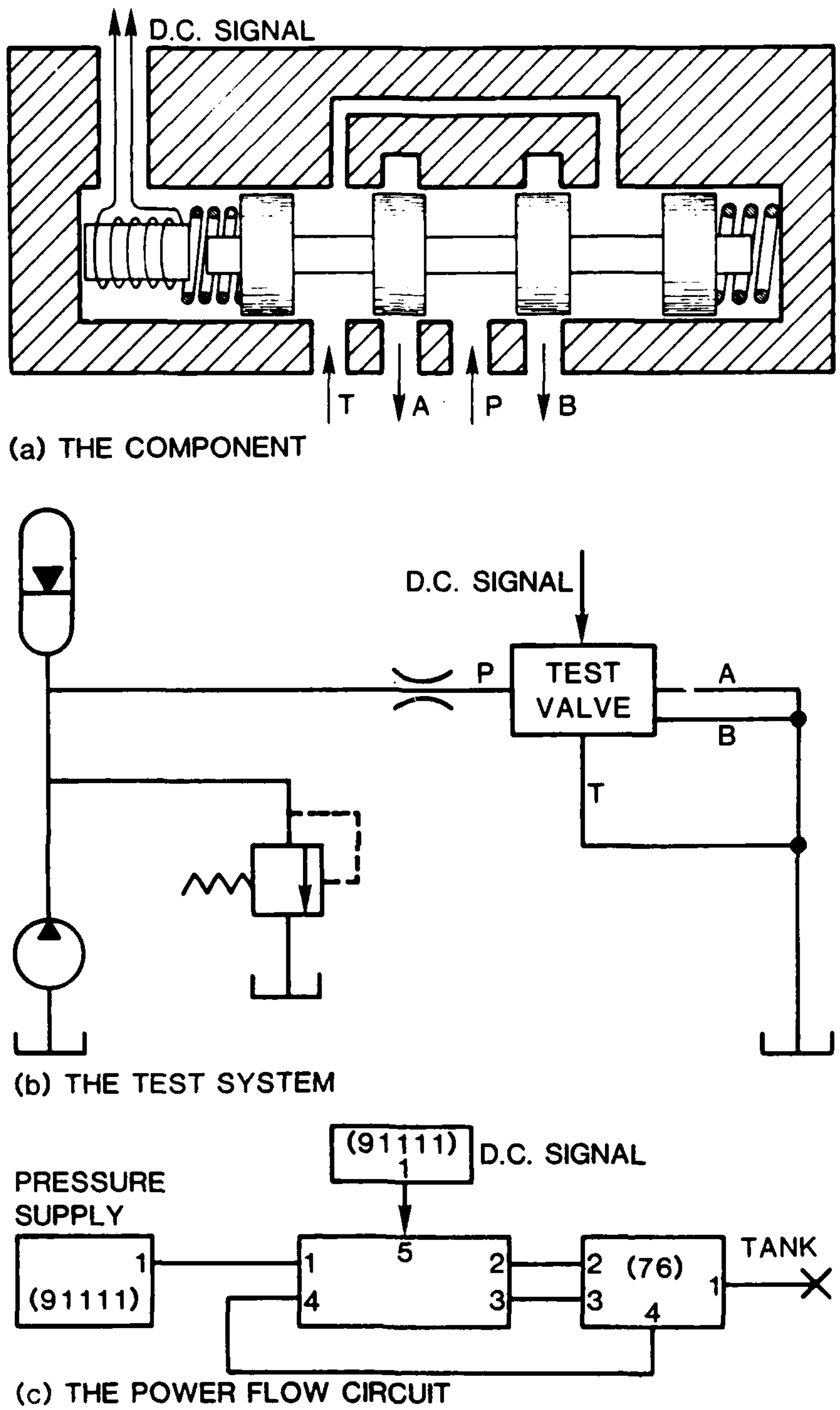


Fig. 4.7. The Test System of 4-way Directional Control Valve

Leakage flow coefficient: 0

Spool clearance: 0.00005 in.

Mass of spool: 0.0034 lbf-sec^2/in

Viscous damping length: 1 in.

Unsteady flow force coefficient: 0

Spool diameter: 0.625 in.

Initial conditions:

Spool displacement: 0 in.

Spool velocity: 0 in/sec.

D. Results Presentation

1. Static performance: See Fig. 4.8.
2. Dynamic performance: See Fig. 4.9.

TEST 4

A. Test Component

1. Name: Restrictive type, pressure compensated flow control valve.
2. I. D. No.: 223
3. Schematic Diagram: See Fig. 4.10(a).

B. Experimental Verification

1. Set-up: Figure 4.10(b) illustrates the test rig used for dynamic and static response of restrictive-type pressure

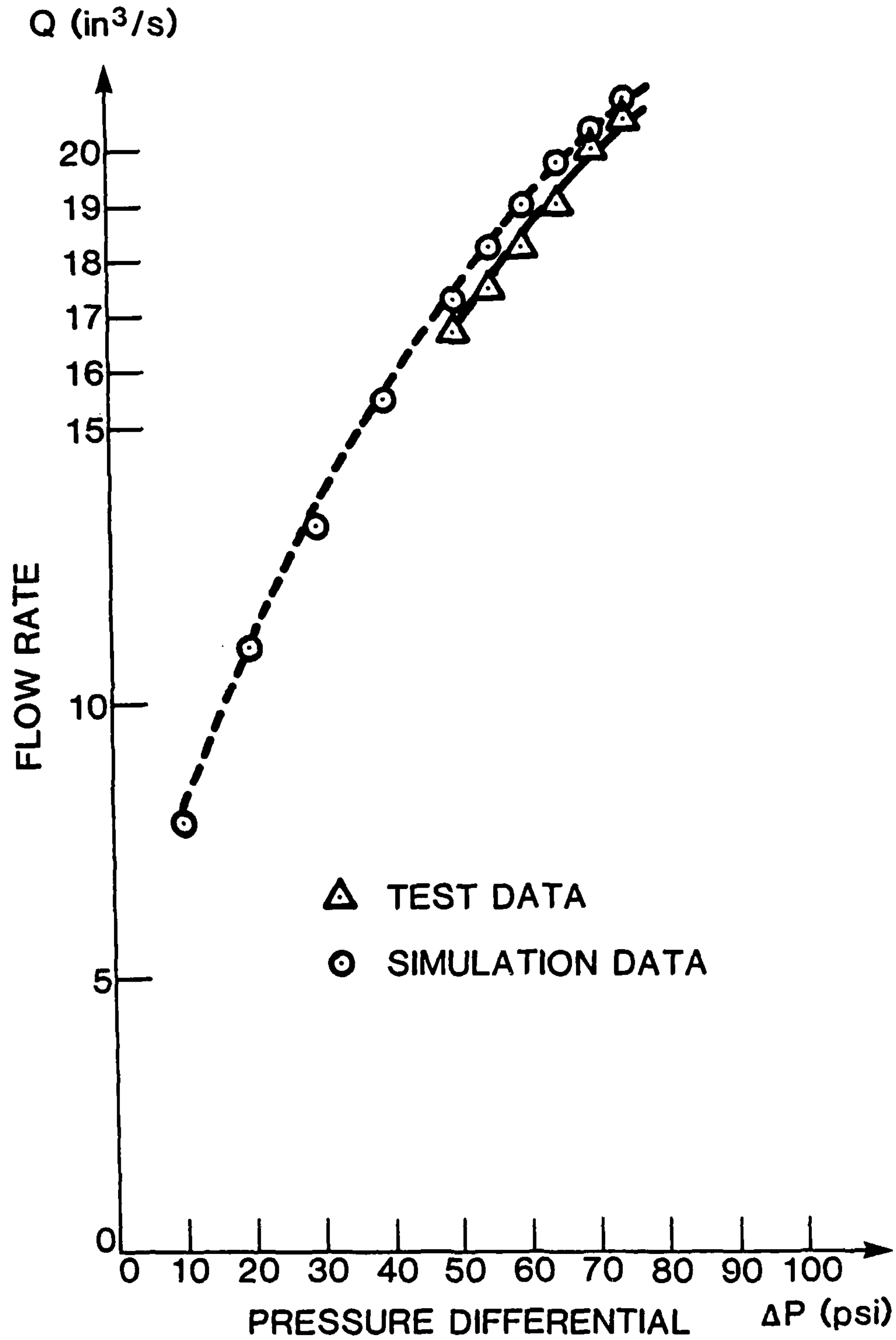


Fig. 4.8. The Static Characteristic of 4-way Directional Control Valve

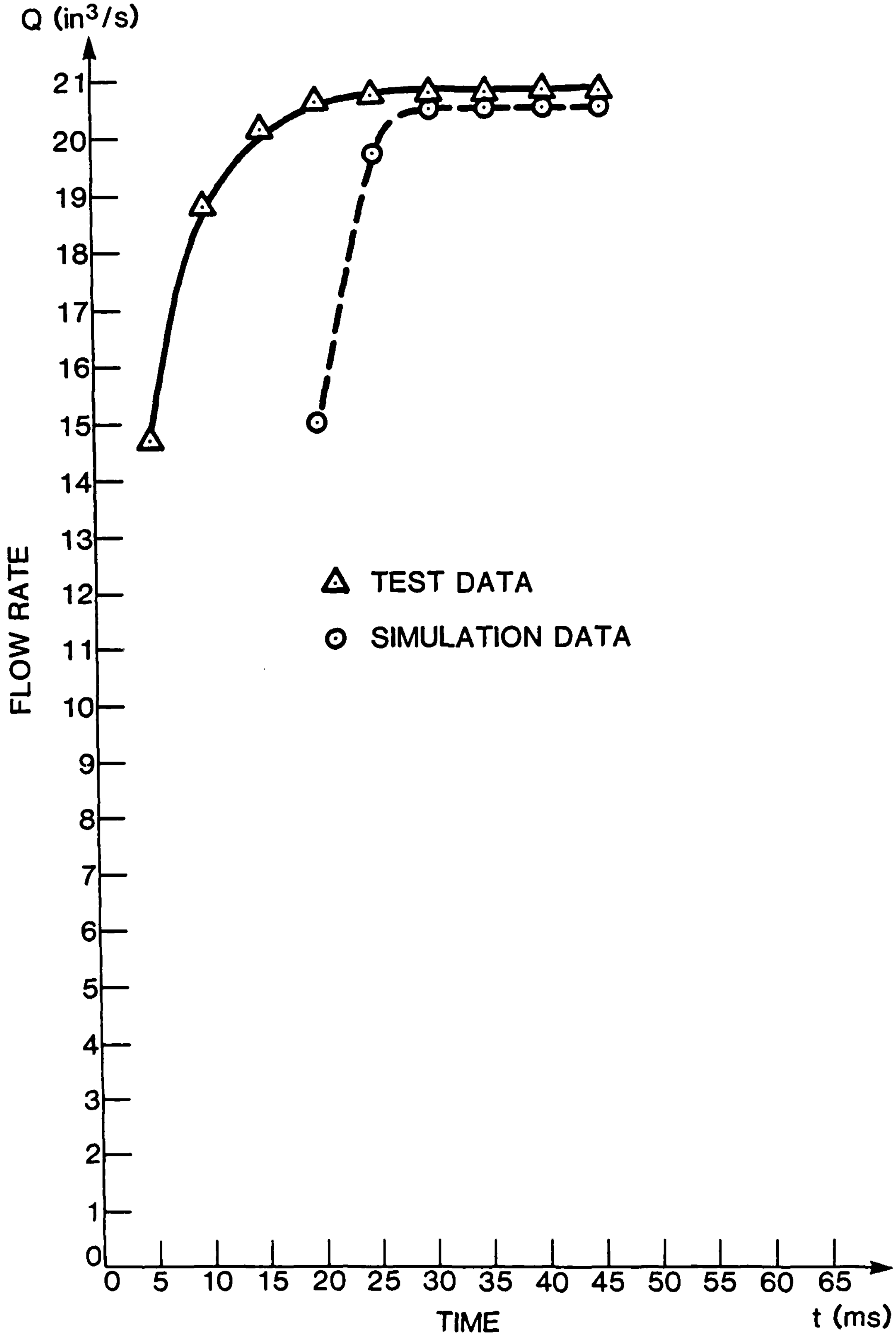


Fig. 4.9. The Dynamic Characteristic of 4-way Directional Control Valve

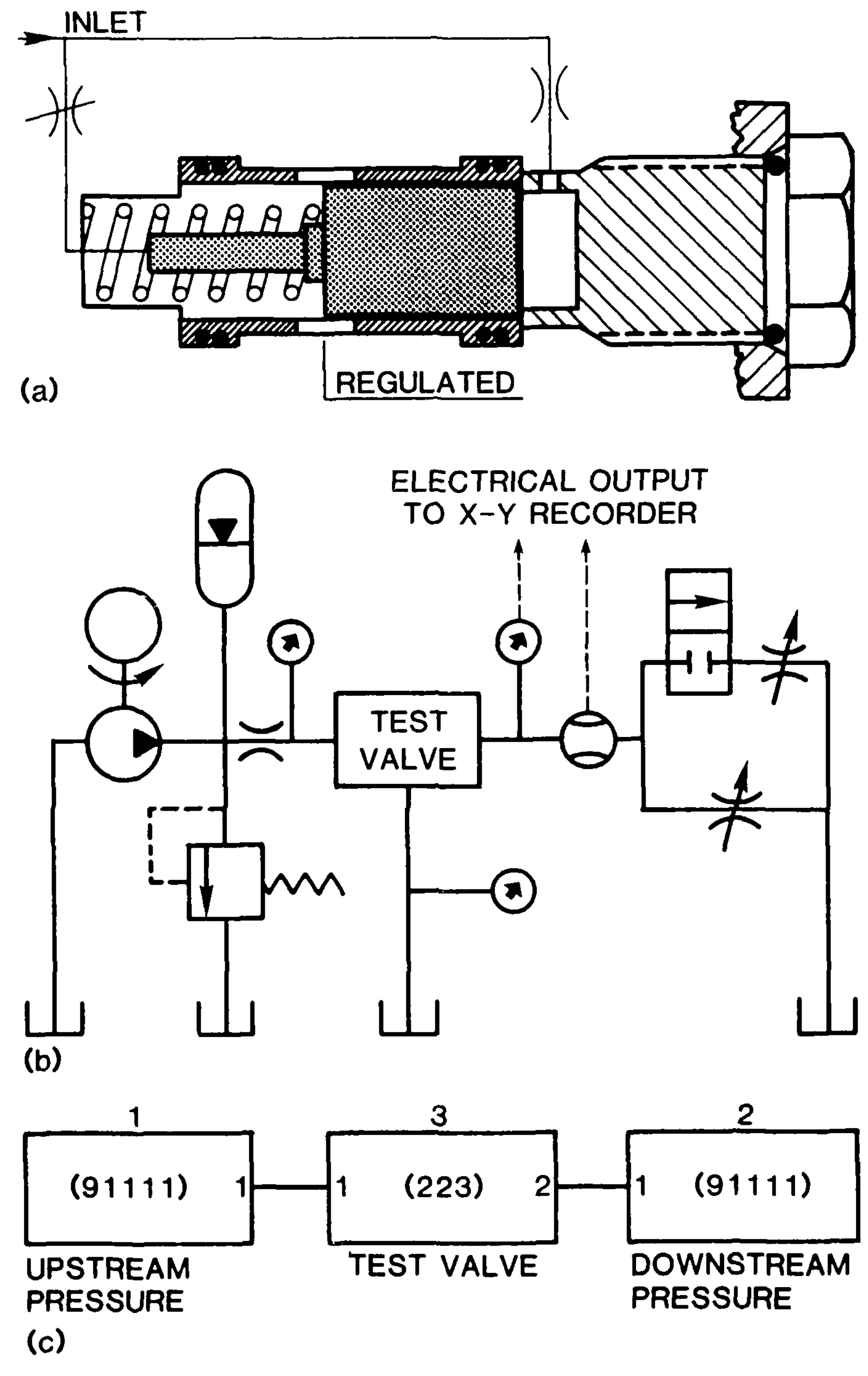


Fig. 4.10. The Test System of Restrictive Type Flow Control Valve

compensated flow control valves. The hydraulic circuit composed of:

- a fixed displacement pump which delivers flow at 40 in^3/sec
- a relief valve to accomplish a constant supply pressure for the test valve.
- an accumulator and an orifice to filter out the excessive hydraulic pulsations.
- a solenoid actuated 2-position, 2-way directional control valve to initiate a step input to the test valve.
- two needle valves, one for the first steady state condition and the other for the second.

2. Test Procedure

Static Test

1. Install the test valve and achieve test temperature.
2. Adjust test system relief valves to vary the upstream pressure level.
3. Measure and record the outlet flow rate in terms of the pressure difference across the inlet and outlet.

Dynamic Test

1. Install the test valve and achieve test temperature.
2. Adjust test system relief valve to the test system pressure desired.

3. Set a test flow rate and load pressure by adjusting the test valve and needle valve 1 with needle valve 2 closed.
4. Close the solenoid valve.
5. Apply a step input of load pressure by opening the solenoid valve.
6. Record the controlled (outlet) flow and load pressure as a function of time until the steady state is achieved.

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.10(c). Component 1 (91111) provides constant supply pressure to the test valve (component 3). Component 2 (91211) loads the test valve with a ramp pressure signal. This circuit diagram was used to simulate both static and dynamic performance of the test valves.

2. Input Data: Opening area of adjustable orifice: 0.021 in^2
Compensation spool reaction area: 0.373 in^2
Damping orifice area: 0.0024 in^2
Spring constant: 42.1 lbf/in.
Preload displacement of spring: 0.537 in.
Flow discharge coefficient of compensator orifice: 0.6

Flow discharge coefficient of adjustable orifice: 0.6

Damping orifice discharge coefficient: 0.6

Area gradient of compensator orifice: 0.288 in^2/in.

Flow jet angle of compensator orifice: 69 degrees

Minimum compensator spool displacement: 0.081 in

Maximum compensator spool displacement: 0.185 in.

Viscous damping coefficient: 0.1 lbf-sec/in

Mass of compensator spool: 0.000078 $lbf\text{-}sec^2/in$.

D. Results Presentation

1. Static performance: See Fig. 4.11.
2. Dynamic performance: See Fig. 4.12.

TEST 5

A. Test Component

1. Name: Bypass type, 2-way pressure compensated flow control valve.
2. I. D. No.: 2221
3. Schematic diagram: See Fig. 4.13(a).

B. Experimental Verification

1. Set-up: (same as the set-up of Test 4).
2. Test Procedure: (same as the Test procedure of Test 4 except having one more step used to record the pressure of by-pass

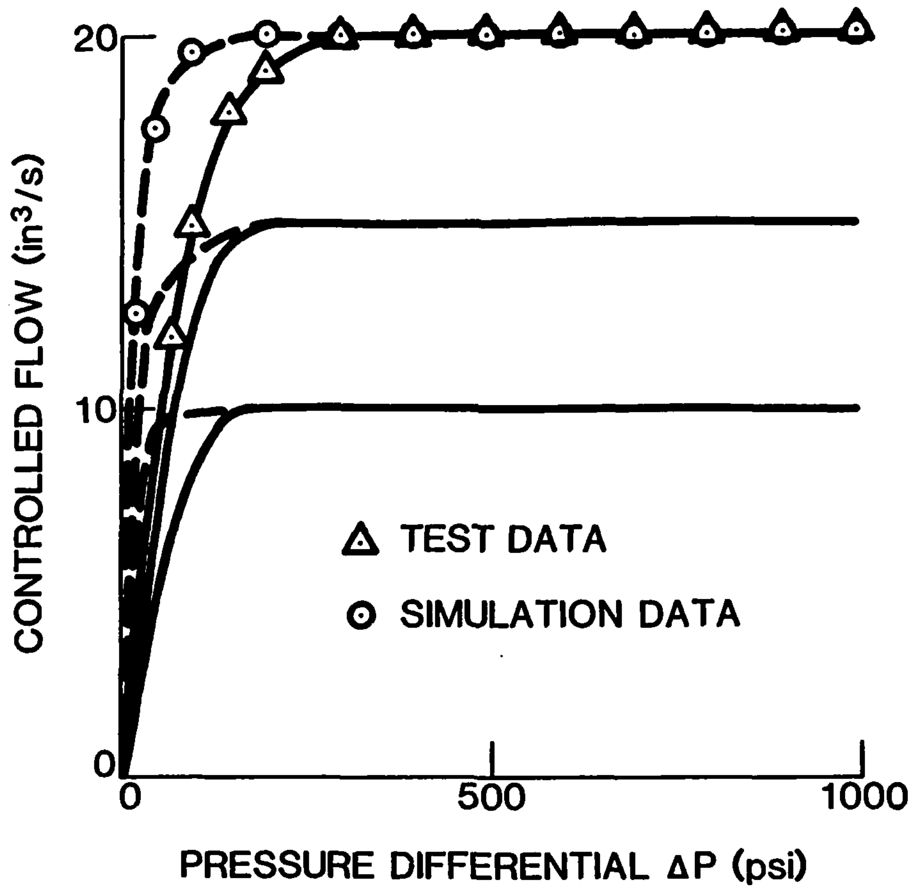


Fig. 4.11. The Static Characteristic of Restrictive Type Flow Control Valve

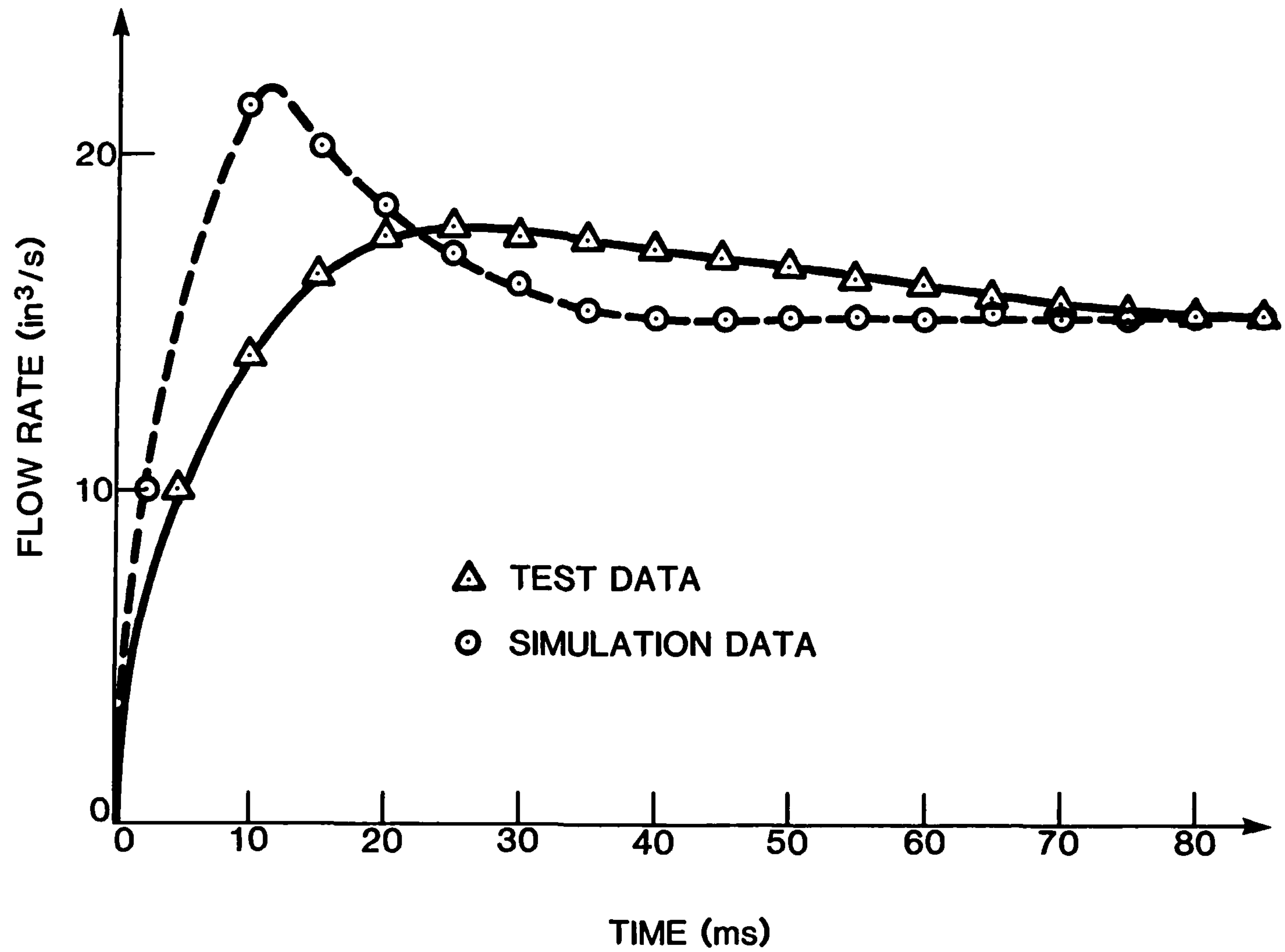


Fig. 4.12. The Dynamic Characteristic of Restrictive Type Flow Control Valve

at steady state condition in the Dynamic Test Section).

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.13(c). The configuration of the circuit is similar to that of Test 4. In this test, one more component (91111) is required to supply a constant by-pass pressure.

2. Input Data: System relief pressure: 1541 PSI

   Opening area of adjustable orifice: 0.01 in^2

   Compensator spool reaction area: 0.373 in^2

   Damping orifice area: 0.00071 in^2

   Spring constant: 112 lbf/in

   Preload displacement of spring: 0.193 in.

   Flow discharge coefficient of adjustable orifice: 0.6

   Flow discharge coefficient of compensator orifice: 0.6

   Flow discharge coefficient of damping orifice: 0.6

   Area gradient of compensator orifice: 0.144 in^2/in

   Flow jet angle of compensator orifice: 69 degrees

   Minimum compensator spool displacement: 0.039 in

   Maximum compensator spool displacement: 0.264 in.

   Mass of compensator spool: 0.000357 lbf-sec^2/in,

   Viscous damping coefficient: 0.1 lbf-sec/in.

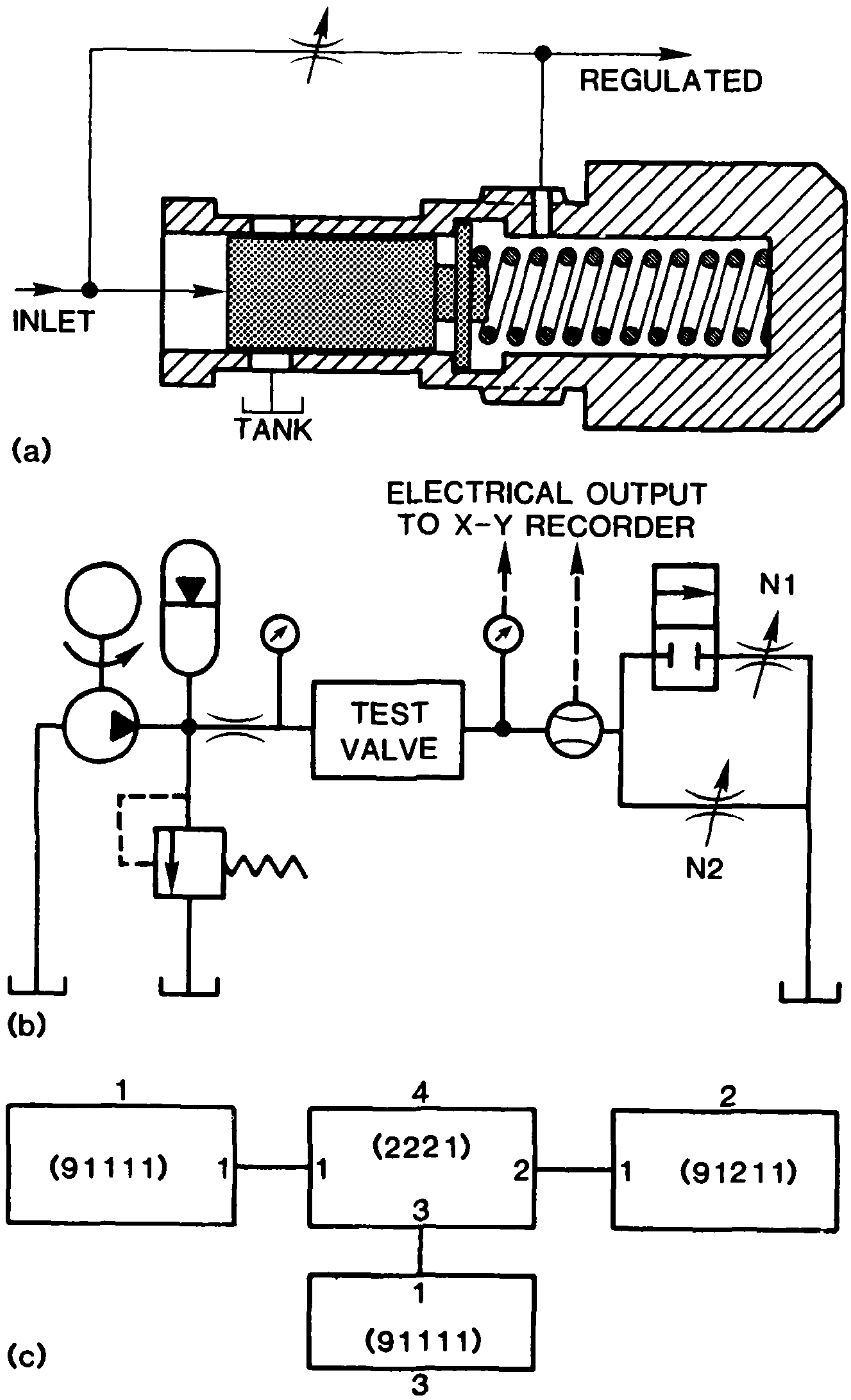


Fig. 4.13. The Test System of 2-way Pressure Compensated Flow Control Valve

D. Results Presentation

1. Static performance: See Fig. 4.14.

2. Dynamic performance: See Fig. 4.15

TEST 6

A. Test Component

1. Name: Bypass type 3-way pressure compensated flow control valve.

2. I. D. No. 2222

3. Schematic diagram: See Fig. 4.16 (a).

B. Experimental Verification

1. Set-up: See Fig. 4.16 (b). The test circuit is similar to that used in Test 5 except there is a relief valve used in the bypass port of the test valve to regulate the required bypass pressure.

2. Test Procedure

<u>Static Test</u> (same as in the Test 5).

<u>Dynamic Test</u>

1. Install a test valve in the test circuit.

2. Adjust the test system relief valve to the desired test pressure level.

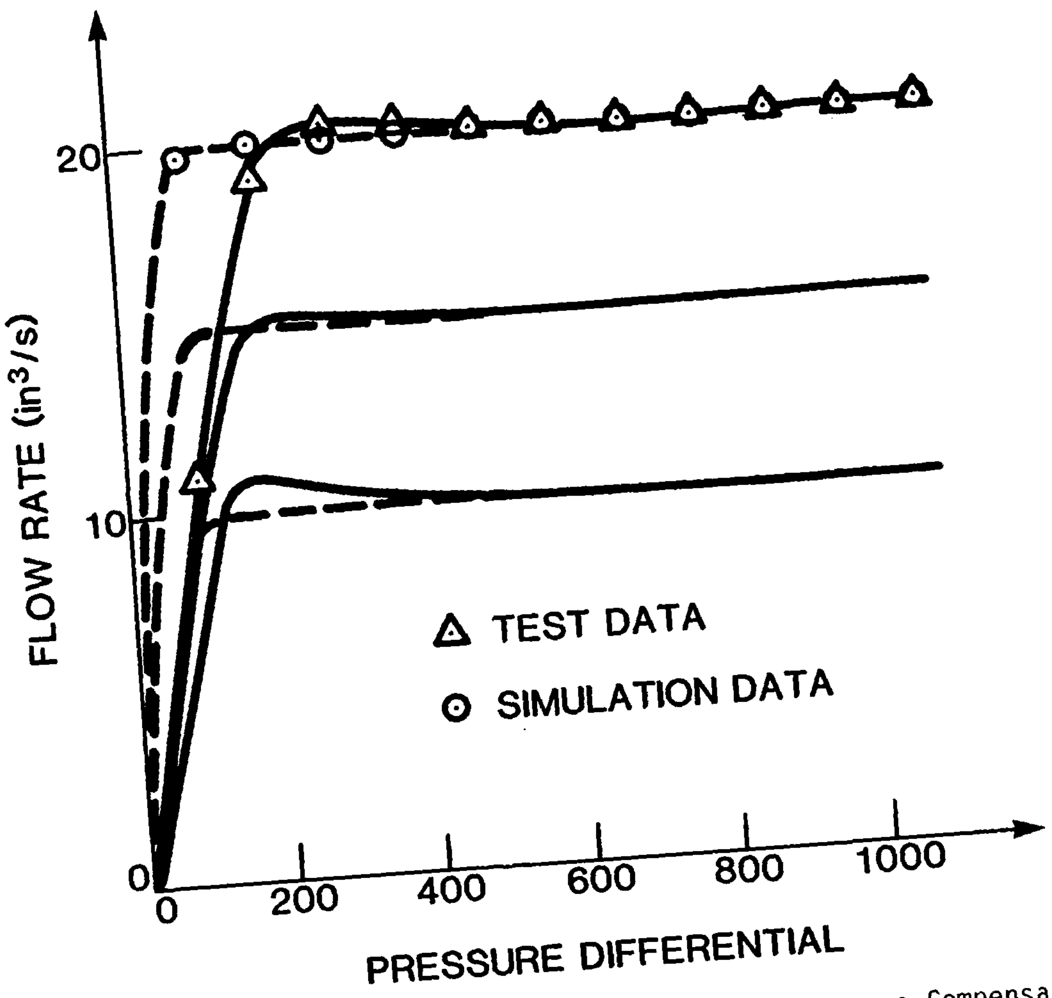


Fig. 4.14. The Static Characteristic of 2-way Pressure Compensated Flow Control Valve

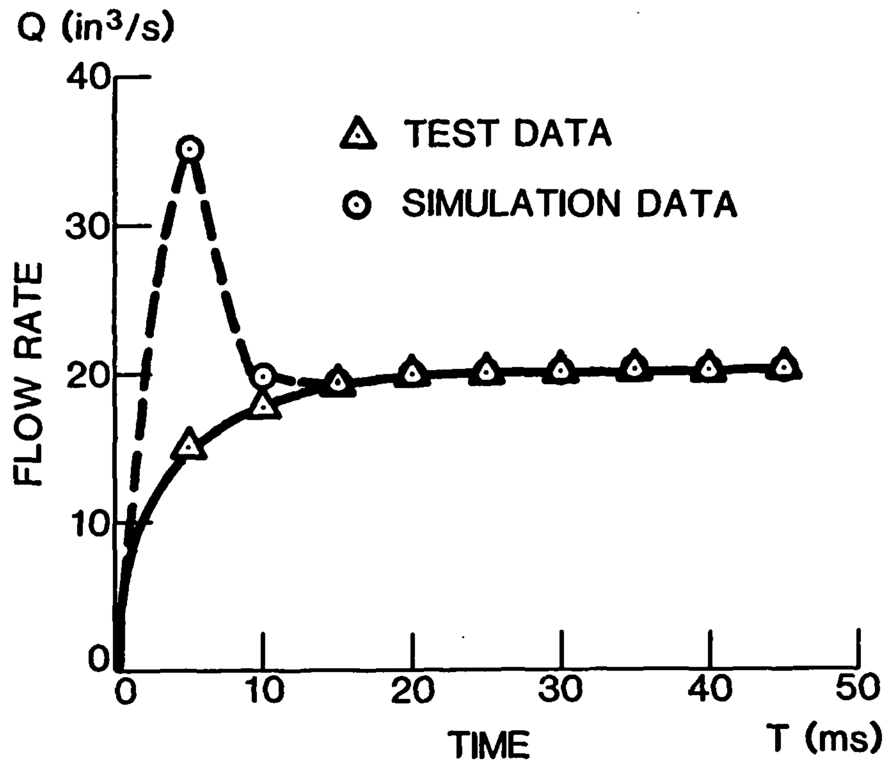


Fig. 4.15. The Dynamic Characteristic of 2-way Pressure Compensated Flow Control Valve

3. Set the test flow rate and load pressure by adjusting the test valve and needle valve 1 with needle valve 2 closed.
4. Set the bypass port pressure to the desired level by adjusting the bypass line relief valve.
5. Close the solenoid valve.
6. Apply a step input of the load pressure by opening the solenoid valve.
7. Record the controlled flow and load pressure as a function of time until the steady state condition is reached.

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.16 (c). The operating function is same as that described in Test 5.
2. Input Data:

   System relief pressure : 1524 PSI

   Opening area of adjustable orifice: 0.012 in^2

   Compensator spool reaction area: 0.307 in^2

   Area of damping orifice: 0.00071 in^2

   Spring constant: 71.1 lbf/in.

   Preload displacement of spring: 0.316 in.

   Opening displacement of orifice at bypass port: 0.183 in.

   Flow discharge coefficient of compensator orifice at regulated port: 0.6

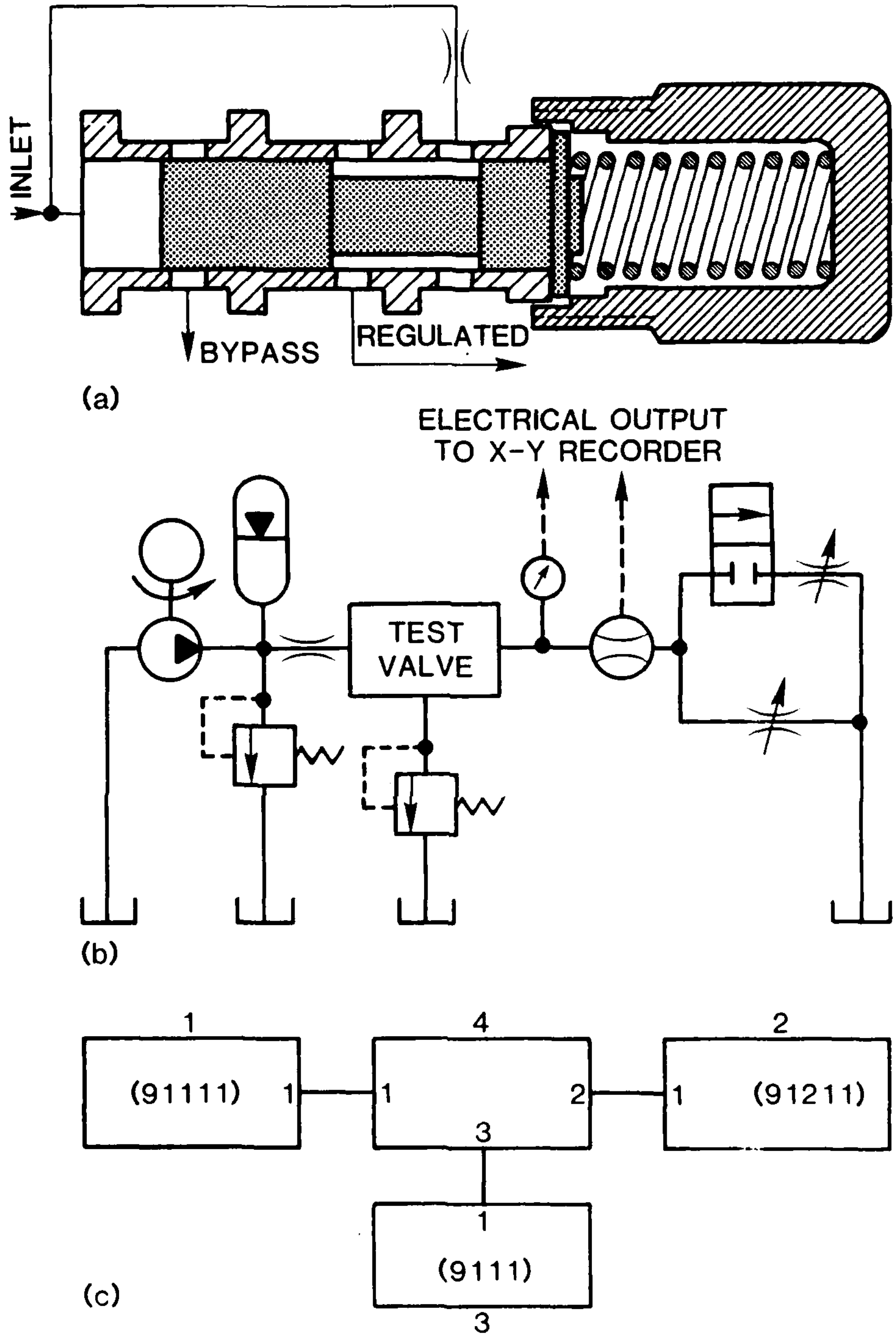


Fig. 4.16. The Test System of 3-way Pressure Compensated Flow Control Valve

Flow discharge coefficient of compensator orifice at bypass port: 0.6

Flow discharge coefficient of adjustable orifice: 0.6

Flow discharge coefficient of damping orifice: 0.6

Area gradient of compensator orifice at regulated port: 0.575 in^2/in

Flow jet angle of compensator orifice: 69 degrees

Maximum compensator spool displacement: 0.222 in.

Minimum compensator spool displacement: 0.184 in.

Mass of compensator spool: 0.000472 $lbf\text{-}sec^2/in.$

Viscous damping coefficient: 0.1 lbf-sec/in.

D. Result Presentation

1. Static performance: See Fig. 4.17.
2. Dynamic Performance: See Fig. 4.18.

TEST 7

A. Test Component

1. Name: Direct acting relief valve.
2. I. D. No.: 31111
3. Schematic Diagram: See Fig. 4.19(a).

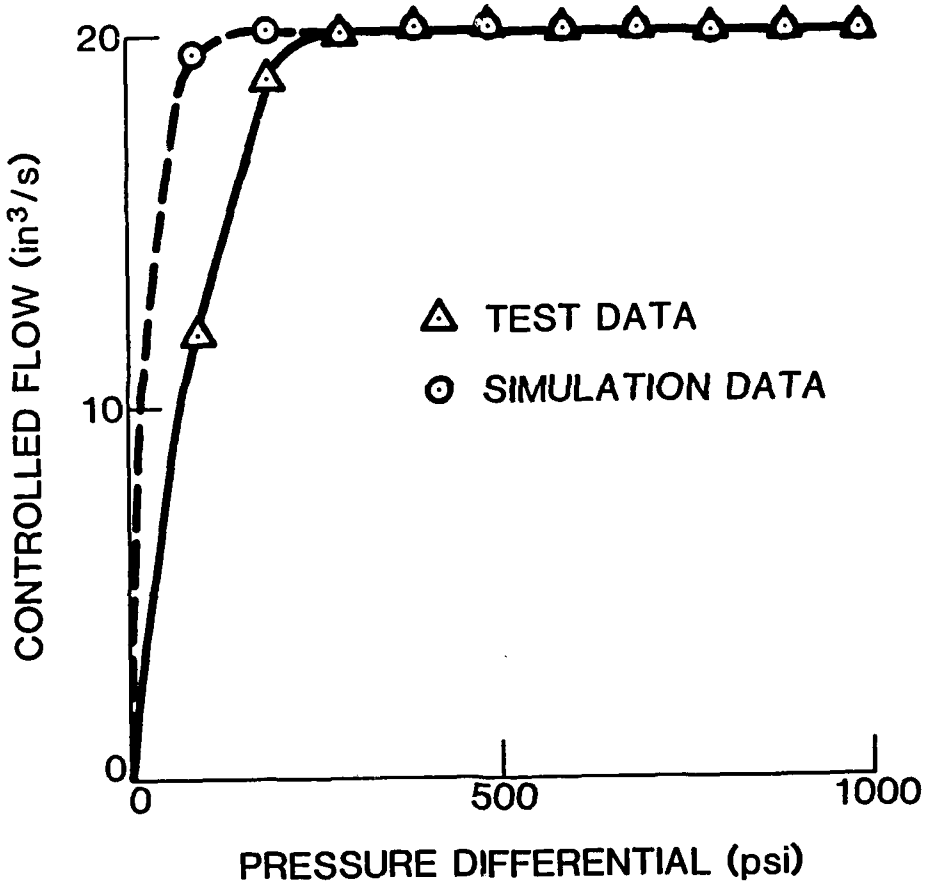


Fig. 4.17. The Static Characteristic of 3-way Pressure Compensated Flow Control Valve

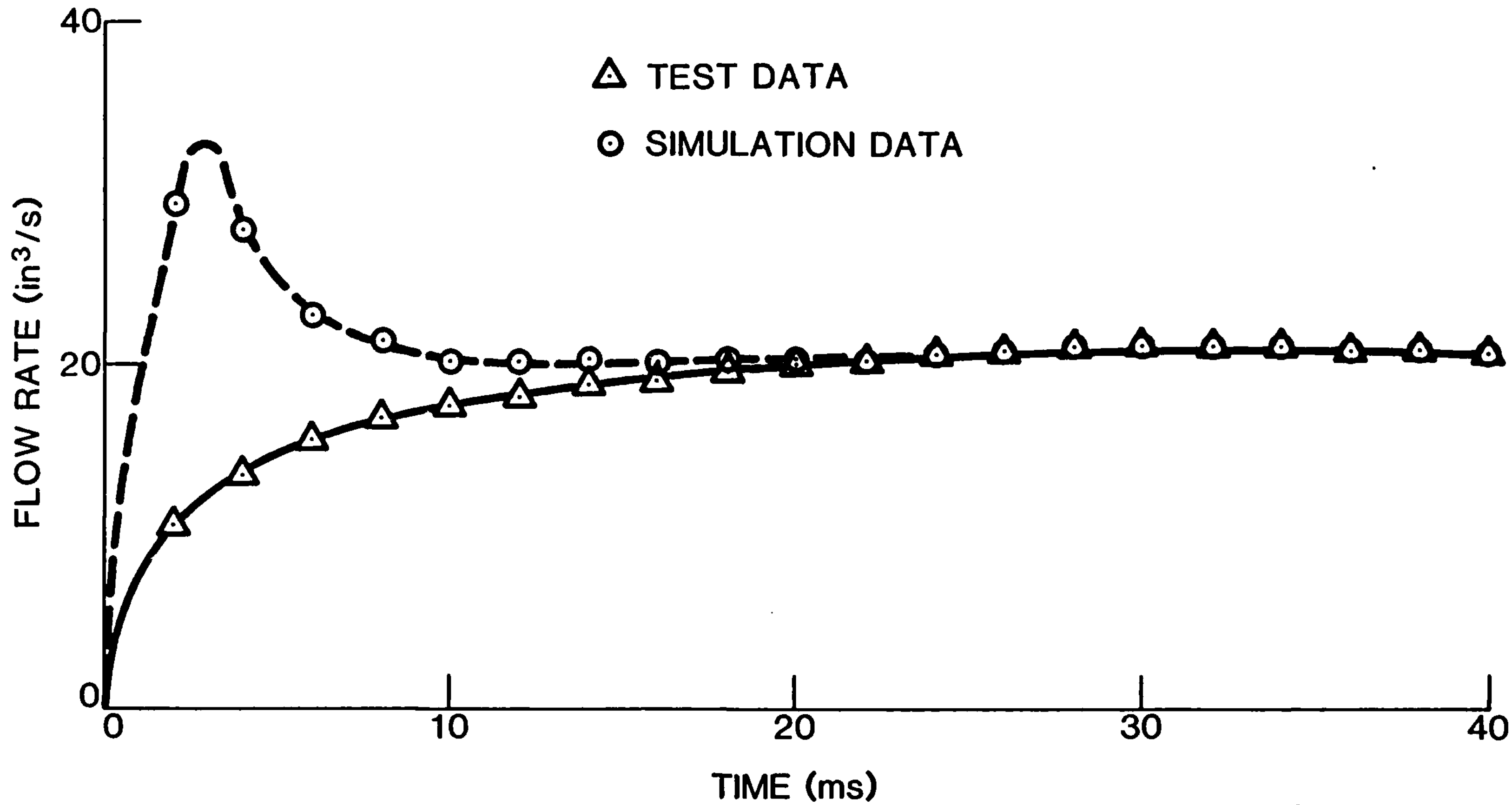


Fig. 4.18. The Dynamic Characteristic of 3-way Pressure Compensated Flow Control Valve.

Flow discharge coefficient of compensator orifice at bypass port: 0.6

Flow discharge coefficient of adjustable orifice: 0.6

Flow discharge coefficient of damping orifice: 0.6

Area gradient of compensator orifice at regulated port: 0.575 in^2/in

Flow jet angle of compensator orifice: 69 degrees

Maximum compensator spool displacement: 0.222 in.

Minimum compensator spool displacement: 0.184 in.

Mass of compensator spool: 0.000472 $lbf\text{-}sec^2/in.$

Viscous damping coefficient: 0.1 lbf-sec/in

D. Result Presentation

1. Static performance: See Fig. 4.17.
2. Dynamic Performance: See Fig. 4.18.

TEST 7

A. Test Component

1. Name: Direct acting relief valve.
2. I. D. No.: 31111
3. Schematic Diagram: See Fig. 4.19(a).

B. Experimental Verification

1. Set-up: Figure 4.19(b), illustrates the test system used for the dynamic and static performance test of a direct acting relief valve. The circuit includes:

   - a fixed displacement pump which drives flow at 20 in^3/sec.

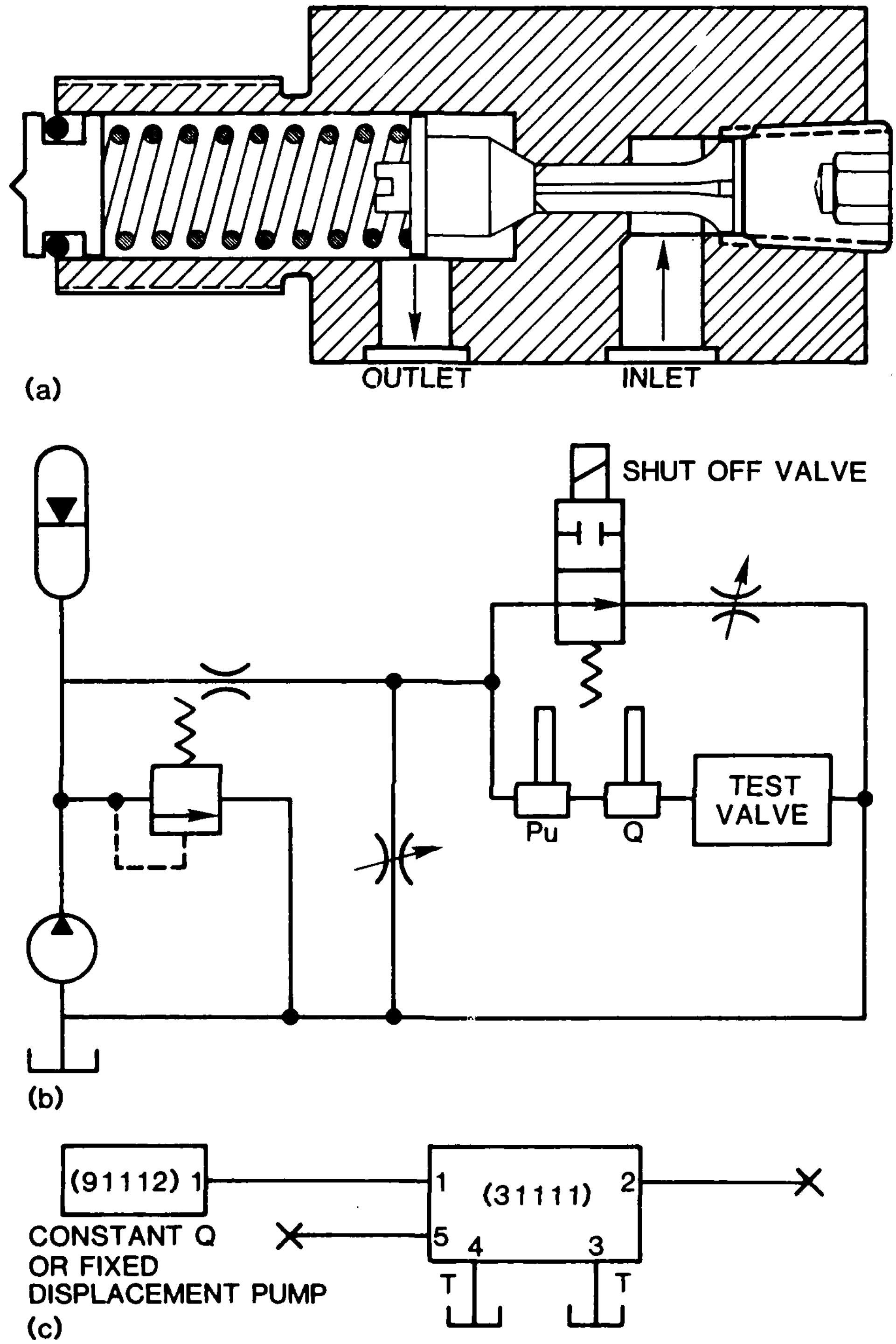


Fig. 4.19. The Test System of Direct Acting Relief Valve

- a relief valve to accomplish a constant supply pressure for the test valve.
- an accumulator and an orifice to filter our the excessive hydraulic pulsations.
- a A.C. signal to control the open/close function of the shut-off valve.

2. Test Procedure

Static Test

1. Install the test valve and achieve test temperature.
2. Open shut-off valve.
3. Adjust the flow control valve to vary the system flor rate.
4. Close the shur-off valve.
5. Measure and record the inlet pressure and outlet flow rate of the test valve.

Dynamic Test

1. Install the test valve and achieve test temperature.
2. Open the shur-off valve.
3. Adjust the system pressure and flow rate to 250 psi and 20 in^3/sec respectively.
4. Close the shur-off valve.
5. Measure and record the inlet pressure as a function of time.

C. Computer-Aided Simulation

1. Power flow circuit: See Fig. 4.19(c). Component (91112) provides a constant flow to component 2 (31111).

2. Input Data

(1) Cracking pressure: 800 PSI

(2) Upstream pressure receiving area: 0.05 in^2

(3) Downstream pressure receiving area: 0.05 in^2

(4) Spring constant: 500 lbf/in.

(5) Flow discharge coefficient: 0.61

(6) Area gradient: 0.389 in^2/in.

(7) Flow jet angle: 20^o

(8) Leakage flow coefficient: 0. in^3/sec/psi

(9) Mass of spool: 0.00012 lbf-sec^2/in.

(10) Fluid reaction volume: 50 in^3

(11) Maximum spool displacement: 0.2 in.

(12) Damping coefficient of the spool: 0.05

(13) Unsteady flow force coefficient: 0.

(14) System's initial pressure: 250 PSI

D. Result Presentation

1. Static performance: see Fig. 4.20

2. Dynamic performance: see Fig. 4.21

TEST 8

A. Test Component

1. Name: Pilot-operated relief valve

2. I. D. No. 3121

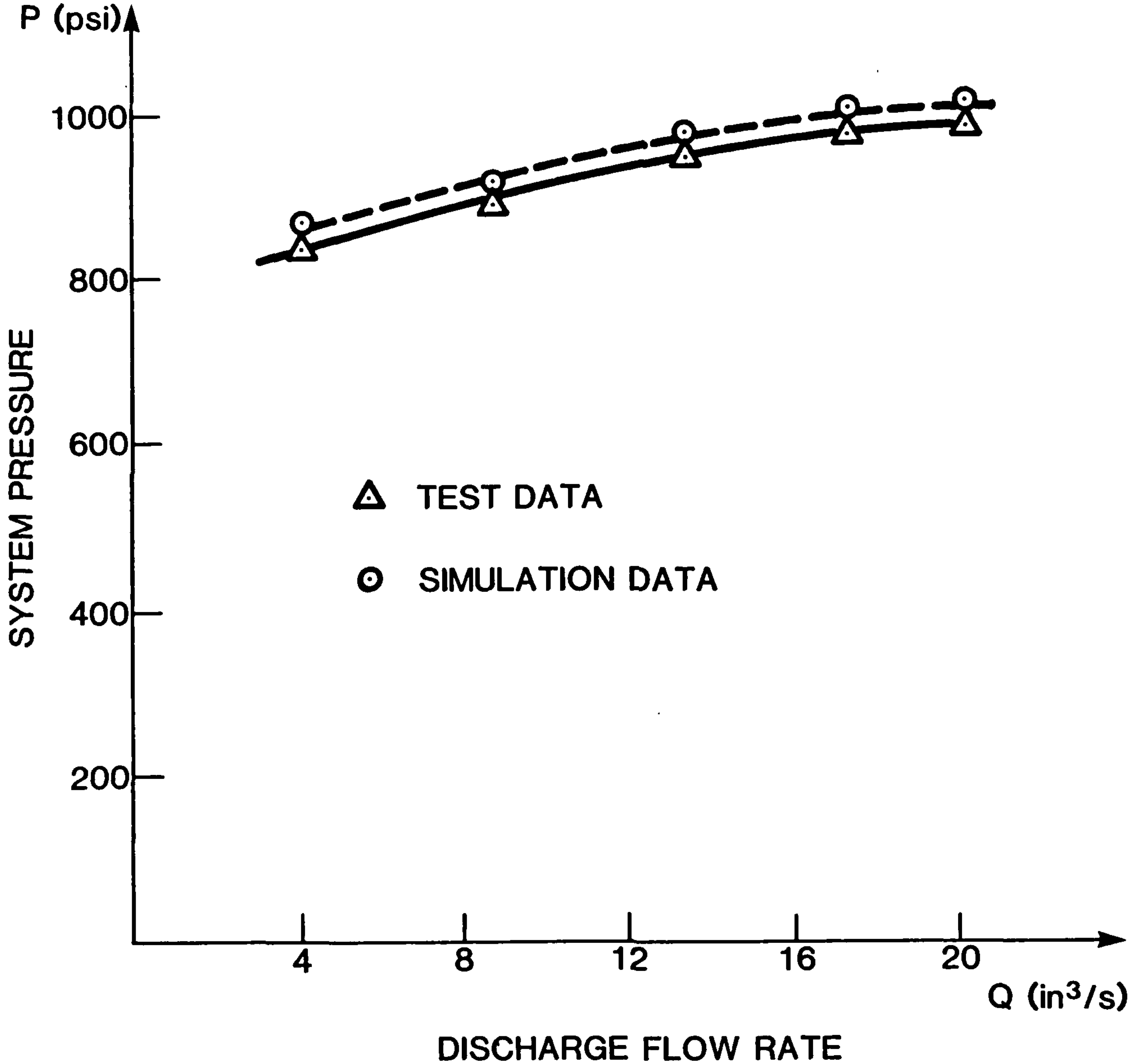


Fig. 4.20. The Static Characteristic of Direct Acting Relief Valve

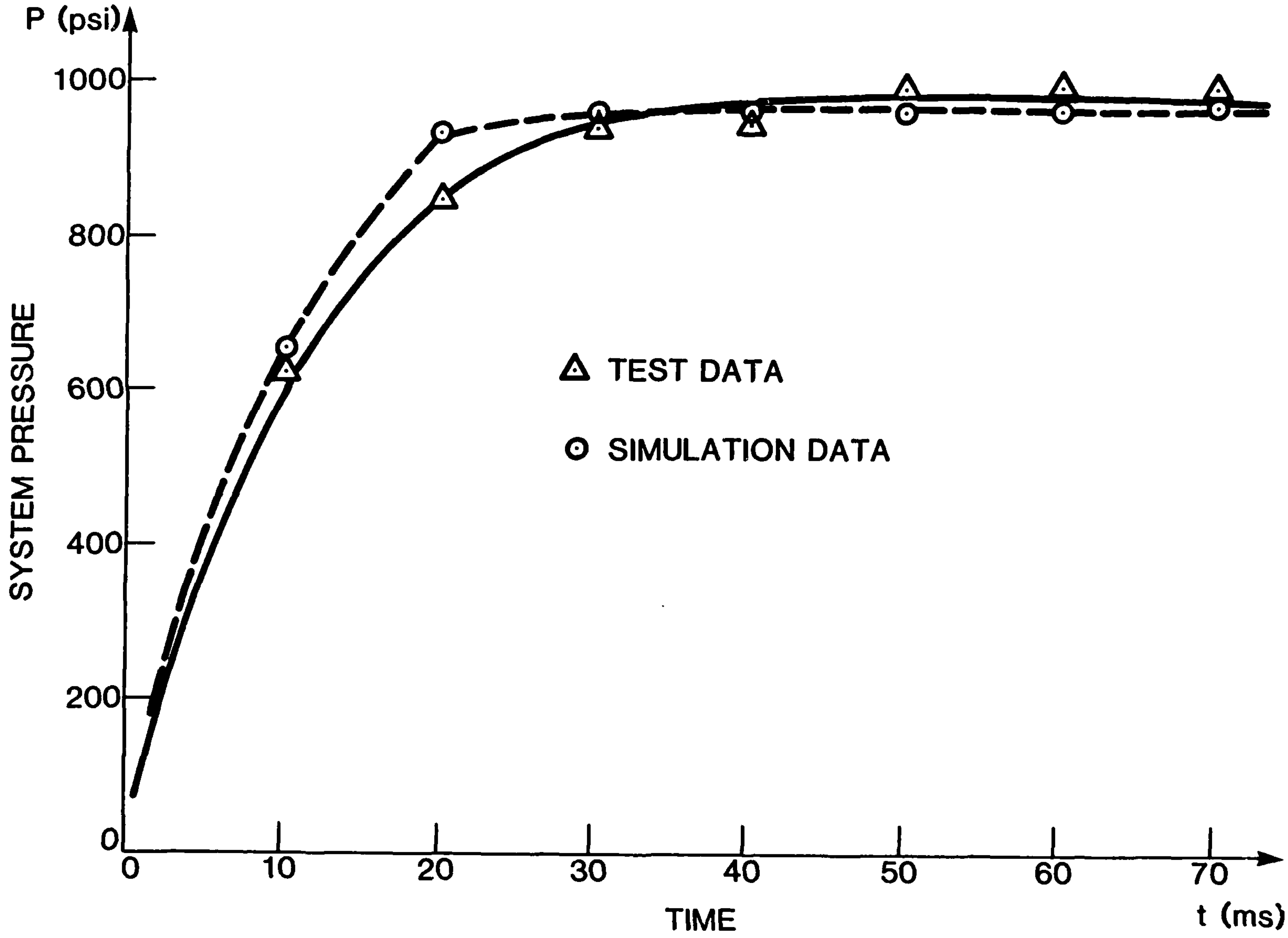


Fig. 4.21. The Dynamic Characteristic of Direct Acting Relief Valve.

3. Schematic Diagram: See Fig. 4.22(a).

B. Experimental Verification

(Same as the Experimental Verification section of Test 7)

C. Computer-Aided Simulation

1. Power flow circuit: see Fig. 4.22(c). Component 1 (91112) provides a constant flow to component 2 (3121).

2. Input Date

(1) Cracking pressure: 1000 PSI

(2) Main stage orifice coefficient: 0.61

(3) Main stage spring stiffness: 37 lbf/in

(4) Main stage preload: 5 lbfs

(5) Main stage outlet area: 0.196 in

(6) Pilot stage orifice coefficient: 0.61

(7) Pilot stage spring stiffness: 77 lbf/in

(8) Pilot stage outlet area: 0.0031 in^2

(9) Discharge coefficient of balance piston: 0.61

(10) Area of damping orifice: 0.00061 in^2

(11) Area of balance piston in main stage: 0.5 in^2

(12) Area of balance piston in pilot stage: 0.5 in^2

(13) Area gradient of main discharge port: 0.45 in^2/in

(14) Area gradient of pilot stage discharge port: 0.1626 in^2/in

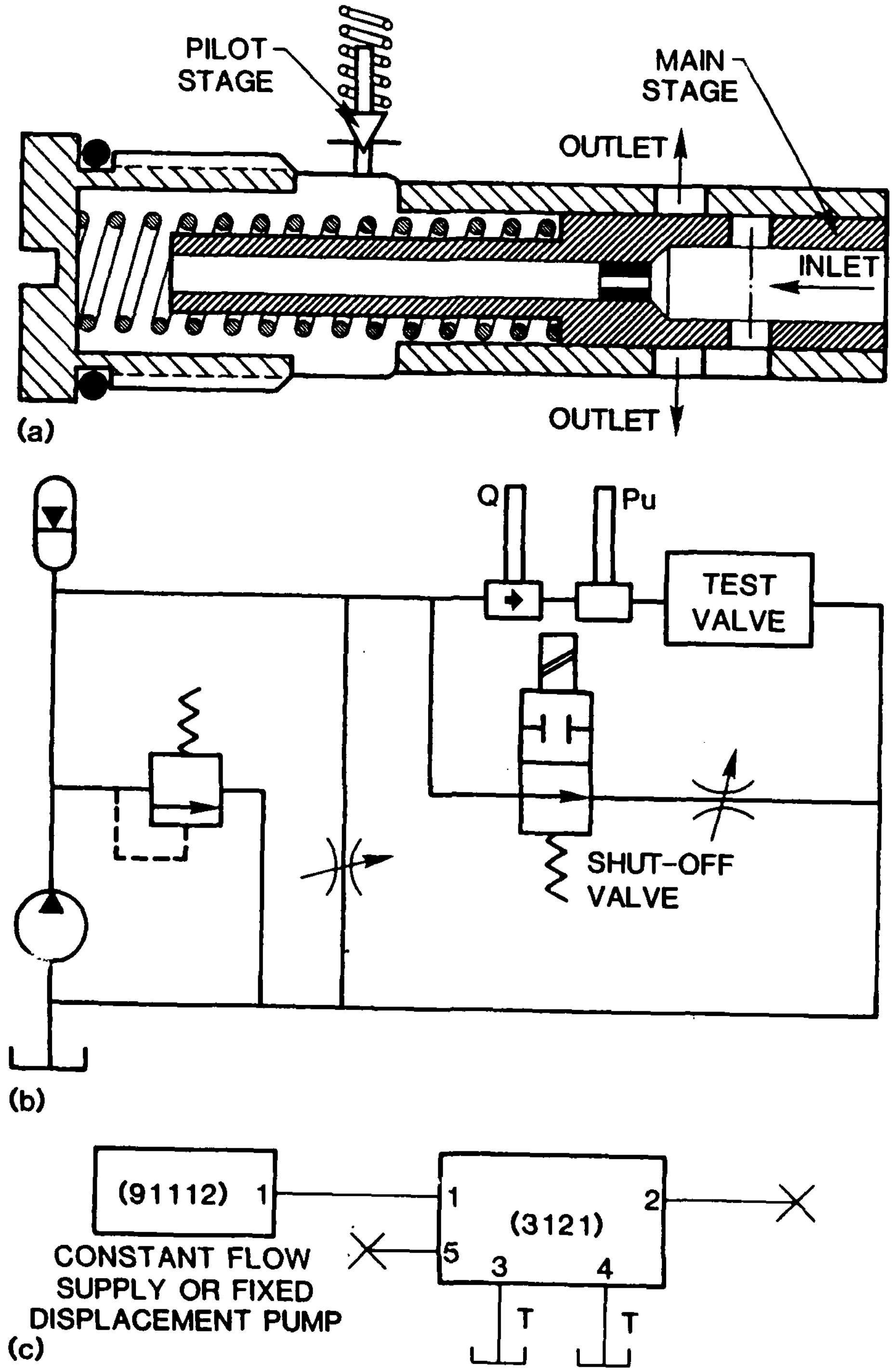


Fig. 4.22. The Test System of Pilot Operated Relief Valve

(15) Discharge angle at main stage: 69

(16) Discharge angle at pilot stage: 20^{o}

(17) Leakage coefficient of main stage: 0

(18) Leakage coefficient of pilot stage: 0

(19) Mass of main spool: 0.0502 lbs.

(20) Compression volume of main stage: 50 in^3

(21) Viscous damping coefficient: 1.09 lbf-sec/in

(22) Mass of pilot spool: 0.0045 lbs.

(23) Viscous damping coefficient: 0.58 lbf-sec/in

(24) Unsteady flow force coefficient on main spool: 0.1bf-sec/in

(25) Unsteady flow force coefficient on pilot spool: 0.1bf-sec/in

(26) Initial condition of system pressure: 250 PSI

D. Results Presentation

1. Static performance: see Fig. 4.23
2. Dynamic performance: see Fig. 4.24

TEST 9

A. Test Component

1. Name: Pilot-operated reducing valve
2. I. D. No: 3221
3. Schematic Diagram: see Fig. 4.25(a)

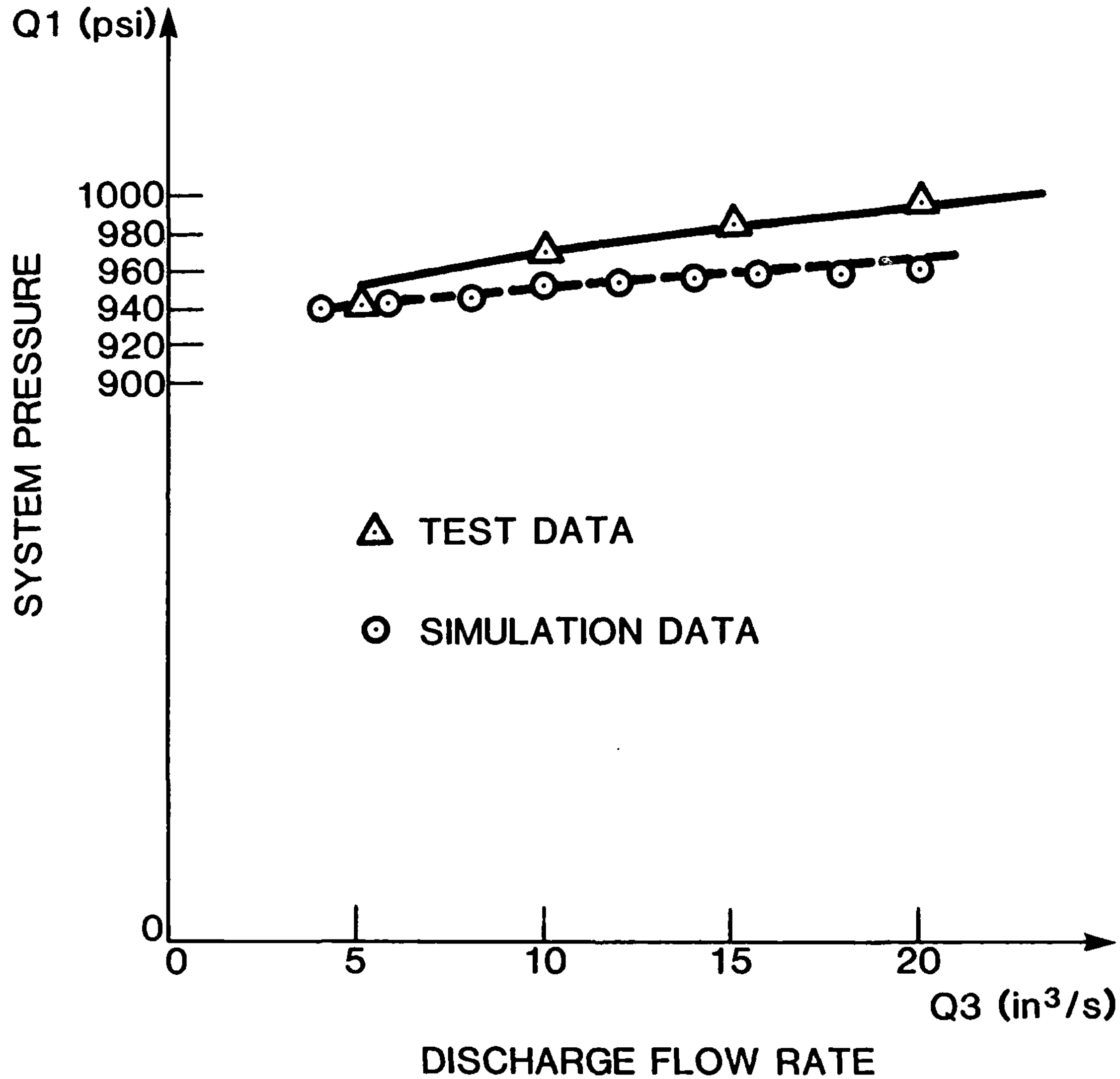


Fig. 4.23. The Static Characteristic of Pilot Operated Relief Valve

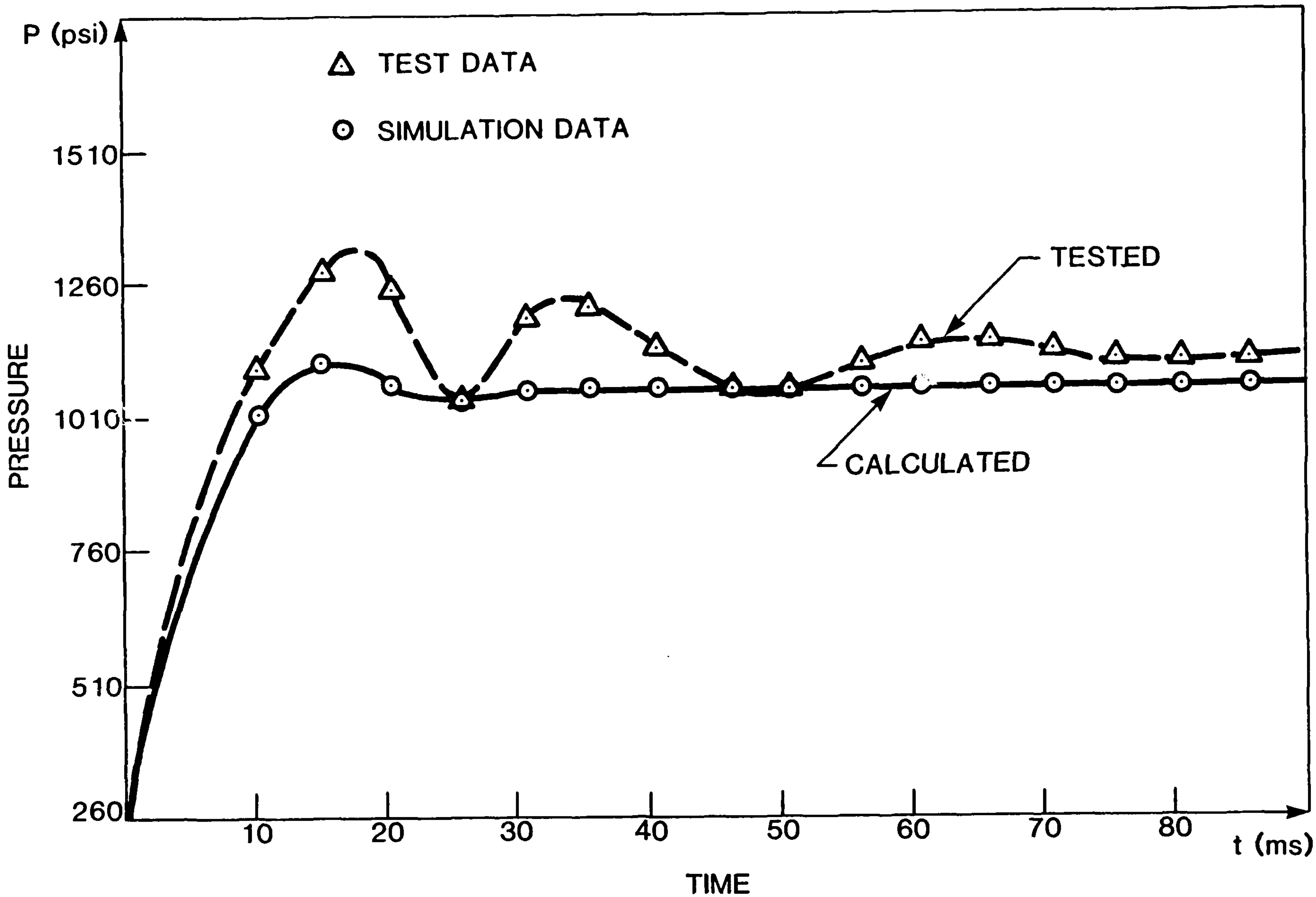


Fig. 4.24. The Dynamic Characteristic of Pilot Operated Relief Valve.

B. Experimental Verification

1. Set-Up: The same as the Set-Up in Test 7.

2. Test Procedure

<u>Static Test</u>

1. Install the test valve and achieve test temperature.
2. Open the shut-off valve
3. Adjust flow control valve to vary the system pressure.
4. Close the shut-off valve
5. Measure the upstream and downstream pressure of the test valve.

<u>Dynamic Test</u>

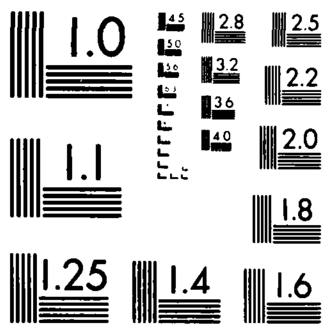
1. Install the test valve and achieve test temperature,
2. Open the shut-off valve
3. Adjust the upstream pressure of the test valve to 800 psi
4. Close the shut-off valve
5. Measure and record the downstream pressure as a function of time.

C. Computer-Aided Simulation

1. Power-flow circuit: see Fig. 4.25(c). Component 1 (91111) provides a constant pressure to component 2 (3221). Component 3 (91112) provides constant flow rate to component 2.

AD-A149 196 USER-ORIENTED COMPUTER-AIDED HYDRAULIC SYSTEM DESIGN 2/2
(U) OKLAHOMA STATE UNIV STILLWATER FLUID POWER RESEARCH
CENTER I T HONG JUN 83 OSU-FPRC-83-A-F1
UNCLASSIFIED DAAK70-81-C-0042 F/G 13/7 NL

END
FILMED
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

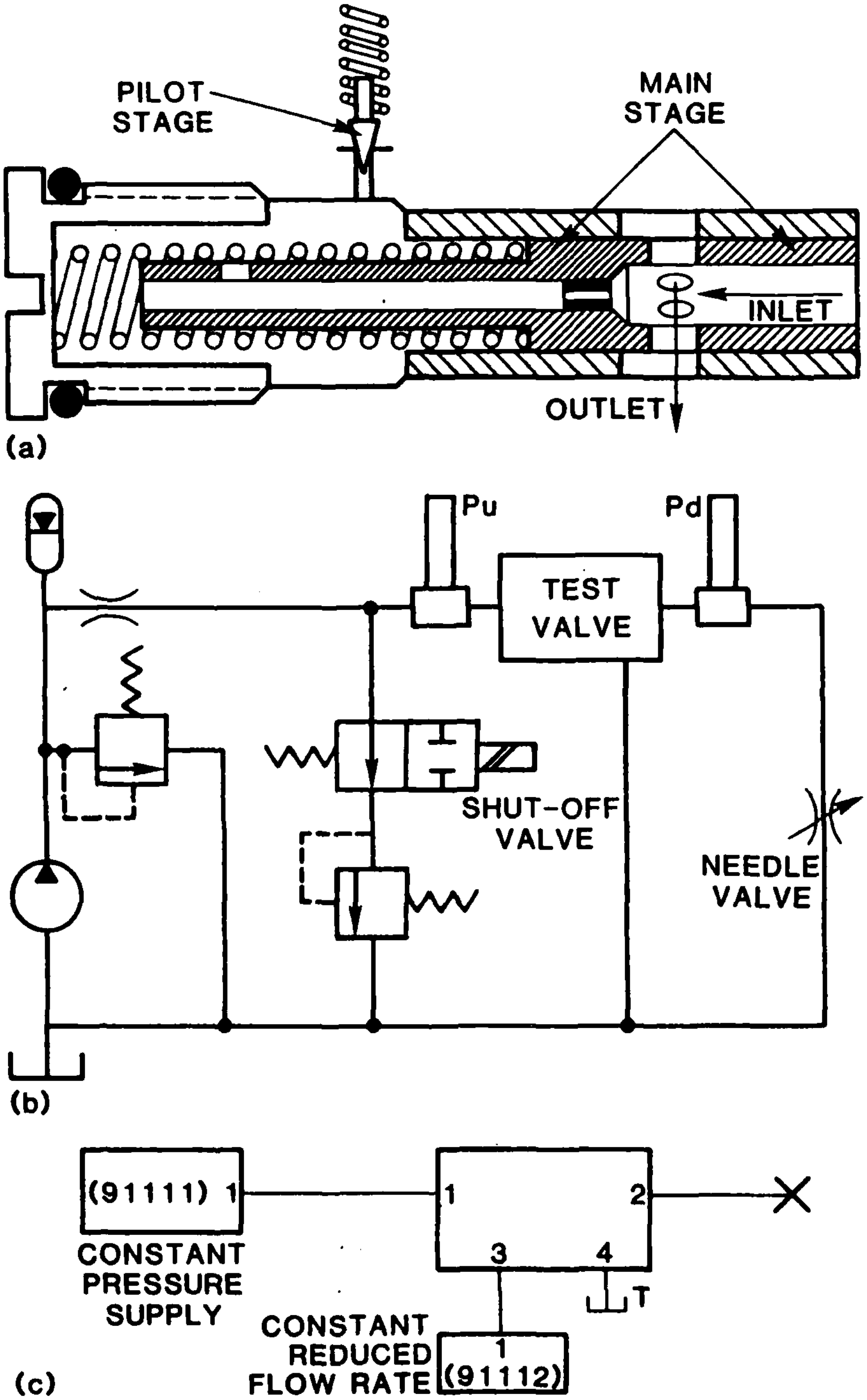


Fig. 4.25. The Test System of Pilot Operated Reducing Valve

2. Input Data

(1) Cracking Pressure: 250 PSI

(2) Main stage orifice coefficient: 0.61

(3) Main stage spring stiffness: 37 lbf/in

(4) Main stage spring preload: 10 lbf

(5) Pilot stage orifice coefficient: 0.61

(6) Pilot stage spring stiffness: 80 lbf/in

(7) Pilot stage outlet area: 0.0031 in^2

(8) Discharge coefficient of balance piston: 0.001

(9) Area of orifice: 0.00061 in^2

(10) Area of piston in feedback side: 0.05 in^2

(11) Area of piston in pilot stage side: 0.05 in^2

(12) Area gradient of main discharge port: 0.45 in^2/in

(13) Area gradient of pilot discharge port: 0.1626 in^2/in

(14) Discharge angle at main stage: 69^o

(15) Discharge angle at pilot stage: 20^o

(16) Leakage coefficient of main stage: 0.

(17) Leakage coefficient of pilot stage: 0.

(18) Maximum displacement of main spool: 0.02 in.

(19) Maximum displacement of pilot spool: 0.02 in.

(20) Mass of main spool: 0.00013 lbs.

(21) Mass of pilot poppet: 0.0000114 lbs.

(22) Compression volume of main stage: 50 in^3

(23) Viscous damping coefficient of main spool: 0.08 lbf-sec/in

(24) Compression volume of pilot stage: 5 in^3

(25) Viscous damping coefficient of pilot poppet:

0.036 lbf-sec/in.

(26) Unsteady flow force on main spool: 0. lbf-sec/in

(27) Unsteady flow force on pilot poppet: 0. lbf-sec/in

(28) Initial condition of output pressure: 250 PSI

D. Result Presentation

1. Static performance: see Fig. 4.26

2. Dynamic performance: see Fig. 4.27

Discussion of Tests

Because of the similarity in test procedures, test conditions, and test valve configuration, the discussion of tests is divided into three categories: directional control valve, flow control valve, and pressure control valve.

Directional Control Valves

Static Characteristic

Figures 4.2, 4.5, and 4.8 illustrate the comparison between the test results and simulation results of the static characteristic of direction control valves. It is seen that the results correlate very well. This was expected because the pressure-flow characteristic of most directional control valves is governed by the orifice

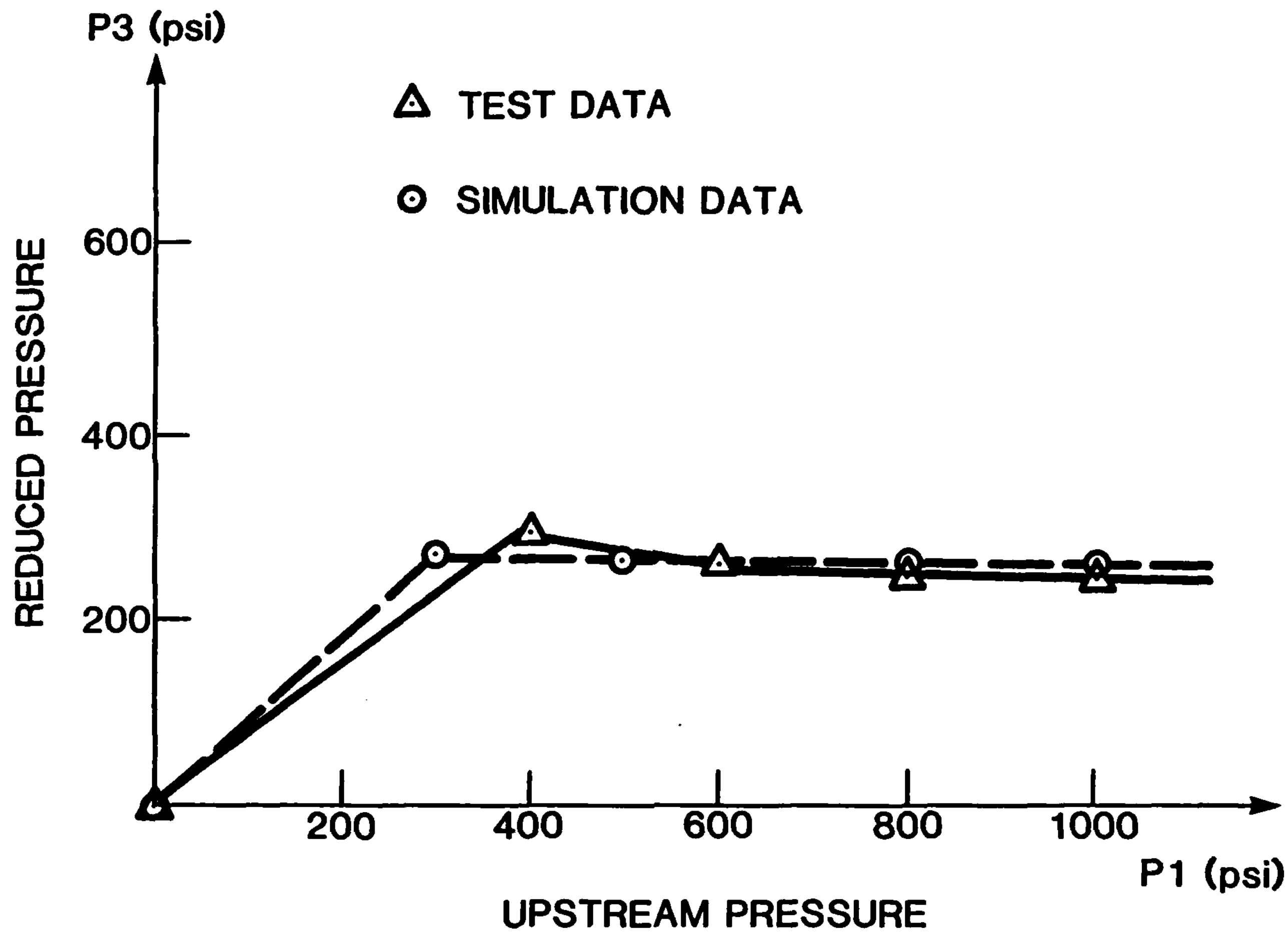


Fig. 4.26. The Static Characteristic of Pilot Operated Reducing Valve

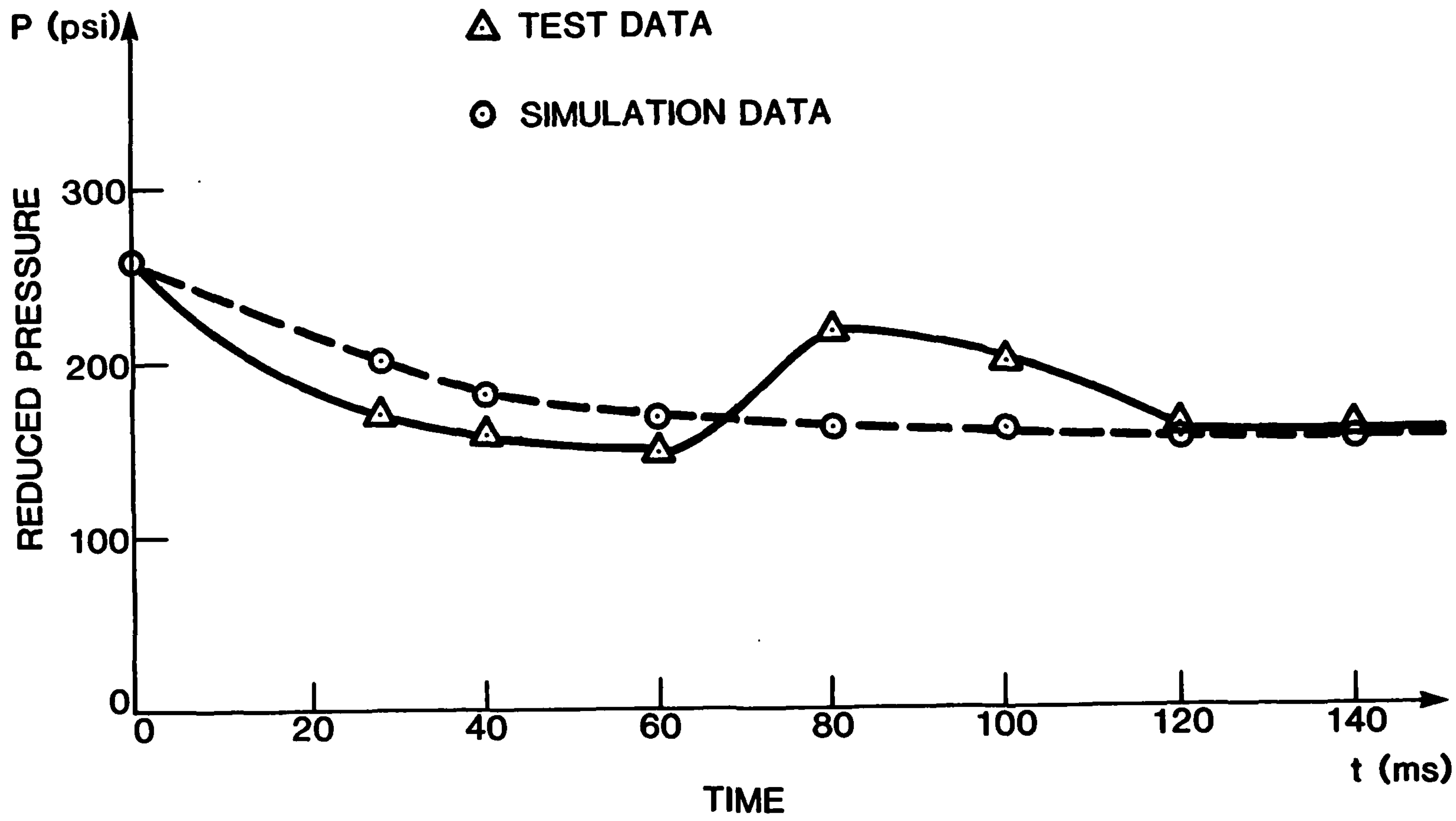


Fig. 4.27. The Dynamic Characteristic of Pilot Operated Reducing Valve.

discharge equation which is used in the directional control valve models. Due to the specification of the valves and the operation limitation of the test system, no observations were made when differential pressure across the upstream and downstream of the valve was less than 50 psi.

Dynamic Characteristic

Figures 4.3, 4.6, and 4.9 illustrate the comparison between the test results and simulation results of the dynamic characteristic of directional control valves. It is found that there is a good result coincidence of the 2-way directional control valve. However, results of 3-way and 4-way valves do not coincide well in the first few points. The reason that causes the discrepancy is that there are no informative data available to describe the characteristic of the solenoid actuation force. In this study, it was assumed that the valve was actuated by a step electric signal (force) only.

Flow Control Valves

Static Characteristic

Figures 4.11, 4.14, and 4.17 show the static characteristic curves of flow control valves. It appears that the simulation results coincide well with the test results except in the low differential pressure range. There are several reasons for the discrepancy, primarily that the spring might not function linearly throughout the operating range. In addition, it is difficult to evaluate the steady state force accurately because its magnitude alters with the jet angle when the fluid pass through the orifice. The angle has been reported to depend upon the displacement of the spool opening, although a 69-degree jet angle is widely accepted by most designers and researchers.

The discrepancy could also be caused by the non-ideal characteristics of the components which were used to establish the test condition. The surrounding components such as the system relief valve, accumulator, and the conduits are presumed to have an ideal characteristic in the simulation. It is planned to further investigate the entire actual system characteristics in Phase III.

It is noted that the system relief valve contributes strong influence on the test performance of the flow control valve. The major cause of the non-ideality stems from the pressure override of the relief valve which causes the supply pressure to vary according to the amount of relief flow. Thus, at high load pressure, the flow through the test valve is not enough to actuate the compensator; therefore, no flow control can be accomplished. The excessive flow generated by the pump is then bypassed through the relief valve. Due to the pressure override, the supply pressure for the test valve tends to increase as the load pressure raises. As a result, the differential pressure could not increase at the same rate as that of the decreased load pressure. Therefore, a longer period of unregulated flow was expected in the experimental results.

In spite of the discrepancy, the validity of the simulation results can be made through physical explanation. Figure 4.28 shows the characteristics of the orifice equation, the simulation, and the experimental results of a flow control valve. Obviously, the functional mechanism of a pressure compensated flow control valve can be divided into three modes in terms of the function of the compensator: inactive, semiactive, and active modes.

In the first mode, the compensation mechanism is inactive because of the low actuation force created by the low flow rate across the adjustable orifice. The valve characteristic curve just follows the orifice equation because there is an adjustable orifice upstream of the compensator, Fig. 4.28.

In the second mode, the compensator starts to function, but it does not receive enough force to compensate for the flow.

In the third mode, the compensator functions to retain the regulated flow at a constant flow level in spite of the load pressure.

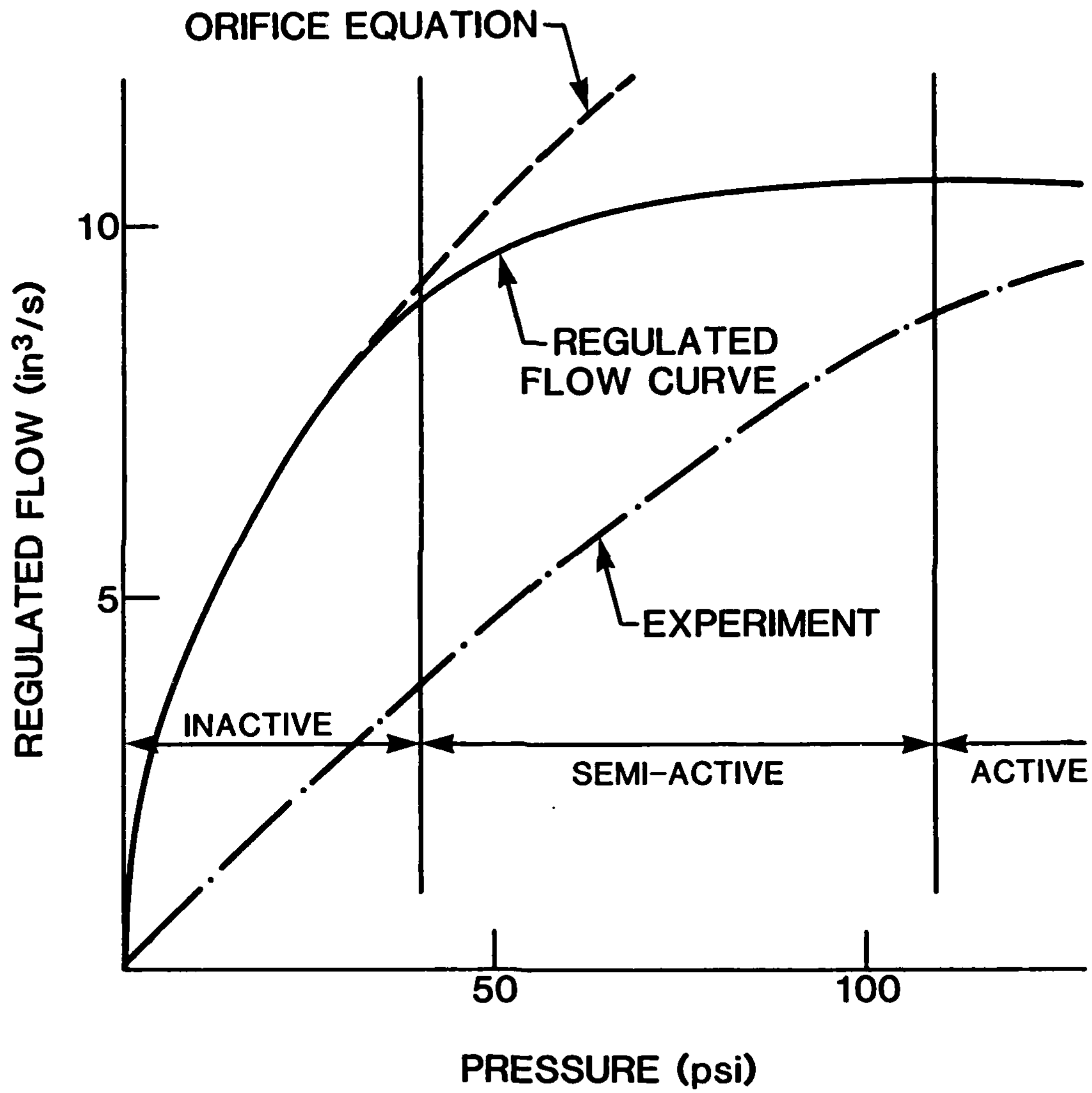


Fig. 4.28. Explanation of the Operating Modes of Flow Control Valve.

Dynamic Characteristic

Figures 4.12, 4.15, and 4.18 show that the experimental results behave similar to a first order system response, while the simulation results behave similar to a second order system (overshoot exists). This discrepancy is generated from the different boundary conditions used between the experiment and the simulation. The simulation condition was set without any hydraulic conduits effect (the effect of conduit will be studied in detail - Phase III), which play an important role in flow response. In the experiment, there are hydraulic lines included. The hydraulic lines functional as a filter and absorbed the high frequency contents of the natural frequency of the flow control valves. This effect minimizes the occurence of the overshoot of flow response. However, there is no hydraulic line effect considered in the simulation. Therefore, the discrepancy occurs in the starting period and then decreases due to the pressure compensation mechanisms which corrected the discrepancy automatically. Thus if the models for transmission line dynamics are considered, the simulation results will be closer to the actual experimental results.

Pressure Control Valves

Static Characteristics

The static characteristics of pressure control valves, Fig. 4.20, 4.23, and 4.26, show a good agreement between the simulation results and the experimental results. An average simulation error obtained is less than 5 percent.

Dynamic Characteristics

The dynamic characteristics of pressure control valves are shown in Fig. 4.21, 4.24, and 4.27. In general, there is a good correlation between the simulation results and the experimental results. The errors occured at the overshoot points during the starting period. Like the reasons of discrepancy pointed out in the discussion of other type of control valves, the errors may be generated due to the non-ideal condition of the test system. For instance, the consideration of hydraulic line dynamics, the non-linearity of the spring, etc.

Obviously, from the above discussions, it is found that the hydraulic line dynamic is one of the major factors affecting system performance. In order to minimize the simulation error, the characteristic of hydraulic line as well as the integrity of system simulation should be considered. These factors are the principle objectives of Phase III. As can be expected, the simulation error will be reduced to the minimum after the completion of Phase III.

CHAPTER V

DISCUSSION

During the second year of this effort, the objectives of Phase II have been met. Specifically, the results are as follows:

1. Nine hydraulic control valves were selected and tested to verify the valve models developed in Phase I. The test valves include three of each of directional control valves, flow rate control valves, and pressure control valves. Both the static performance and dynamic performance were investigated.
2. The CAAS program was used to simulate the performance of the test valves. The actual component design data were used in the simulation.
3. The simulation results were compared to the actual test results. It was found that most of the static performances coincide very well with the test results. However, in the dynamic simulation, the results do not coincide well during the first few points of starting period. After that period, it behaves well, although there were discrepancies, the validity of the simulation results can be made through physical explanation (see Discussion of Tests in Chapter IV). The major factors that affect the accuracy of the developed

models can be summarized as:

- Hydraulic line effects were not considered in the simulation (The Phenomena of transmission line will be investigated in Phase III.)
- Some of the parametric coefficients, for instance the orifice discharge coefficient and the spring stiffness coefficient, are either set to be a constant or to bear a linear relationship. These are widely accepted approaches in hydraulic performance simulation. Nevertheless, they may not behave so well in an actual system. As a result, some discrepancy may occur.
- The purpose of a mathematical model is intended to represent a physical system as close to the actual condition as possible. However, a model essentially is an "ideal" description of the system. This inherently generates some discrepancy.

4. Twenty-seven component models were developed in Phase II. These include 11 hydraulic pump models, five hydraulic motor models, and 11 hydraulic cylinder models. The details are listed in Figs. 2.1, 2.3, 2.4, respectively.

5. The CAAS program was entirely re-written in the FORTRAN language. This activity included the conversion of Problem Oriented Language from the PL/I language to the FORTRAN

language, the modification of component models that they might be compatible to the new CAAS program structure, the expansion of the package interactive function (for example, the function of input data reconfirmation and on-line data correction), and the development of on-line user's help modules.

6. A User's Manual for the CAAS package is furnished. The manual enables any hydraulic engineer who is inexperienced in computer work to operate the program. It is included in the Appendix of this report.

7. A Maintainer's Manual of the CAAS program is also furnished. The Manual is intended to serve as a guide to programmers responsible for maintaining or updating the CAAS package. It consists of the entire program listing, the description of programs, the cross-reference of every variable used in the program, the engineering information of component models, flow-charts of the entire control program and major subprograms, formats of disk files used by the program, and other vital topics.

8. A magnetic tape containing the entire CAAS program developed to date has been provided for MERADCOM use.

CHAPTER VI

CONCLUSIONS AND RECOMMENDATIONS

The objectives of Phase II have been achieved. Throughout the efforts of Phase I and II, the most commonly used hydraulic valves, pumps, motors, and cylinders have been developed. The results of the experimental work show that the developed power-flow technique and component models are valid to analyze the performance of hydraulic components. The success of the CAAS package in analyzing both the static and dynamic performance of individual components provides great confidence to extend the entire philosophy to complete the analysis of an actual hydraulic control system. It is also noted that the hydraulic line property may significantly affect the performance of components. As a result, it is necessary to develop the model for hydraulic line and fitting before the simulation of an entire hydraulic control system can be confidently carried out.

Consequently, it is recommended that Phase III of the project, including verification of the pump, motor, and cylinder models developed in Phase II, the development of the models of hydraulic transmission lines and fittings, the extension of the POL to analyze the actual hydraulic system, and the study of the fundamental basis of adapting degradation parameters (for instance, thermal, wear) into the CAAS program, begin immediately.

It is expected that after the completion of Phase III, the CAAS package should be able to analyze an actual hydraulic system. In addition, it will provide a firm basis for extending the CAAS program to achieve the long term objectives such as system optimization, system reliability, contamination sensitivity, micro-computer and computer graphics applications, process control, etc.

BIBLIOGRAPHY

1. Bashta, T. M., "Machine Construction Hydraulics, A Reference Manual," Translation Division Foreign Technology, WP AFB, Ohio, Nov. 13, 1973.
2. Blackburn, J. F. et. al., Fluid Power Control, The MIT Press, Massachusetts, 1960.
3. Doebelin, E. O., System Modelling and Response, Theoretical and Experimental Approaches, John Wiley & Sons, Inc., New York, 1980.
4. Faber, K., "Hydraulic Seal Efficienty as a Factor in Energy Conservation," National Conference on Fluid Power, 1974.
5. Felicio, L. C., "A Theoretical and Experimental Study of the Static and Dynamic Behavior of Vane-Type Pressure Compensated Hydraulic Pumps with Proportional-Type Regulator," Ph.D. Dissertation, Ohio State University, 1981.
6. He, Z. C., "Computer-Aided Simulation of Non-Rotating Cylinders," The BFPR Journal, 1983.
7. Hong, I. T., "User-Oriented Computer-Aided Hydraulic System Design," Interim Report, Contract No. DAAK70-81-C-0042, U.S. Army Mobility Equipment Research and Development Command, Fort Belvoir, Virginia, May, 1982.

8. Hydraulic Pumps, Industrial Pistons, Vane & Gear Types, Parker Fluid Power Catalog, No. 2600, 1980.

9. Industrial Hydraulics Manuals, Vickers Co., 1965.

10. Ishihara, T., et.al. Fluid Power Engineering Handbook, Asakura Book Ltd, Japan, 1972.

11. Merritt, H. E., Hydraulic Control Systems, John Wiley & Sons, Inc., New York, 1967.

12. Morris, A. E., Fluid Power Handbook and Directory, 1967.

13. Morse, W. L., Fluid Power Handbook, Design Engineering Handbooks, Summit House, 1968.

14. Stein, G., "Hydraulic System Pumps - 4, Piston Pumps," Machine Design, 1970.

15. Schinik, R., and J. Kauffman, "Hydraulic System Pumps - 3, Vane Pumps," Machine Design, 1970.

16. Stuart, R. and J. Holdeman, "Hydraulic System Pumps - 2, Gear Pumps," Machine Design, 1970.

17. Stuntz, R. M. et.al. "Hydraulic Component Modeling Manual," Annual Report of the Basic Fluid Power Research Program, Oklahoma State University, Stillwater, OK., Vol. 2, 1968.

18. Wilson, N. E., "Positive Displacement Pumps and Fluid Motors," Pitman Publishing Co., New York, 1950.

APPENDIX A

USER'S MANUAL

USER'S ORIENTED COMPUTER-AIDED HYDRAULIC SYSTEM DESIGN

USER'S MANUAL

The CAAS System

Version 2

W. R. Hensley and I. T. Hong

Fluid Power Research Center

Oklahoma State University

March 1983

TABLE OF CONTENTS

CHAPTER I

INTRODUCTION

The CAAS package is a group of computer programs written in the FORTRAN IV language (extended) which aid in the analysis and testing of hydraulic systems of many different types and configurations.

The version of CAAS for which this manual is written is Version 2. The purpose of this user's manual is to assist persons who want to utilize the CAAS system in order to minimize their individual time and effort spent in hydraulic design. The CAAS package simplifies the tasks of creating a workable design in a much shorter time by numerically simulating the system, enabling the designer to easily determine whether or not the system is performing as desired. This saves much time and effort which would traditionally be wasted either in the shop experimenting with components until the correct performance was found, or at a desk, performing laborious and complicated calculations to determine theoretical system performance.

Most of the critical component simulation procedures used in the CAAS system have been verified experimentally in the workshop. This assures the designer that a model which has been simulated will indeed perform as predicted.

This user's manual is divided into sections, each section being one of the areas of user communication with the CAAS system. The material is grouped in the order in which operations are performed as the user builds and simulates his model.

In addition to, and supplementing this user's manual, there is an on-line HELP facility build into the CAAS system. At any point where the user is being asked for input, the word HELP or the letter H may be typed, and the on-line tutorial file activated. There are two levels of the tutorial: the first time that the user enters H or HELP, a short one or two-line message explaining the desired input is displayed at his terminal. If the explanation is not clear enough, a second H or HELP entered immediately afterwards puts the user into the second level of the tutorial. This level of the tutorial consists of full-page explanations of the reason for and allowable limits for the input the CAAS system needs from the user. The user can move around through the tutorial at will, examining any part of the topics covered. When the user is satisfied that he understands what the system wants from him, he can exit the tutorial and the system reminds him of what it needs by re-displaying the question. The only exception to this procedure can be found when logging on the system. While being asked for his user I.D. number or password, the user only has access to Section 2 in the tutorial (Section 2 covers the procedures necessary to log on the system). The tutorial also has a short glossary of design terms that are frequently used in hydraulic design work and their meanings.

The user also has one other means of obtaining on-line help from the system. There are many situations in design work where a variable depends on several other parameters in a system. These various coefficients and constants usually have to be calculated by hand for use in a simulation. The CAAS system has several routines which can perform the calculations for the user, at the time that the user needs to have them calculated. Some of these procedures include a least-squares approximation (for calculating the coefficients of a performance curve), a procedure for determining the unsteady-state flow in a region, finding the area gradient of a discharge orifice, and several others which can further save time, energy, and effort during the design of a hydraulic system. If such a procedure related to the question being requested is available, then the user will be informed when he conducts the first level HELP.

CHAPTER II

WORKING WITH THE CAAS SYSTEM

2.1. General Consideration of the CAAS Program

The CAAS system is divided into several logical units. The major units are:

Defining the CAAS environment: The CAAS system is designed to be run by users with different computer terminal equipment. CAAS needs the user to tell it what kind of a terminal he is using, as this is a factor in several of CAAS's actions (such as output plotting, screen clearing, and so on).

Selecting components to be simulated: The CAAS system has a database residing on magnetic disk which contains all of the possible components that the system is capable of simulating. The user is shown a series of component menus that get increasingly detailed, until the component is completely defined. The user is allowed to 'back up' in the series of menus if he believes that he is in the wrong place, or can delete a component that he feels should not be in his model.

Defining the nature (type) of the simulation to be performed: The CAAS system can simulate systems both statically (time-independent) and dynamically (time-dependent). For static simulations (also called steady-state), two different modes of operation are available. For the dynamic simulation, two different numerical integration procedures are available, and the user can select all starting and ending times, along with the output step size and the time interval between system sampling.

Defining component properties: The CAAS system communicates with the user interactively in order to define component properties. It allows the user the option of selecting either the emperical data or detail component design data for system simulation.

Defining the system interconnections: The CAAS system needs to know how the components which have been selected are connected to each other. A table showing all of the interconnections made so far show the user exactly what he has specified so far, and the program allows the user to alter the connections at anytime before he goes into the simulation section.

Examining the results of the simulation: The CAAS system allows the user the option of producing tables or plots of the output results, and allows any number of tables or plots to be produced and observed.

Other factors: The CAAS system allows the user to examine and vary such system-dependent parameters as fluid properties, system conditions, and external conditions.

Each of these major functions of the CAAS system are covered in complete detail in the following sections, along with other functions which are important, but are global in nature (the on-line tutorial is an example of this).

Each section is cross-referenced to other sections when necessary to allow greater understanding of the interconnections between the various subsystems of CAAS.

2.2. Logging on to the CAAS System (A)*

The CAAS program is equipped with a security system that allows each separate user to have his own user I.D. number and confidential password. After invoking the package, the system prints a title and prompts the user to enter his user I.D. number. After the user enters it, it is checked against the user I.D. file of the system and the password is retrieved. The system then prompts the user for the password associated with the user I.D. The user is given three attempts to respond with the correct password, and if he fails to do so he is logged off the system.

*A. examples related to this Section are shown in Section A of the Illustrative Example, Appendix III.

The System's Programmer in charge of maintaining the system is the person responsible for issuing user I.D. numbers and passwords, and has the ability to change passwords for a user when the user's current password has become unsecure.

If the HELP facilities are invoked during the process of logging on the the system, the user is limited by the security procedures to Section 2 of the HELP file.

After the user has correctly specified the user I.D. number and password, he is asked by the package about what kind of a computer terminal he is using. The CAAS system supports several kinds of terminals, and a listing of the major terminals that are supported is displayed to the user. CAAS uses the terminal type identifier in several of it's functions, including screen clearing and the production of plots after a simulation is run.

If the user invokes the second-level HELP tutorial, a table of the terminals directly supported by CAAS and some terminals that are equivalent is displayed. This can assist users whose terminals are not on the first list that is displayed by the system.

The CAAS system has several different but related primary functions, such as system simulation, system optimization, etc.

In this version (2), the only function that is operating is that of system simulation. The user builds a model in the computer's memory of the system that he wishes to simulate, using standard hydraulic components. After the system is completely defined, it is simulated and results are produced in either tables or plots. The user can modify it and run the simulation again, or simply quit after he is finished.

After choosing to do a system simulation, the user is prompted to create a new model, then, the system immediately begins to prompt the user for components to be used in simulating the system.

For more information see:

Selecting components to be used in the model rerunning or modifying a model HELP tutorial (on-line).

2.3. Selecting Components to be Used in the Model (B)

The CAAS system has an extensive inventory of fluid power components available to the user. There are nine different classifications of components, and many different varieties of each type of component.

The component selection primary menu allows the user several options in selecting components. If the user knows exactly what the

I.D. number of the component that he wants is, he may enter it directly. It is checked for accuracy, and it's description is printed on the screen so that the user can verify his choice. The component selection primary menu is then displayed again.

If the user does not know the I.D. number of the component that he wants, a menu of the main types of components available is displayed. This is called component define mode. While in this mode, several new commands are available to make component selection easier. Table A-1 shows these commands and their effects. By selecting a menu item, the user defines the component more precisely, until all of the components functions are known. By using the define mode commands and the menu data, the user is given a very easy method of selecting the components needed for his model.

If a user selects a component that he later decides is not needed, he can select the option to delete the component. The delete procedure displays a list of the components selected up to that point, and asks the user for which component to delete. This is done until the user decides that he has deleted everything that he needs to, and then he is returned to the component selection primary menu.

For more information see:

CAAS Global Functions - Listing a Component's Description HELP Tutorial (on-line).

Table A.1 - Component Definition Mode Commands

Command	Function
B or BACK	Moves to the previous menu, allowing the user to alter component descriptions previously defined.
E or END	Aborts the entire component definition that has been entered, and returns the user to the component selection primary menu.
H or HELP	On-line CAAS tutorial.

2.4. Defining the Nature of the Simulation to be Performed (C)

The CAAS system can run a simulation in two modes, dynamic (transient-state) or static (steady-state). The nature of the simulation is controlled by separate menus for the system and for each individual component.

The dynamic system simulation used time-varying values for a time-based analysis of the model. The user must specify starting (beginning) and final (ending) times, the time increment, and an increment for which system sampling is to be done to determine output variables.

The starting time is the time at which system monitoring begins. If this time is 0.0, then the monitoring begins at system simulated startup. The final time is the time when system monitoring ends. The step size is the time incremented of the system. The simulated real-time clock is incremented by this amount. The output step size is the interval between output variable measurements. It must be greater than or equal to the step size. Another way to think of this is: simulate the system from T (start) seconds to T (end) seconds by T (incr.) seconds, looking at output variables every T (output) seconds.

In dynamic simulation, the user also must specify a numerical integration method to use during the analysis. Two different methods of numerical integration are available to users: the Euler's method and the Runge-Kutta 4th-Order approximation method.

Two forms of static simulation of system are available to the user: the operating point static simulation and the performance curve simulation.

The operating point method of static simulation uses the input values of the components to determine the final equilibrium conditions for every other component in the system.

The performance curve method is used to evaluate system static performance over the components operating range. Normally, it requires a various source input to investigate the related changes of components. For example, if it is intended to investigate the pressure override characteristic of a relief valve, then a varying source flow rate input (say it ranges from 0 to 50 cubic inches per second, and there are 20 static operating points of interest) is required. Usually, this varying flow rate can be implemented by using a ramp type signal input (component I.D. number 91212).

The nature of the simulation for individual components is set by the user when he specifies input data for the component. The user may select the emperical, static, or dynamic component model to meet the specific simulation problem. Note that if you are in the system dynamic simulation mode, you may use the emperical, static or dynamic component model to simulate the system; however, at least one of the component models must be a dynamic model. If system static simulation is selected, it is not recommended to use dynamic component models.

The current version of CAAS program (CAAS version 2) allows the user to investigate both the static and dynamic performance of any hydraulic component which has a mathematical model developed. It also allows the user to simulate the dynamic performance of a hydraulic system. If the static simulation of a complete hydraulic system is desired, it may be obtained by doing the dynamic simulation and observing its steady-state performance.

For more information see:

General Consideration of the CAAS program

Setting Component properties.

HELP tutorial (on-line).

Final Report of Phase II (discussion).

2.5. Setting the System Fluid Properties (D)

The CAAS system allows the user to alter the value of the various properties of the fluid used by the model. This lets the user use special fluids in his model like those that would be used in an actual working hydraulic system.

CAAS lets the user specify the three most important property descriptors of a fluid: the bulk modulus, density, and viscosity of the working fluid.

The CAAS system supplies default values for the working fluid, and these values are representative of the hydraulic fluid typically used in hydraulic systems. The default values of the working fluid that are used by the CAAS system are listed in Table A-2.

If the user elects to alter one of the values of the working fluid, the system requests the number of the property that he wants to change (the user selects the number from the menu), and then asks the user for the numerical value of the property. After the user enters the number, the system replaces the previous value and then redisplays the proper+y menu so that the user can alter other values if he wants to.

Option #4 (in the rerun mode only) on the property menu allows the user to reset all of the fluid properties to their default values at the same time. After resetting the properties, the property menu is re-displayed with the default values for the user to inspect.

Option #5 (in the rerun mode only) on the property menu allows the user to return to the main program menu.

For more information see:

HELP tutorial (on-line HELP).

TABLE A-2: THE DEFAULT VALUES OF THE WORKING FLUIDS

Fluids Type	Reference Fluids	Bulk Modulus (PSI)	Density ($lbf\text{-}sec^2/in^4$)	Viscosity (lbf-sec/in.)
Petroleum Base	MIL-H-5606	150,000	7.80×10^{-5}	2.0×10^{-6}
Water Glycol	HOUGHTU-SAFE (620)	259,100	9.57×10^{-5}	7.0×10^{-6}
Water/Oil Emulsion	STAYSOL-FR	290,000	8.30×10^{-5}	12.0×10^{-6}
Oil/Water Emulsion	HYDROLUBRIC 120-B	310,000	9.40×10^{-5}	0.07×10^{-6}
Phosphate Ester	SKYDROL 500-A	308,000	9.70×10^{-5}	1.90×10^{-6}

2.6. Setting Component Properties (E)

The CAAS program requires the user to input the component parametric data before it can do the simulation. In addition, the CAAS also requires the user to set the simulation mode of the individual component. There are three component simulation modes available to the user: the emperical, static, or dynamic modes.

The emperical modes uses emperical data (for example measured in a test rig or obtained from the manufacturers) to determine an approximate performance curve for the component, which is then evaluated in the simulation. The user may conduct the HELP to use the curve fitting method to correlate the performance curve to emperical data at this stage if necessary.

If either the static or dynamic component mode is selected, CAAS will prompt the user information and request that he input the required data for that individual component. Normally, the parameter data are related to the design specifications, for instance, the diameter of flow discharge port or the preset system cracking pressure. The explanation of the design terms is available in the Glossary Section of the HELP. The specific term will be displayed by simply pressing H for help when the explanation is necessary.

The CAAS allows the user to alter the input data, either due to an inproper input or when a new value is preferred. The change of input data can be done after all the data for that specific component have been entered or during the rerun model. The user will be informed whenever the data change function is available during the process.

For more information see:

General Consideration of the CAAS System.

Defining the Nature of the Simulation to be Performed.

Rerunning or Modifying a Model.

HELP tutorial (on-line).

2.7. Defining Component Interconnections (F)

The CAAS system cannot simulate a series of components that have no relationship to each other. Some method of defining the various interconnections between the components must therefore be used.

In the CAAS system, each component has a definite number of ports. Each port provides a passageway for the various fluid logic control signals and power transmitters through the component, where the signal is modified or acted upon in some fashion.

Each port has a transmission line connected to it. This transmission line connects the port to other ports that are in other components. This port number/transmission line interconnection must be numbered and used to define the model's interconnections.

When a new model is being created, the user will be immediately prompted, component by component, and port by port for each component, for the number of the transmission line connected to the port. After each port for each component has been defined, a table will be displayed which graphically shows all port/transmission line connections. The user is invited to examine the table to verify that the connections are correct. He is given an opportunity to correct any connections that are incorrect. If the connections are all correct, the system will procede to the next task.

If the connections are not correct, the user can enter the number of the port and component and then the new line number. The table is displayed again and the user can examine it and repeat this process until a correct configuration is obtained.

It is recommended that the user have a sketch of the circuit ready before starting to enter interconnection data. In larger circuits, the line numbers can grow quickly (there are about three lines required for each component), and the user can easily lose track of what is supposed to be connected where.

When assembling a circuit, care must be taken to ensure that the output of one energy port is the same as input for the port at which it is connected. This means that only an arrow to dot power bond configuration is permissible. (See Figure A.1 and Chapter 4, the Interim report of Phase I).

Furthermore, when constructing the circuit to be modelled, it is important to enter each component in the proper sequential order. Because the power-flow modelling technique is based on the concept of power transmission, it is highly recommended to arrange the component sequence to coincide with the power transmission direction in the actual physical system. Normally, the priority for entering the component to be modelled, from first to last, is as follows:

1. Signal Control or boundary elements
2. Power elements
3. Hydraulic control elements
4. Actuators

Once the circuit has been constructed, the lines connecting the energy ports must be numbered. In Appendix III is an example sketch of a typical system that can be modelled using the CAAS system. Note the numbers associated with each line. The numbers are arbitrary and chosen by the user. They match those in the table below the diagram (which is exactly what the CAAS system prints for the user to examine and verify his model's structure).

For more information see:

Component's model Information Data Sheet (Maintainer's Manual)

HELP tutorial (on-line.

2.8. Selecting and Examining Output Results (G)

The user must define to the CAAS system what simulation results to output. This can be tables or plots of component input or responses for any component in the system.

All output from the simulation is generated as a function of two variables. The two variables are chosen by the user using a set of menus.

POWER BOND COMBINATION	FIRST STAGE OUTPUT	SECOND STAGE OUTPUT	CORRECT COMBINATION	AUXILIARY TRANSFORMER	REMARK
o— o—	PRESSURE	PRESSURE	YES	—	—
—o o—	FLOW RATE	PRESSURE	NO	CAPACITANCE	—o —o [C] o— o—
o— —o	PRESSURE	FLOW RATE	NO	RESISTANCE	o— o— [R] —o —o
o+ o—	FORCE	PRESSURE	NO	INVERSE AREA	o+ o+ [A] o— o—
o— o+	PRESSURE	FORCE	NO	AREA	o— o— [A] o+ o+
o+ o#	FORCE	TORQUE	NO	REACTION LENGTH	o+ o+ [L] o# o#
o# o+	TORQUE	FORCE	NO	INV REAC LENGTH	o# o# [C] o+ o+

LEGEND.

Q P
o—

V F
o+

W T
o#

FIGURE 4.1. POWER BONDS COMBINATION CRITERION

The primary menu begins the process. The user can define the type of output generated, and define or delete plot requests. Two types of output are possible. Only one type can be used for any particular simulation. If tables are selected, a table of data points will be printed. If plots are selected, an x-y plot will be generated along with an exact table of data points.

A plot request is essentially a single plot of a set of data points. "Plot Request" refers to any type of output, whether the output is a plot or a table.

All plot requests have to have x and y axis labels. These labels are either a measurable variable at a component port, or time. It is often desirable to plot some output variable as a function of time in the dynamic simulation (as a measure of response for example). It is also possible to plot some output variable as a function of another variable (such as valve output pressure versus pump input flow rate).

Each axis has the same label requirements that must be entered. If the desired variable is time, no other data needs to be entered. If a measurable variable is to be plotted, the user needs to specify a component number, a specific port on the component, and the variable to be measured.

If the user has defined a plot request that he later decides is not needed, it can be deleted. This can be very useful when CAAS is in the rerun mode and the user has decided that certain plots are no longer needed.

For more information see:

Logging onto the CAAS system.

Rerunning or Modifying A Model.

HELP tutorial (on-line).

2.9. Rerunning or Modifying a Model (I)

After the user successfully simulates his model, the system displays a message indicating that the simulation is complete. A menu is displayed which gives the user the opportunity to rerun the simulation after altering some aspect of the model or to stop the simulation.

The rerun primary menu gives the user the option of altering any of the properties, components, or connections that he has already specified. A menu of every one of the CAAS system's main functions is displayed, and the user is asked to select by number which of those functions he wishes to perform. After he makes a valid selection, the system displays the appropriate menu. The user will note that the menus displayed and functions performed are the same ones that he used while in the process of building the model in the first place. The only difference in this case is that he is able to perform these functions at random instead of in an ordered sequence.

The user must be watchful that he does not cause the basic nature of the model to be changed to the point where it will be impossible to successfully simulate. The system provides very little error checking on a model, so it is possible that the user could enter a quantity that could cause a catastrophic error that results in the loss of the model under construction.

As long as the "changes" made to a model do not result in the rendering of the model as unrecognizable as the original model, the user may be reasonably assured that the simulation will be performed as expected.

For more information see:

Selecting Components to be used in the Model

Defining Component Interconnections

Specifying System Fluid Properties

Defining the Type of Simulation to be Performed

Specifying Individual Component Data

Selecting Output Formats

HELP tutorial (on-line).

2.10. On-line User Assistance

The CAAS system provides three different on-line help facilities to assist the user in successfully performing the simulation of a model that he inputs. There are two levels of written clarification and the third on-line aid is a series of subsystems that do calculations for the user.

The first level of tutorial is invoked when the user enters H or HELP in response to a prompt for information by the system. It is a set of short one- or two-line sentences that try to help the user decide what the system needs from him. After the short sentences are displayed, the system waits for the user to respond.

If the user responds with a second H or HELP, the second-level tutorial is invoked. This is a series of full-page explanations that detail exactly what the system is asking for. The second-level tutorial is like a version of this user's manual that is available to the user any time that he is logged on to the system. The second-level tutorial has it's own set of commands to supplement to set of standard CAAS commands. The commands and their functions are given in Table A.4.

When the user first enters the second-level tutorial, he is at the area of the tutorial that deals specifically with the information that the CAAS system requires him to enter. He can, however, move around in the tutorial and review any of the other topics covered. The tutorial has a Table of Contents which lists all of the major sections of the tutorial. The user can go from any panel in the tutorial directly to the Table of Contents by entering T or TOC in response to the command prompt. If H or HELP is entered as a command while in the second-level tutorial, the system displays a section of the tutorial which deals with how to use the tutorial commands (Section 13 in the Table of Contents).

Some parts of the tutorial have menus of subsections of the tutorial. The user can jump directly to one of these subsections by entering the menu number of the desired subsection.

The last part of the on-line help system deals with assisting the user in the determination of various coefficients and constants. There is no second-level help directly accessable from these questions. Instead, when a user enters a second H or HELP response for one of the subjects that falls into this category, he is routed directly to the subsystem that takes care of making calculations for the user. The user is led through any of the calculation subsystems and then is returned to the main CAAS system where he can enter the value that he has just calculated or he can go back and make another one if he doesn't like the one that he just made.

After the user exits the second-level tutorial, the screen is cleared and the question is reprinted so that the user can remember what the question was that caused him to go into the tutorial.

TABLE A.4. Summary of Tutorial Commands - Level 2

Command	Action
B or BACK	Displays the page of tutorial that was previously displayed. If the user is in the Tutorial Table of Contents, he is returned to the question previously asked.
N or NEXT	Displays the next page of tutorial in the series being displayed. If the user is at the last page of the series, he is returned to the question previously asked.
T or TOC	Displays the Tutorial Table of Contents.
E or END	Returns directly to the question asked in the simulation.
H or HELP	Displays page 1 of Section 13 of the tutorial (How to Use the Tutorial).
Q or QUIT	Exit the CAAS system immediately. The model and data being worked on will not be saved.
A number	If the user is in a section of the tutorial that has a sub-menu, a number command will select the tutorial subsection.

CHAPTER III

PREPARATION OF THE INPUT DATA

The purpose of this chapter is to demonstrate the CAAS procedure described in the previous chapter with an illustrative example. As noted, the CAAS system can not simulate a system without a description of the relationship of each component used. In addition, the CAAS requires the user to provide component parametric data before it can do any simulation.

In practice, the CAAS system allows the user to select the components to construct the proposed circuit. It then prompts the users for the required input information. This usually consists of the nature of simulation, the working fluid properties, the component design data, the relationship of power port connection, and the output interpretation. In order to assist the user in preparing the required information before he actually uses the CAAS system, a CAAS simulation work sheet is prepared. A copy of the work sheet is included in Appendix I of this Manual. The procedure (shown in the Work Sheet) is described as follows:

Step 1: Draw the system hardware circuit. Normally, the circuit is represented by the ISO or ANSI graphic symbols to signify the relationship between components.

Step 2: Draw the related power flow symbol for each individuals components used in the hardware circuit. The component power-flow symbols are shown in the CAAS Component Catalog, Appendix II of this Manual.

Step 3: Initially construct the power-flow circuit based on Steps 1 and 2.

Step 4: Check the power flow connection consistency between each connection port. Only the Effort-Flow connection pair is allowed. If inconsistency occurs, use the capacitive line (I.D. 711) or resistive line (I.D. 712) to correct it.

Step 5: Complete the power-flow circuit to represent the system according to the information of Steps 2, 3, and 4.

Step 6: Assign component sequence number. It should be noted that when constructing the circuit to be modeled, it is important to enter each component in the proper sequence. Because the power-flow modeling technique is based on the concept of power transmission, it highly recommended that the component sequence be arranged to coincide with the power transmission direction as in the actual physical system. Normally, the priority for entering the component to be modelled from first to last are as follows:

1. Signal control or boundary element
2. Power elements
3. Hydraulic control elements
4. Actuators.

Step 7: Label the connection lines sequence number. The order in which the lines are labeled is optional.

Step 8: Complete the power flow circuit data sheet according to the information of Steps 5, 6, and 7.

Step 9: Determine the working fluid properties. It specifies the three most important property descriptors of a fluid; the bulk modulus, density, and viscosity.

Step 10: Define the nature of the simulation to be performed. This includes the static and the dynamic simulation.

Step 11: Determine output format and simulation results. This requires the user to select the output format, either Table or Plot, and to specify the simulation results to be observed.

Step 12: Obtain the parametric data for the components. These data are usually obtained from component technical data sheet or from a direct measurement of component physical quantities. The required input information for each component is shown in the component model data base subprograms which usually have a format of DBXXXX. The corresponding subprogram to each component is shown in Appendix II, The CAAS Component Catalog.

Upon completing the above 12 Steps, the user is ready to use the CAAS system to perform simulation. In order to manifest the function of the CAAS package and to illustrate the procedures described in this manual, the simulation of a simple hydraulic system was chosen for the demonstration. The circuit selected for this example consists of a fixed displacement pump, a constant rotation speed prime mover, a direct acting pressure relief valve, and a tank. The function of the circuit is to investigate the performance of a relief valve. The hardware set-up and its related power-flow circuit are shown in the work sheet of Appendix III.

The input data of components were from a direct measurement of component physical quantities. The flow out of the pump (source flow rate) is 20 cubic in/sec (5.2 GPM). The set cracking pressure of the relief valve is 800 PSI. It is assumed that the pump provided a "step" input of flow rate from 0 to 20 cubic in/sec on the relief valve to achieve the required dynamic simulation at the starting point. This condition also can be achieved by using a solenoid valve along with the relief valve. By turning on and off of the valve, it provides the required step signal. The input data and the simulation result are shown in Section H, Appendix III.

APPENDIX I

THE CAAS SIMULATION WORK SHEET

THE CAAS SIMULATION WORK SHEET

Prepared by: ________________ Date ___/___/___

STEP	DESCRIPTION
1	System Hardware Circuit
2	Components Power Flow Symbol

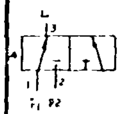
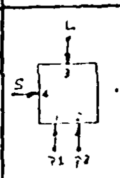
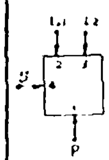
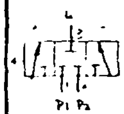
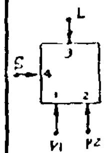
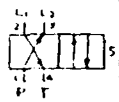
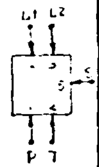
STEP	DESCRIPTION
3 4 5 6 7	Check Power-Flow Consistance of Circuit (only effort-flow pair allowed). Construct the Power-Flow Circuit. Assign Components Sequence Number (signal-power-control-actuator). Label Connection Lines Sequence Number (no sequential priority).
8	Complete the Power Flow Circuit Data Sheet

COMPONENT SEQ. NO.	COMPONENT I.D. NO.	NUMBER OF PORTS	COMPONENT PORT NO. TO CONNECTOR LINE NO. 1	2	3	4	5	6	7

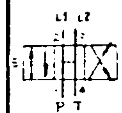
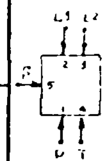
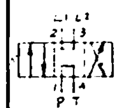
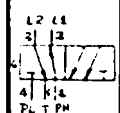
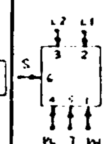
STEP	DESCRIPTION
9	Determine the Nature of System Simulation. Static ___ 1. Operating Point ___ 2. Performance Curve ___ (required at least one various performance as input) Dynamic ___ 1. Simulation Starting Time ________ Sec. 2. Simulatio Step Size ________ Sec. 3. Desired Output Step Size ________ Sec. 4. Simulation Final Time ________ Sec. 5. Integration Method: 1. Euler's Method ____ 2. Runge-Kutta Method ______ 3. Others ____________________
10	Determine Working Fluid Properties. Use Default Values ____ Use User's Input Values ____ 1. Fluid Bulk Modulus __________ psi 2. Fluid Density __________ lbf-sec**2/in**4 3. Absolute Viscosity _______ lbf-sec/in**2
11	Determine Output Format and Simulation Results. Format: Table ____ (defaulted) Plot ____ Output Parameters: PLOT NO. X-AXIS Y-AXIS
12	Obtain Components Design Data.

APPENDIX II

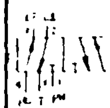
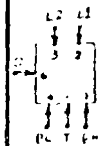
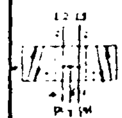
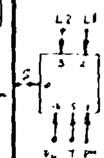
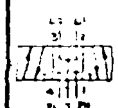
THE CAAS COMPONENT CATALOG

I.D. NO.	DESCRIPTION	A.N.S.I.	CAAS	SUBPROGRAM
131411	4. Selector 1. Mechanical 1. Spring			DCV3WM DCV3WS DEDCV3
131412	2. Detent			
131421	2. Pilot 1. Spring			
131422	2. Detent			
131431	3. Solenoid 1. Spring			
131432	2. Detent			
132111	2. 3-Position 1. Diverter 1. Mechanical 1. Spring			
132112	2. Detent			
132121	2. Pilot 1. Spring			
132122	2. Detent			
132131	3. Solenoid 1. Spring			
132132	2. Detent			
132211	2. Selector 1. Mechancal 1. Spring			
132212	2. Detent			
132221	2. Pilot 1. Spring			
132222	2. Detent			
132231	3. Solenoid 1. Spring			
132232	2. Detent			
141111	4. 4-Way 1. 2-Position 1. Open Center Cross Over 1. Mechnical 1. Spring			DC4W2M DC4W2S DEDCV4
141112	2. Detent			
141121	2. Pilot 1. Spring			
141122	2. Detent			
141131	3. Solenoid 1. Spring			
141132	2. Detent			
141211	2. Closed Center Cross Over 1. Mechanical 1. Spring			
141212	2. Detent			
141221	2. Pilot 1. Spring			
141222	2. Detent			
141231	3. Solenoid 1. Spring			
141232	2. Detent			

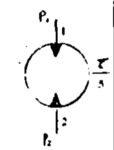
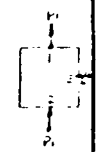
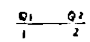
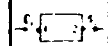
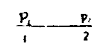
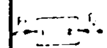
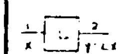
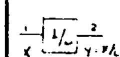
THE CAAS COMPONENT CATALOG

I.D. NO.	DESCRIPTION	A.N.S.I.	CAAS	SUBPROGRAM
142111	2. 3-Position 1. Open Center 1. Mechnical 1. Spring			DC40CM DC40CS DBD40C
142112	2. Detent			
142121	2. Pilot 1. Spring			
142122	2. Detent			
142131	3. Solenoid 1. Spring			
142132	2. Detent			
142211	2. Closed Center 1. Mechnical 1. Spring			DC4CCM DC4CCS DBD4CC
142212	2. Detent			
142221	2. Pilot 1. Spring			
142222	2. Detent			
142231	3. Solenoid 1. Spring			
142232	2. Detent			
142311	3. Tandem Center 1. Mechanical 1. Spring			DC4TCM DC4TCS DBD4TC
142312	2. Detent			
142321	2. Pilot 1. Spring			
142322	2. Detent			
142331	3. Solenoid 1. Spring			
142332	2. Detent			
142411	4. Float Center 1. Mechanical 1. Spring			DC4FCM DC4FCS DBD4FC
142412	2. Detent			
142421	2. Pilot 1. Spring			
142422	2. Detent			
142431	3. Solenoid 1. Spring			
142432	2. Detent			
151111	5. 5-Way 1. 2-Position 1 High Pressure Normally Open 1. Mechnical 1. Spring			DC5HOM DC5HOS DBD5HO
151112	2. Detent			
151121	2. Pilot 1. Spring			
151122	2. Detent			
151131	3. Solenoid 1. Spring			
151132	2. Detent			

THE CAAS COMPONENT CATALOG

I.D. NO.	DESCRIPTION	A.N.S.I.	CAAS	SUBPROGRAM
151211	2. Low Pressure Normally Open 1. Mechnical 1. Spring			DC5LOM DC5LOS DBD5LO
151212	2. Detent			
151221	2. Pilot 1. Spring			
151222	2. Detent			
151231	3. Solenoid 1. Spring			
151232	2. Detent			
152111	2. 3-Position 1. Open Center 1. Mechanical 1. Spring			DC530M DC530S DBD530
152112	2. Detent			
152121	2. Pilot 1. Spring			
152122	2. Detent			
152131	3. Solenoid 1. Spring			
152132	2. Detent			
152211	2. Closed Center 1. Mechancial 1. Spring			DC53CM DC53CS DBD53C
152212	2. Detent			
152221	2. Pilot 1. Spring			
152222	2. Detent			
152231	3. Solenoid 1. Spring			
152232	2. Detent			
152311	3. Tandem Center 1. Mechanical 1. Spring			DC53TM DC53TS DBD53T
152312	2. Detent			
152321	2. Pilot 1. Spring			
152322	2. Detent			
152331	3. Solenoid 1. Spring			
152332	2. Detent			
152411	4. Float Center 1. Mechanical 1. Spring			DC53FM DC53FS DBD53F
152412	2. Detent			
152421	2. Pilot 1. Spring			
152422	2. Detent			
152431	3. Solenoid 1. Spring			
152432	2. Detent			

THE CAAS COMPONENT CATALOG

I.D. NO.	DESCRIPTION	A.N.S.I.	CAAS	SUBPROGRAM
621	2. Rotary Type 1. Single Vane			CYLDRM CYLDRS DBCYLD
622	2. Double Vane			
711	7. Conductors and Conditioners 1. Line 1. Capacitive Type			CAPLNM CAPLNS DBCAPL
712	2. Resistive Type			RESLNM RESLNS DBRESL
72xx	2. Fittings (N/A as of March 1983)			
73XX	3. Filters (N/A as of March 1983)			
74xx	4. Heat Exchanger (N/A as of March 1983)			
7511	5. Transformer 1. Area 1. Regular			
7512	2. Inversed			
7521	2. Length 1. Regular			
7522	2. Inversed			
81xx	8. Special Components 1. Accumulators (N/A as of March 1983)			
82xx	2. Servo Components (N/A as of March 1983)			

APPENDIX III

THE ILLUSTRATIVE EXAMPLE

NOTE: Upper Case character : Computer Message
Lower Case character : User's Input

A

```
COMPUTER AIDED FLUID POWER SYSTEM DESIGN PACKAGE
              THE CAAS PROGRAM

ENTER YOUR USER I.D. NUMBER -
1192

ENTER CURRENT PASSWORD FOR USER I.D. 1192-
fffc

INVALID PASSWORD.

ENTER CURRENT PASSWORD FOR USER I.D. 1192-
bill
```

Logging onto the CAAS system

```
PLEASE ENTER THE TERMINAL TYPE YOU ARE USING:
(1) IBM 3270
(2) DATAGRAPHIX
(3) TEKTRONIX
(4) OTHERS
OPTIONS?
h                                                  <- Enter "h" for the 1st level on-line HELP
 ENTER THE NUMBER OF THE TERMINAL THAT YOU ARE USING.   <- Message from the 1st level on-line HELP
h                                                  <- Enter "h" again for the 2nd level on-line HELP
          TERMINALS SUPPORTED BY THE CAAS SYSTEM   <- Message from the 2nd level on-line HELP

       THE CAAS SYSTEM CAN BE USED ON ALMOST ANY DATA
    TERMINAL AVAILABLE.  IN THE DEFAULT MODE, THE SYSTEM
    OPERATES ON THE ASSUMPTION THAT THE TERMINAL BEING USED
    IS A 'LINE-MODE' TERMINAL.  AN EXAMPLE OF A LINE-MODE
    TERMINAL IS A DIGITAL EQUIPMENT CORPORATION DECWRITER
    TERMINAL OR AN ADDS ADM-3 TERMINAL.

       HOWEVER, THERE ARE SOME TERMINALS THAT ALLOW THE CAAS
    SYSTEM TO USE THE TERMINAL-DEPENDENT FEATURES AVAILABLE, SUCH
    AS SCREEN-CLEARING FUNCTIONS, OR IN THE CASE OF THE TEKTRONIX
    4000-SERIES OF GRAPHICS TERMINALS, VERY HIGH-QUALITY OUTPUT
    PLOTS.

       A LIST OF THE TERMINALS CURRENTLY SUPPORTED BY THE CAAS
    SYSTEM IS GIVEN ON THE NEXT PAGE.
COMMAND:
n                                                  <- "n" to turn the "NEXT" page on HELP
          SPECIAL TERMINALS SUPPORTED BY THE CAAS SYSTEM   <- Message on the "NEXT" page

          TERMINAL                        SPECIAL FEATURES

       IBM 3270-TYPE                    SCREEN-CLEAR, GRAPHICS
       DATAGRAPHIX 0410                 SCREEN CLEAR
       TEKTRONIX 4000-SERIES            SCREEN-CLEAR, HIGH-
                                           RESOLUTION GRAPHICS


       ADDITIONS TO THIS LIST WILL BE MADE WITH EACH RELEASE
    OF THE CAAS PROGRAM.
```

```
COMMAND:
e
```

— "e" to end the HELP and return to the main manual

```
PLEASE ENTER THE TERMINAL TYPE YOU ARE USING:
(1) IBM 3270
(2) DATAGRAPHIX
(3) TEKTRONIX
(4) OTHERS
OPTIONS?
4
 THE FUNCTIONS OF THE CAAS PROGRAM ARE:

1 - SYSTEM SIMULATION
2 - SYSTEM OPTIMIZATION
3 - SERVICE LIFE PREDICTION
OPTION?
1
CAAS SYSTEM SIMULATION PACKAGE


ENTER A DESCRIPTIVE TITLE FOR THIS JOB -
(THE TITLE MUST BE 79 CHARACTERS OR LESS).
this is an  illustrative example of the caas package.
```

A

```
SPECIFY COMPONENTS TO BE USED IN THE MODEL.....
```

B

```
1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
h
 SELECT ONE OF THE OPTIONS ABOVE.
h
                 ENTERING A KNOWN COMPONENT NUMBER

        IF THE USER KNOWS A COMPONENT NUMBER THAT HE WANTS TO
    USE, HE CAN ENTER IT DIRECTLY BY SELECTING 1 ON THE COM-
    PONENT SELECTION MENU.  HE IS PROMPTED FOR THE NUMBER, AND
    AFTER IT IS ENTERED IT IS CHECK FOR VALIDITY.  IF IT IS A
    VALID COMPONENT NUMBER, IT IS PLACED INTO THE MODEL AND THE
    PROGRAM RETURNS TO THE COMPONENT SELECTION MENU.
COMMAND:
n
```

Selecting Components to be used in the model

```
        SELECTING A COMPONENT - UNKNOWN COMPONENT NUMBER

        IF THE USER HAS AN IDEA OF THE NAME OF HIS COMPONENT
    OR EVEN A GENERAL GUESS OF ITS FUNCTION, IT IS AN EASY
    MATTER TO USE THE CAAS SYSTEM TO ASSIGN A COMPONENT
    NUMBER TO IT.

        AFTER THE USER SELECTS OPTION 2 ON THE COMPONENT
    SELECTION MENU, A LIST OF THE MAJOR TYPES OF COMPONENTS
    IN THE DATABASE IS DISPLAYED.  IF, FOR EXAMPLE, THE USER IS
    LOOKING FOR A SPECIFIC TYPE OF PUMP, HE SELECTS OPTION 4
    (PUMPS) FROM THE MENU.  A LIST OF DIFFERENT TYPES OF PUMPS
    IS DISPLAYED NEXT, AND THE USER SELECTS ONE OF THESE PUMP
```

```
CHARACTERISTICS.  THIS NARROWING DOWN PROCESS CONTINUES UNTIL
EVERY CHARACTERISTIC OF THE PUMP IS DEFINED, AT WHICH POINT IT
IS ENTERED INTO THE USERS MODEL.

   A SERIES OF SPECIAL COMMANDS CAN BE ISSUED IN THIS MODE.
A LISTING OF THESE COMMANDS ARE ON THE NEXT PAGE.
COMMAND:
e


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3   YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
1


ENTER THE COMPONENT NUMBER -
91112
COMPONENTS SELECTED SO FAR:

     91112


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
2

SELECTING A COMPONENT FROM THE DATABASE
SELECT ONE OF THE FOLLOWING-
 1     DIRECTIONAL CONTROL VALVE
 2     FLOW CONTROL VALVE
 3     PRESSURE CONTROL VALVE
 4     PUMP
 5     MOTOR
 6     CYLINDER
 7     CONDUCTOR OR CONDITIONER OR TANK
 8     SPECIAL COMPONENT
 9     SIGNAL CONTROLLER

OPTION?
4

SELECTING A COMPONENT FROM THE DATABASE
SELECT ONE OF THE FOLLOWING-
 1     GEAR
 2     VANE
 3     AXIAL PISTON
 4     RADIAL PISTON

OPTION?
3

SELECTING A COMPONENT FROM THE DATABASE
SELECT ONE OF THE FOLLOWING-
 1     FIXED DISPLACMENT
 2     VARIABLE DISPLACEMENT
```

B

← Assume you know the I.D No for the Component. The I.D No. MUST BE Entered According to the Sequence of Component in the Power Flow Circuit!

B

```
OPTION?
1

SELECTING A COMPONENT FROM THE DATABASE
SELECT ONE OF THE FOLLOWING-
 1    SWASH PLATE
 2    BENT AXIS

OPTION?
1
COMPONENTS SELECTED SO FAR:

      91112
       4311


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
1


ENTER THE COMPONENT NUMBER -
31111
COMPONENTS SELECTED SO FAR:

      91112
       4311
      31111


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
1


ENTER THE COMPONENT NUMBER -
76
COMPONENTS SELECTED SO FAR:

      91112
       4311
      31111
         76


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4   YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
1
```

```
ENTER THE COMPONENT NUMBER -
312
   312 IS AN INVALID COMPONENT NUMBER.          <--- Improper I.D. Number
1 - RETURN TO MAIN MENU                         <--- Message from CHAS
2 - TRY ANOTHER COMPONENT
H - HELP
OPTION?
2

ENTER THE COMPONENT NO.-
31112
COMPONENTS SELECTED SO FAR:

     91112
      4311
     31111
        76
     31112


1 - YOU KNOW THE COMPONENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
3 - YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4 - YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?                                         <--- To delete components
3                                                    from selection Table


1 - DELETE A COMPONENT.
2 - GO BACK TO MAIN MENU.
1


ENTER THE NUMBER OF THE COMPONENT THAT
YOU WISH TO DELETE -

 1    91112
 2     4311
 3    31111
 4       76
 5    31112

OPTION?
5

COMPONENT NO.     5 DELETED.


1 - DELETE A COMPONENT.
2 - GO BACK TO MAIN MENU.
2
COMPONENTS SELECTED SO FAR:

     91112
      4311
     31111
        76


1 - YOU KNOW THE COMPUNENT NUMBER.
2 - YOU DON'T KNOW THE COMPONENT NUMBER.
```

B

```
3   YOU WOULD LIKE TO DELETE A CHOSEN COMPONENT.
4   YOU HAVE FINISHED CHOOSING/DELETEING COMPONENTS
    AND YOU WOULD LIKE TO CONTINUE.
OPTION?
4

NEXT IT IS NESSARY TO SET THE VALUE OF FLUID PROPERTIES

ENTER THE TYPE OF WORKING FLUIDS (SET THE DEFAULT VALUES)

   FLUIDS TYPE              REFERENCE FLUIDS
                            ----------------
1. PETROLEUM BASE          MIL-H-5606
2. WATER GLYCOL            HOUGHTO - SAFE 620
3. WATER/OIL EMULSION      STAYSOL - FR
4. OIL/WATER EMULSION      HYDROLUBRIC 120-B
5. PHOSPHATE ESTER         SKYDROL 500A
OPTIONS ?
1
THE VALUES OF FLUID PROPERTIES SET ARE:
1 - FLUID BULK MODULUS       :   150000.0     PSI
2 - FLUID DENSITY            :   .7800000E-04 LBF-SEC**2/IN**4
3 - FLUID ABSOLUTE VICOSITY  :   .2000000E-05 LBF-SEC/IN**2
DO YOU WANT TO CHANGE ANY VALUE OF THESE PARAMETERS ?
ENTER YES OR NO
n
PLEASE SET THE NATURE OF SIMULATION FOR THE SYSTEM:
(1) STATIC SIMULATION
(2) DYNAMIC SIMULATION
CHOOSE ONE OF THE ABOVE :
h
 SELECT THE MANNER IN WHICH YOU WANT TO RUN YOUR SIMULATION.
2
YOU HAVE CHOSEN TO PERFORM A DYNAMIC SIMULA,ION
NOW SET SYSTEM DYNAMIC SIMULATION INFORMATION :


(1) SIMULATION STARTING TIME (SEC):
0.,0

YOUR ANSWER MUST BE A REAL NUMBER

PLEASE REENTER...
FOR MORE INFORMATION, HIT H FOR HELP
0.0

(2) SIMULATION STEP SIZE (SEC):
h
 STEP SIZE DEPENDS ON THE MAXIMUM NATURAL FREQUENCY OF THE SYSTEM.
 IT SHOULD BE ABOUT 1/20 - 1/100 OF THE MAXIMUM FREQUENCY.  IF YOU
 WOULD LIKE TO CALCULATE THE OPTIMAL STEP SIZE, ENTER HELP OR H.
h
THIS PROGRAM HELPS THE USER TO DETERMINE THE
PROPER INTEGRATION STEPSIZE.

ENTER THE MASS OF THE CRITICAL COMPONENT (LBF/IN/SEC**2)-
1.2e-4

ENTER THE SPRING STIFFNESS CONSTANT OF THE
CRITICAL COMPONENT-
500.


THE STEP SIZE MUST BE LESS THAN    .1538279E-03

(2) SIMULATION STEP SIZE (SEC):
```

B / C

Specify Working Fluid Properties

C / D

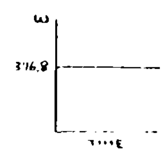
Defining the Nature of Simulation to be Performed

```
1.0e-4

(3) SIMULATION FINAL TIME (SEC):
h
 ENTER A VALUE FOR TIME.
h
                       TIME PARAMETERS

        THE BEGINNING TIME IS THE TIME AT WHICH SYSTEM MON-
    ITORING BEGINS.  IF THIS TIME IS 0.0, THEN THE MONITORING
    BEGINS AT SYSTEM SIMULATED STARTUP.

        THE ENDING SYSTEM TIME IS THE TIME WHEN SYSTEM MONITORING
    ENDS.

        THE STEP SIZE IS THE TIME INCREMENT OF THE SYSTEM.  THE
    SIMULATED REAL-TIME CLOCK IS INCREMENTED BY THIS AMOUNT.

        THE OUTPUT STEP SIZE IS THE INTERVAL BETWEEN OUTPUT
    VARIABLE MEASUREMENTS.  IT MUST BE GREATER THAN OR EQUAL
    TO THE STEP SIZE.

        ANOTHER WAY TO THINK OF THIS IS: SIMULATE MY SYSTEM
    FROM T(START) SECONDS TO T(END) SECONDS BY T(INCR) SECONDS,
    LOOKING AT OUTPUT VARIABLES EVERY T(OUTPUT) SECONDS.

COMMAND:
e

(3) SIMULATION FINAL TIME (SEC):
0.01

(4) DESIRED OUTPUT STEP SIZE (SEC):
2.0e-3


SET INTEGRATION METHOD DESIRED IN SIMULATION PROCESS:
(1) EULER'S METHOD
(2) RUNGE - KUTTA 4TH ORDER METHOD
CHOOSE ONE OF THE ABOVE:
h
 SELECT ONE OF THE INTEGRATION METHODS TO USE WHEN SIMULATING YOUR
 MODEL.
1


NOW THE DESIGN PARAMETERS OF EACH OF THE
COMPONENTS MUST BE SET.


PLEASE ENTER THE FOLLOWING INFORMATION FOR COMPONENT:      91112
(1) STEP SIGNAL STARTING TIME :
0.0
(2) SIGNAL AMPLITUDE BEFORE STARTING TIME :
0.0
(3) SIGNAL AMPLITUDE AFTER STARTING TIME :
376.8
IS THERE ANY CHANGE OF INPUT DATA OF THIS COMPONENT
TO BE MADE?
ENTER YES OR NO
y
```

D

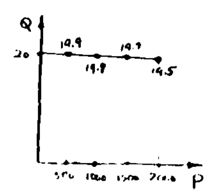
E

Setting Component Properties



Assume the prime mover provides a constant angular velocity (376.8)

```
COMPONENT...     9111
(1) STEP SIGNAL STARTING TIME :
    .0
(2) SIGNAL AMPLITUDE BEFORE STARTING TIME :
    .0
(3) SIGNAL AMPLITUDE AFTER STARTING TIME :
    376.2998

ENTER THE SEQUENCE NUMBER OF THE PARAMETER THAT
NEEDS TO BE CHANGED :
2
ENTER THE NEW VALUE FOR THIS PARAMETER:
376.8


ANY OTHER PARAMETER OF THIS COMPONENT THAT NEEDS
TO BE CHANGED?
ENTER YES OR NO
n


PLEASE ENTER THE FOLLOWING INFORMATION FOR COMPONENT:      4311


SET THE NATURE OF SIMULATION FOR COMPONENT:      4311 USED
    (1) STEADY STATE -- PERFORMANCE DATA KNOWN
    (2) STEADY STATE -- DETAIL DESIGN PARAMETERS KNOWN
    (3) DYNAMIC STATE - DETAIL DESIGN PARAMRTERS KNOWN
CHOOSE ONE OF THE ABOVE:
1

ENTER THE COEFFICIENTS OF PERFORMANCE CURVE
FLOW = K1 + K2*PRESS + K3*PRESS**2 +K3*PRESS**3
WHERE  PRESS : LOAD PRESSURE  (PSI)
       FLOW : PUMP FLOW RATE (IN**3/SEC)
(1) COEFFICIENT K1:
h
 THE PERFORMANCE CURVE IS REPRESENTED BY A PLOYNOMIAL.  THE
 COEFFICIENTS OF THE EQUATION OF THE CURVE ARE NEEDED.  IF YOU NEED
 TO CALCULATE THE COEFFICIENTS, ENTER HELP OR H.
h

THIS PROGRAM COMPUTES THE COEFFICIENTS OF THE M-DEGREE
POLYNOMIAL THAT BEST FITS N-DATA POINTS USING THE LEAST
SQUARES METHOD.

THE PROGRAM WILL COMPUTE COEFFICIENTS FOR EACH DEGREE OF
POLYNOMIAL FROM DEGREE 1 TO DEGREE 3.  YOU NEED TO INPUT AT
LEAST 4 DATA POINTS.

ENTER THE NUMBER OF DATA POINTS TO BE FITTED
5
 DATA POINT  1
 ENTER X VALUE:
0.
 ENTER Y VALUE:
20.
 DATA POINT  2
 ENTER X VALUE:
500.
 ENTER Y VALUE:
19.9
 DATA POINT  3
 ENTER X VALUE:
1000.
```



Pump's characteristic Curve

E

E

```
 ENTER Y VALUE:
19.8
 DATA POINT  4
 ENTER X VALUE:
1500.
 ENTER Y VALUE:
19.6
 DATA POINT  5
 ENTER X VALUE:
2000.
 ENTER Y VALUE:
19.5


POINT       X              Y
  1        .0             20.00
  2       500.0           19.90
  3       1000.           19.80
  4       1500.           19.60
  5       2000.           19.50


DO YOU WISH TO CHANGE A POINT?
(ENTER YES OR NO):
n
COEFFICIENTS ARE LISTED FROM COEFFICIENT OF LOWEST
ORDER TERM TO COEFFICIENT OF HIGHEST ORDER TERM.
FOR DEGREE OF  1 COEFFICIENTS ARE
               20.020     -0.000
          THE VARIANCE IS    0.00100
FOR DEGREE OF  2 COEFFICIENTS ARE
               20.006     -0.000      -0.000
          THE VARIANCE IS    0.00114
FOR DEGREE OF  3 COEFFICIENTS ARE
               19.996     -0.000      -0.000      0.000
          THE VARIANCE IS    0.00129


PLEASE ENTER THE PROPER VALUE OF COEFFICIENT.....
20.020
(2) COEFFICIENT K2:
0.0
(3) COEFFICIENT K3:
0.0
(4) COEFFICIENT K4:
0.0
IS THERE ANY CHANGE OF INPUT DATA OF THIS COMPONENT
TO BE MADE?
ENTER YES OR NO
n


PLEASE ENTER THE FOLLOWING INFORMATION FOR COMPONENT:     31111


SET THE NATURE OF SIMULATION FOR COMPONENT:     31111 USED
    (1) STEADY STATE -- PERFORMANCE DATA KNOWN
    (2) STEADY STATE -- DETAIL DESIGN PARAMETERS KNOWN
    (3) DYNAMIC STATE - DETAIL DESIGN PARAMRTERS KNOWN
CHOOSE ONE OF THE ABOVE:
3
(5) CRACKING PRESSURE (PSI):
h
  CRACKING:  THE PRESSURE AT WHICH A PRESSURE OPERATED VALVE
      BEGINS TO OPEN.
```

← Enter numerical value obtained from the above calculation. The smaller the variance, the better the input data correlate the characteristic curve

← Enter "h" if you need more explanation of the statement.

E

```
800.
(5) UPSTREAM PRESSURE REACTION AREA (IN**2):
0.05
(7) DOWNSTREAM PRESSURE REACTION AREA (IN**2):
0.05
(8) SPRING CONSTANT (LB/IN):
h
  LINEAR SPRING STIFFNESS CONSTANT:  THE RATE OF SPRING FORCE
      AND ITS ASSOCIATED DISPLACEMENT.
500.
(9) FLOW DISCHARGE COEFFICIENT AT PORT 3:
0.61
(10) AREA GRADIENT OF PORT 3 (IN**2/IN):
0.389
(11) FLOW JET ANGLE OF PORT 3 (DEGREE):
20.
(12) LEAKAGE FLOW COEFFICIENT (IN**3/SEC/PSI):
h
 THE LEAKAGE FLOW COEFFICIENT CORRELATES THE FLOW RATE AND DIFFER-
 ENTIAL PRESSURE.  IF YOU NEED TO CALCULATE THIS, ENTER HELP OF H.
h


THIS PROGRAM HELPS USERS TO DETERMINE THE COEFFICIENT OF FLOW LEAKAGE.

ENTER THE PISTON OR SPOOL DIAMETER (IN):
0.5
THE RADIAL CLEARANCE (IN):
1.0e-4
THE EFFECTIVE LENGTH OF PASSAGE (IN):
0.5


THE COEFFICIENT OF LEAKAGE IS :      .26180E-12

PLEASE ENTER THE PROPER VALUE OF LEAKAGE COEFFICIENT.....
0.0
(13) MASS OF POPPET OR SPOOL (LBF-SEC**2/IN):
1.2e-4
(14) FLUID REACTION VOLUME (IN**3):
50.
(15) MAXIUM POPPET OR SPOOL DISPLACEMENT (IN):
0.2
(16) THE DAMPING COEFFICIENT OF THE SPOOL
0.05
(17) UNSTEADY FLOW FORCE COEFFICIENT (LBF-SEC/IN):
0.0
INITIAL CONDITIONS:
(103) SYSTEM PRESSURE (PSI):
0.a

YOUR ANSWER MUST BE A REAL NUMBER

PLEASE REENTER...
FOR MORE INFORMATION, HIT H FOR HELP
250.0
IS THERE ANY CHANGE OF INPUT DATA OF THIS COMPONENT
TO BE MADE?
ENTER YES OR NO
n


PLEASE ENTER THE FOLLOWING INFORMATION FOR COMPONENT:        76
(1) THE TANK PRESSURE (PSI):
```

```
N.O
IS THERE ANY CHANGE OF INPUT DATA OF THIS COMPONENT
TO BE MADE?
ENTER YES OR NO
N
NEXT IT IS NECESSARY TO CONSTRUCT THE POWER FLOW CIRCUIT FOR THE HYDRAULIC SYSTEM
ENTER THE CONNECTION LINE NO. TO THE ASSOCIATED ENERGY PORT

COMPONENT SEQUENCE NO.:   1
COMPONENT I. D. NO.   :     91112
NUMBER OF PORTS       :   1

ENTER THE LINE NO. AT PORT    1
1

COMPONENT SEQUENCE NO.:   2
COMPONENT I. D. NO.   :     4311
NUMBER OF PORTS       :   3

ENTER THE LINE NO. AT PORT    1
1
ENTER THE LINE NO. AT PORT    2

ENTER THE LINE NO. AT PORT    3

COMPONENT SEQUENCE NO.:   3
COMPONENT I. D. NO.   :     31111
NUMBER OF PORTS       :   5

ENTER THE LINE NO. AT PORT    1
H
YOU ARE CONSTRUCTING A POWER-FLOW REPRESENTATION.  FOR MORE
INFORMATION ENTER HELP OR H.
H
              DEFINING A CIRCUIT MODEL TO CAAS

      BEFORE THE CAAS SYSTEM CAN SIMULATE A FLUID POWER
   SYSTEM, A DESCRIPTION OF THE CIRCUIT ITSELF MUST BE GIVEN
   BY THE USER.  THIS CIRCUIT IS TO BE DESCRIBED IN TERMS OF
   LINES AND PORTS ON THE COMPONENTS USED IN THE MODEL.

      1 - CORRECTING A POWER-FLOW (PF) DIAGRAM.
      2 - PORT NUMBERS.
      3 - LINE NUMBERS.
      4 - DISPLAY AN EXAMPLE PF DIAGRAM.

   SELECT ONE OF THE ABOVE FOR MORE INFORMATION.
COMMAND:
1
            CORRECTING THE POWER-FLOW (PF) DIAGRAM

      IF THE USER ENTERS A WRONG LINE NUMBER FOR A COM-
   PONENT'S PORT, HE WILL NOTICE IT ON THE PF DIAGRAM
   THAT THE CAAS SYSTEM DISPLAYS.

      TO CORRECT THE INCORRECT INFORMATION, ENTER THE
   COMPONENT SEQUENCE NUMBER, THEN THE NUMBER OF THE PORT,
   AND FINALLY THE NEW LINE NUMBER (THE CORRECT ONE) THAT
   SHOULD BE ATTACHED TO THE PORT.

      THE CAAS SYSTEM WILL REDISPLAY THE PF DIAGRAM AFTER
   THE CORRECTION SO THAT THE USER CAN VERIFY THAT THE
```

E

F

Defining Components Interconnections

```
    CORRECTION IS THE ONE THAT HE WANTED TO MAKE.

COMMAND:
b
                DEFINING A CIRCUIT MODEL TO CAAS

        BEFORE THE CAAS SYSTEM CAN SIMULATE A FLUID POWER
    SYSTEM, A DESCRIPTION OF THE CIRCUIT ITSELF MUST BE GIVEN
    BY THE USER.  THIS CIRCUIT IS TO BE DESCRIBED IN TERMS OF
    LINES AND PORTS ON THE COMPONENTS USED IN THE MODEL.

         1 - CORRECTING A POWER-FLOW (PF) DIAGRAM.
         2 - PORT NUMBERS.
         3 - LINE NUMBERS.
         4 - DISPLAY AN EXAMPLE PF DIAGRAM.

    SELECT ONE OF THE ABOVE FOR MORE INFORMATION.
COMMAND:
4
        LINE 4      +----------+         LINE 3
        +-----------]1  1111  2]-------------------+
        ]           +----------+                   ]
   +-------+                                       ]
   ]  1   ]    LINE 6                        +---------+
   ]  222 3]---------------+       LINE 2  ]    1    ]
   ]  2   ]                ]         +-----]2  444   ]
   +-------+       +-----------+     ]     ]    3    ]
        ] LINE 5   ]     1     ]     ]     +---------+
        +----------] 4  333   2]-----+           ]
                   ]     3     ]                 ]LINE 1
                   +-----------+                 ]
                          +----------------------+

    COMPONENT   COMPONENT   NUMBER    COMPONENT PORT NO. TO LINE NO.
    SEQ. NO.    I.D. NO.   OF PORTS    1    2    3    4    5    6

        1          333         4       6    2    1    5
        2         1111         2       4    3
        3          444         3       3    2    1
        4          222         3       4    5    6
COMMAND:
e

COMPONENT SEQUENCE NO.:   3
COMPONENT I. D. NO.    :     31111
NUMBER OF PORTS        :   5


ENTER THE LINE NO. AT PORT     1
2
ENTER THE LINE NO. AT PORT     2
3
ENTER THE LINE NO. AT PORT     3
4
ENTER THE LINE NO. AT PORT     4
5
ENTER THE LINE NO. AT PORT     5
6

COMPONENT SEQUENCE NO.:   4
COMPONENT I. D. NO.    :        76
NUMBER OF PORTS        :   4


ENTER THE LINE NO. AT PORT     1
7
```

F

```
ENTER THE LINE NO. AT PORT    2
5
ENTER THE LINE NO. AT PORT    3
4
ENTER THE LINE NO. AT PORT    4
8
                        *  POWER FLOW CIRCUIT TOPOGRAPHY  *
                           ==============================

 COMPONENT   COMPONENT   NUMBER   COMPONENT PORT NO. TO CONNECTOR LINE NO.
 SEQ. NO.    I.D. NO.   OF PORTS    1    2    3    4    5    6    7
 --------    ---------  --------  -------------------------------------
    1          91112       1        1
    2           4311       3        1    2    7
    3          31111       5        2    3    4    5    6
    4             76       4        7    5    4    8

 IS THERE ANY CONNECTION THAT NEED TO BE CHANGED ?
ENTER YES OR NO
n


1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
1


SELECT OUTPUT FORMAT
1 - TABLES
2 - PLOTS
OPTION?
2


1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
2


DEFINE A PLOT OR TABLE

              SELECTED    CURRENTLY
1 - X AXIS      NO            0
2 - Y AXIS      NO            0
3 - END
OPTION?
1


SELECT THE X-AXIS LABEL.

NO.   COMPONENT    PORTS
 0      TIME
 1      91112        1
 2       4311        3
 3      31111        5
 4         76        4

OPTION?
0
```

F

G

Selecting and Examining Output Results

G

```
DEFINE A PLOT OR TABLE

              SELECTED     CURRENTLY
1 - X AXIS      YES          100
2 - Y AXIS      NO             0
3 - END
OPTION?
2


SELECT THE Y-AXIS LABEL.

NO.   COMPONENT    PORTS
 0       TIME
 1      91112        1
 2       4311        3
 3      31111        5
 4         76        4

OPTION?
3


SELECT THE Y-AXIS PORT.

COMPONENT NO. 31111 HAS 5 PORTS.
SELECT THE PORT NUMBER TO MODEL-
1


SELECT THE Y-AXIS VARIABLE.


SELECT THE VARIABLE TO MEASURE FOR COMPONENT
NO. 31111, PORT 1 ON THE Y-AXIS.

1 - PRESSURE
2 - FLOW RATE
3 - FORCE
4 - VELOCITY
5 - TORQUE
6 - ANGULAR VELOCITY

OPTION?
1


DEFINE A PLOT OR TABLE

              SELECTED     CURRENTLY
1 - X AXIS      YES          100
2 - Y AXIS      YES          311
3 - END
OPTION?
3
                SPECIFY DESIRED OUTPUT
                    CURRENT SETUP
             Y-AXIS                                  X-AXIS
NO. COMPONENT  PORT   WHAT            COMPONENT   PORT    WHAT
 1    31111      1   PRESSURE  VS.                TIME

THE OUTPUT FORMAT CHOSEN IS PLOTS
```

```
1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
4

DO YOU WANT THE INPUT DATA TO BE PRINTED OUT ?
ENTER YES OR NO
y

DO YOU WANT A HARDCOPY OF THE RUN?
ENTER YES OR NO.
n

*********************************************************
*                                                       *
*  THE COMPUTER AIDED ANALYSIS AND SIMULATION PROGRAM   *
*       A CAD TOOL FOR HYDRAULIC SYSTEM ANALYSIS        *
*                                                       *
*  CAAS VERSION 1.0                      DECEMBER,1982  *
*********************************************************


                    *************
                    * JOB TITLE *
                    *************

THIS IS AN  ILLUSTRATIVE EXAMPLE OF THE CAAS PACKAGE.

                **********************************
                * SYSTEM SIMULATION PARAMETERS *
                **********************************

* NATURE OF SYSTEM SIMULATION *
  ----------------------------

DYNAMIC SIMULATION

* DYNAMIC SIMULATION INFORMATION *
  -------------------------------

SIMULATION STARTING TIME:      0.0        (SEC)
SIMULATION STEP SIZE    :      0.100E-03  (SEC)
SIMULATION FINAL TIME   :      0.100E-01  (SEC)
DESIRED OUTPUT STEP     :      0.200E-02  (SEC)

* INTEGRATION METHOD USED *
  ------------------------

EULER'S METHOD

* FLUID PROPERTIES USED *
  ----------------------

FLUID BULK MODULUS       :   150000.0      PSI
FLUID DENSITY            :   .7800000E-04 LBF-SEC**2/IN**4
FLUID ABSOLUTE VISCOSITY:    .2000000E-05 LBF-SEC/IN**2
```

G

H

Display Input Data
and
Simulation Results

```
*****************************************
* SYSTEM POWER FLOW CIRCUIT TOPOGRAPHY *
*****************************************

COMPONENT   COMPONENT   NUMBER     COMPONENT PORT NO. TO CONNECTOR LINE NO.
SEQ.  NO.   I.D.  NO.   OF PORTS    1    2    3    4    5    6    7
---------   ---------   --------   -----------------------------------
   1          91112        1        1
   2           4311        3        1    2    7
   3          31111        5        2    3    4    5    6
   4             76        4        7    5    4    8

*************************************
* COMPONENT SIMULATION PARAMETERS *
*************************************

COMPONENT...      91112
(1) STEP SIGNAL STARTING TIME :
   .0
(2) SIGNAL AMPLITUDE BEFORE STARTING TIME :
   376.7998
(3) SIGNAL AMPLITUDE AFTER STARTING TIME :
   376.7998


COMPONENT...       4311

NATURE OF SIMULATION: (1) STEADY STATE -- PERFORMANCE DATA KNOWN

ENTER THE COEFFICIENTS OF PERFORMANCE CURVE
FLOW = K1 + K2*PRESS + K3*PRESS**2 +K3*PRESS**3
WHERE  PRESS : LOAD PRESSURE  (PSI)
       FLOW : PUMP FLOW RATE (IN**3/SEC)
(1) COEFFICIENT K1:
   20.01999
(2) COEFFICIENT K2:
   .0
(3) COEFFICIENT K3:
   .0
(4) COEFFICIENT K4:
   .0


COMPONENT...      31111

NATURE OF SIMULATION: (3) DYNAMIC STATE - DETAIL DESIGNPARAMETERS KNOWN
(5) CRACKING PRESSURE (PSI):
   800.0000
(6) UPSTREAM PRESSURE REACTION AREA (IN**2):
   .5000000E-01
(7) DOWNSTREAM PRESSURE REACTION AREA (IN**2):
   .5000000E-01
(8) SPRING CONSTANT (LBF/IN):
   500.0000
(9) FLOW DISCHARGE COEFFICIENT AT PORT 3:
   .6100000
(10) AREA GRADIENT OF PORT 3 (IN**2/IN):
```

```
   .3899999
(11) FLOW JET ANGLE OF PORT 3 (DEGREE):
   20.00000
(12) LEAKAGE FLOW COEFFICIENT (IN**3/SEC/PSI):
   .0
(13) MASS OF POPPET OR SPOOL (LBF SEC**2/IN):
   .1200000E 03
(14) FLUID REACTION VOLUME (IN**3):
   50.00000
(15) MAXIUM POPPET OR SPOOL DISPLACEMENT (IN):
   .2000000
(16) THE DAMPING COEFFICIENT OF THE SPOOL
   .5000000E-01
(17) UNSTEADY FLOW FORCE COEFFICIENT (LBF-SEC/IN):
   .0
INITIAL CONDITIONS:
(103) SYSTEM PRESSURE (PSI):
   250.0000


COMPONENT...         76
(1) THE TANK PRESSURE (PSI):
   .0

                           *********************
                           * SIMULATION RESULT *
                           *********************

UNITS USED:
          PRESSURE         : POUND PER SQUARE INCH (PSI)
          FLOW RATE        : CUBIC INCH PER SEC
          FORCE            : POUND (LBF)
          VELOCITY         : INCH PER SECOND
          TORQUE           : INCH - POUND
          ANGULAR VELOCITY : RADIANS PER SECOND
          TIME             : SECOND
                                              COMPONENT  : 3

                                              I.D. NO.   : 31111
           TIME                * VERSUS *
                                              PORT NO.   : 1

         PERFORMANCE: TIME                    PERFORMANCE: PRESSURE
           (X-AXIS)                               (Y-AXIS)

                                                       PAGE   1

                            MIN                                                         MAX
                         2.500E+02                                                   8.440E+02

           100         311
     0.0           0.250E+03     ]+
     0.200E-02     0.364E+03     ]-----------+
     0.400E-02     0.484E+03     ]-----------------------+
     0.600E-02     0.604E+03     ]-----------------------------------+
     0.800E-02     0.724E+03     ]-----------------------------------------------+
     0.100E-01     0.844E+03     ]-----------------------------------------------------------+
```

```
********   END  OF  SIMULATION   ********

(1) RERUN THIS PROGRAM AFTER MAKING PARAMETER CHANGES.
(2) STOP SIMULATION.
CHOOSE ONE OF THE ABOVE:
1
ARE THERE ANY CHANGES YOU WISH TO MAKE IN THE FOLLOWING

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
1
THE VALUES OF FLUID PROPERTIES SET ARE:
1 - FLUID BULK MODULUS       :   150000.0      PSI
2 - FLUID DENSITY            :   .7800000E-04 LBF-SEC**2/IN**4
3 - FLUID ABSOLUTE VICOSITY :   .2000000E-05 LBF-SEC/IN**2
4 - SET ALL FLUID PROPERTIES TO DEFAULT VALUES.
5 - RETURN TO SYSTEM RERUN MENU.
SELECT ONE OF THE OPTIONS...
1
ENTER THE NEW VALUE FOR THIS PARAMETER:
160000.0
THE VALUES OF FLUID PROPERTIES SET ARE:
1 - FLUID BULK MODULUS       :   160000.0      PSI
2 - FLUID DENSITY            :   .7800000E-04 LBF-SEC**2/IN**4
3 - FLUID ABSOLUTE VICOSITY :   .2000000E-05 LBF-SEC/IN**2
4 - SET ALL FLUID PROPERTIES TO DEFAULT VALUES.
5 - RETURN TO SYSTEM RERUN MENU.
SELECT ONE OF THE OPTIONS...
5
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
y

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
2
YOU HAVE CHOSEN TO CHANGE THE NATURE OF SIMULATION
PLEASE SET THE NATURE OF SIMULATION FOR THE SYSTEM:
(1) STATIC SIMULATION
(2) DYNAMIC SIMULATION
CHOOSE ONE OF THE ABOVE :
2
YOU HAVE CHOSEN TO PERFORM A DYNAMIC SIMULATION
NOW SET SYSTEM DYNAMIC SIMULATION INFORMATION :

THE TIME PARAMETERS SET PREVIOUSLY ARE:
 (1) SIMULATION STARTING TIME (SEC):     0.0
 (2) SIMULATION STEP SIZE (SEC)    :     0.10000E-03
 (3) SIMULATION FINAL TIME (SEC)   :     0.10000E-01
 (4) DESIRED OUTPUT STEP SIZE (SEC):     0.20000E-02
PLEASE RESET TIME PARAMETERS FOR A RERUN....
```

H

I

Rerunning or Modifying a Model

```
(1) SIMULATION STARTING TIME (SEC):
0.0

(2) SIMULATION STEP SIZE (SEC):
1.0e 4

(3) SIMULATION FINAL TIME (SEC):
0.1

(4) DESIRED OUTPUT STEP SIZE (SEC):
2.0e 3

SET INTEGRATION METHOD DESIRED IN SIMULATION PROCESS:
(1) EULER'S METHOD
(2) RUNGE - KUTTA 4TH ORDER METHOD
CHOOSE ONE OF THE ABOVE:
1
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
n

DO YOU WANT THE INPUT DATA TO BE PRINTED OUT ?
ENTER YES OR NO
n

DO YOU WANT A HARDCOPY OF THE RUN?
ENTER YES OR NO.
n

                          **********************
                          * SIMULATION RESULT *
                          **********************

UNITS USED:
          PRESSURE         : POUND PER SQUARE INCH (PSI)
          FLOW RATE        : CUBIC INCH PER SEC
          FORCE            : POUND (LBF)
          VELOCITY         : INCH PER SECOND
          TORQUE           : INCH - POUND
          ANGULAR VELOCITY : RADIANS PER SECOND
          TIME             : SECOND
                                          COMPONENT  : 3

                                          I.D. NO.   : 31111
           TIME             * VERSUS *
                                          PORT NO.   : 1

         PERFORMANCE: TIME                PERFORMANCE: PRESSURE
          (X-AXIS)                            (Y-AXIS)

                                                   PAGE   1

                             MIN                                                  MAX
                          2.500E+02                                            1.050E+03

      100         311
   0.0         0.250E+03     I+
   0.200E-02   0.372E+03     I---------+
   0.400E-02   0.500E+03     I-------------------+
```

```
0.[illegible]E-02   0.6[illegible]E+03   ]--------------------------+
0.[illegible]E-02   0.7[illegible]E+03   ]-----------------------------------+
0.1[illegible]E-01  0.8[illegible]E+03   ]-------------------------------------------------+
0.120E-01   0.960E+03   ]--------------------------------------------------------+
0.140E-01   0.101E+04   ]------------------------------------------------------------+
0.160E-01   0.104E+04   ]---------------------------------------------------------------+
0.180E-01   0.105E+04   ]---------------------------------------------------------------+
0.200E-01   0.105E+04   ]---------------------------------------------------------------+
0.220E-01   0.105E+04   ]---------------------------------------------------------------+
0.240E-01   0.105E+04   ]---------------------------------------------------------------+
0.260E-01   0.105E+04   ]---------------------------------------------------------------+
0.280E-01   0.105E+04   ]---------------------------------------------------------------+
0.300E-01   0.105E+04   ]---------------------------------------------------------------+
0.320E-01   0.105E+04   ]---------------------------------------------------------------+
0.340E-01   0.105E+04   ]---------------------------------------------------------------+
0.360E-01   0.105E+04   ]---------------------------------------------------------------+
0.380E-01   0.105E+04   ]---------------------------------------------------------------+
0.400E-01   0.105E+04   ]---------------------------------------------------------------+
0.420E-01   0.105E+04   ]---------------------------------------------------------------+
0.440E-01   0.105E+04   ]---------------------------------------------------------------+
0.460E-01   0.105E+04   ]---------------------------------------------------------------+
0.480E-01   0.105E+04   ]---------------------------------------------------------------+
0.500E-01   0.105E+04   ]---------------------------------------------------------------+
0.520E-01   0.105E+04   ]---------------------------------------------------------------+
0.540E-01   0.105E+04   ]---------------------------------------------------------------+
0.560E-01   0.105E+04   ]---------------------------------------------------------------+
0.580E-01   0.105E+04   ]---------------------------------------------------------------+
0.600E-01   0.105E+04   ]---------------------------------------------------------------+
0.620E-01   0.105E+04   ]---------------------------------------------------------------+
0.640E-01   0.105E+04   ]---------------------------------------------------------------+
0.660E-01   0.105E+04   ]---------------------------------------------------------------+
0.630E-01   0.105E+04   ]---------------------------------------------------------------+
0.700E-01   0.105E+04   ]---------------------------------------------------------------+
0.720E-01   0.105E+04   ]---------------------------------------------------------------+
0.740E-01   0.105E+04   ]---------------------------------------------------------------+
0.760E-01   0.105E+04   ]---------------------------------------------------------------+
0.780E-01   0.105E+04   ]---------------------------------------------------------------+
0.800E-01   0.105E+04   ]---------------------------------------------------------------+
0.820E-01   0.105E+04   ]---------------------------------------------------------------+
0.840E-01   0.105E+04   ]---------------------------------------------------------------+
0.860E-01   0.105E+04   ]---------------------------------------------------------------+
0.880E-01   0.105E+04   ]---------------------------------------------------------------+
0.900E-01   0.105E+04   ]---------------------------------------------------------------+
0.920E-01   0.105E+04   ]---------------------------------------------------------------+
0.940E-01   0.105E+04   ]---------------------------------------------------------------+
0.960E-01   0.105E+04   ]---------------------------------------------------------------+
0.980E-01   0.105E+04   ]---------------------------------------------------------------+

                                     PAGE   2

                        MIN                                                      MAX
                     2.500E+02                                                1.050E+03

    100        311
0.100E+00   0.105E+04   ]---------------------------------------------------------------+


********   END  OF  SIMULATION   ********
```

```
(1) RERUN THIS PROGRAM AFTER MAKING PARAMETER CHANGES.
(2) STOP SIMULATION.
CHOOSE ONE OF THE ABOVE:
1
ARE THERE ANY CHANGES YOU WISH TO MAKE IN THE FOLLOWING

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
1
THE VALUES OF FLUID PROPERTIES SET ARE:
1 - FLUID BULK MODULUS       :   160000.0      PSI
2 - FLUID DENSITY            :   .7800000E-04 LBF-SEC**2/IN**4
3 - FLUID ABSOLUTE VICOSITY  :   .2000000E-05 LBF-SEC/IN**2
4 - SET ALL FLUID PROPERTIES TO DEFAULT VALUES.
5 - RETURN TO SYSTEM RERUN MENU.
SELECT ONE OF THE OPTIONS...
4
THE VALUES OF FLUID PROPERTIES SET ARE:
1 - FLUID BULK MODULUS       :   150000.0      PSI
2 - FLUID DENSITY            :   .7000000E-04 LBF-SEC**2/IN**4
3 - FLUID ABSOLUTE VICOSITY  :   .2000000E-05 LBF-SEC/IN**2
4 - SET ALL FLUID PROPERTIES TO DEFAULT VALUES.
5 - RETURN TO SYSTEM RERUN MENU.
SELECT ONE OF THE OPTIONS...
5
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
y

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
2
YOU HAVE CHOSEN TO CHANGE THE NATURE OF SIMULATION
PLEASE SET THE NATURE OF SIMULATION FOR THE SYSTEM:
(1) STATIC SIMULATION
(2) DYNAMIC SIMULATION
CHOOSE ONE OF THE ABOVE :
1
YOU HAVE CHOSEN TO PERFORM A STATIC SIMULATION
NOW SET THE TYPE OF STATIC SIMULATION DESIRED:
(1) AT OPERATING POINT ONLY
(2) EVALUATING THE ENTIRE PERFOMANCE CURVE
CHOOSE ONE OF THE ABOVE:
1
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
y

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
3
```

← Rerun the Simulation and change the Simulation from Dynamic into Static.

```
ENTER THE SEQUENCE NUMBER OF THE COMPONENT THAT
NEEDS TO BE CHANGED.
3


COMPONENT...     31111


NATURE OF SIMULATION: (3) DYNAMIC STATE - DETAIL DESIGNPARAMETERS KNOWN


IS THERE ANY CHANGE OF THE NATURE OF MODEL
FOR THIS COMPONENT ? (I.D.:      31111)
ENTER  YES OR  NO
y


SET THE NATURE OF SIMULATION FOR COMPONENT:      31111 USED
    (1) STEADY STATE -- PERFORMANCE DATA KNOWN
    (2) STEADY STATE -- DETAIL DESIGN PARAMETERS KNOWN
    (3) DYNAMIC STATE - DETAIL DESIGN PARAMRTERS KNOWN
CHOOSE ONE OF THE ABOVE:
2
(5) CRACKING PRESSURE (PSI):
   800.0000
(6) UPSTREAM PRESSURE REACTION AREA (IN**2):
   .5000000E-01
(7) DOWNSTREAM PRESSURE REACTION AREA (IN**2):
   .5000000E-01
(8) SPRING CONSTANT (LBF/IN):
   500.0000
(9) FLOW DISCHARGE COEFFICIENT AT PORT 3:
   .6100000
(10) AREA GRADIENT OF PORT 3 (IN**2/IN):
   .3889999
(11) FLOW JET ANGLE OF PORT 3 (DEGREE):
   20.00000
(12) LEAKAGE FLOW COEFFICIENT (IN**3/SEC/PSI):
   .0

ENTER THE SEQUENCE NUMBER OF THE PARAMETER THAT
NEEDS TO BE CHANGED :
5
ENTER THE NEW VALUE FOR THIS PARAMETER:
795.0


ANY OTHER PARAMETER OF THIS COMPONENT THAT NEEDS
TO BE CHANGED?
ENTER YES OR NO
n


IS THERE ANY OTHER COMPONENT'S INPUT DATA THAT NEEDS
TO BE CHANGED?
ENTER YES OR NO
y


ENTER THE SEQUENCE NUMBER OF THE COMPONENT THAT
NEEDS TO BE CHANGED.
2
```

```
COMPONENT...      4311

NATURE OF SIMULATION: (1) STEADY STATE -- PERFORMANCE DATA KNOWN

IS THERE ANY CHANGE OF THE NATURE OF MODEL
FOR THIS COMPONENT ? (I.D.:       4311)
ENTER  YES OR  NO
n

ENTER THE COEFFICIENTS OF PERFORMANCE CURVE
FLOW = K1 + K2*PRESS + K3*PRESS**2 +K3*PRESS**3
WHERE  PRESS : LOAD PRESSURE  (PSI)
       FLOW : PUMP FLOW RATE (IN**3/SEC)
(1) COEFFICIENT K1:
   20.01999
(2) COEFFICIENT K2:
   .0
(3) COEFFICIENT K3:
   .0
(4) COEFFICIENT K4:
   .0

ENTER THE SEQUENCE NUMBER OF THE PARAMETER THAT
NEEDS TO BE CHANGED :
1
ENTER THE NEW VALUE FOR THIS PARAMETER:
20.0


ANY OTHER PARAMETER OF THIS COMPONENT THAT NEEDS
TO BE CHANGED?
ENTER YES OR NO
n


IS THERE ANY OTHER COMPONENT'S INPUT DATA THAT NEEDS
TO BE CHANGED?
ENTER YES OR NO
n
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
y

1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
5
               SPECIFY DESIRED OUTPUT
                    CURRENT SETUP
            Y AXIS                             X-AXIS
NO. COMPONENT  PORT   WHAT        COMPONENT  PORT   WHAT
 1    31111      1   PRESSURE  VS.              TIME

THE OUTPUT FORMAT CHOSEN IS PLOTS   .


1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
```

```
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
2

DEFINE A PLOT OR TABLE

             SELECTED    CURRENTLY
1 - X AXIS     NO            0
2 - Y AXIS     NO            0
3 - END
OPTION?
1

SELECT THE X-AXIS LABEL.

NO.   COMPONENT    PORTS
 0      TIME
 1     91112         1
 2      4311         3
 3     31111         5
 4        76         4

OPTION?
3

SELECT THE X-AXIS PORT.

COMPONENT NO. 31111 HAS 5 PORTS.
SELECT THE PORT NUMBER TO MODEL-
1

SELECT THE X-AXIS VARIABLE.

SELECT THE VARIABLE TO MEASURE FOR COMPONENT
NO. 31111, PORT 1 ON THE X-AXIS.

1 - PRESSURE
2 - FLOW RATE
3 - FORCE
4 - VELOCITY
5 - TORQUE
6 - ANGULAR VELOCITY

OPTION?
1

DEFINE A PLOT OR TABLE

             SELECTED    CURRENTLY
1 - X AXIS     YES          311
2 - Y AXIS     NO            0
3 - END
OPTION?
2

SELECT THE Y-AXIS LABEL.

NO.   COMPONENT    PORTS
```

```
 0      TIME
 1     91112        1
 2      4311        3
 3     31111        5
 4        76        4

OPTION?
3

SELECT THE Y-AXIS PORT.

COMPONENT NO. 31111 HAS 5 PORTS.
SELECT THE PORT NUMBER TO MODEL-
3

SELECT THE Y-AXIS VARIABLE.

SELECT THE VARIABLE TO MEASURE FOR COMPONENT
NO. 31111, PORT 3 ON THE Y-AXIS.

1 - PRESSURE
2 - FLOW RATE
3 - FORCE
4 - VELOCITY
5 - TORQUE
6 - ANGULAR VELOCITY

OPTION?
2

DEFINE A PLOT OR TABLE

            SELECTED    CURRENTLY
1 - X AXIS    YES          311
2 - Y AXIS    YES          332
3 - END
OPTION?
3
               SPECIFY DESIRED OUTPUT
                    CURRENT SETUP
           Y-AXIS                           X-AXIS
NO. COMPONENT  PORT   WHAT          COMPONENT  PORT   WHAT
 1    31111      1    PRESSURE  VS.             TIME
 2    31111      3    FLOW RATE VS.    31111      1   PRESSURE

THE OUTPUT FORMAT CHOSEN IS PLOTS    .

1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
3

ENTER THE NUMBER OF THE PLOT OR TABLE THAT YOU WANT TO DELETE-
1

PLOT OR TABLE  1 DELETED.
               SPECIFY DESIRED OUTPUT
                    CURRENT SETUP
           Y-AXIS                           X AXIS
```

```
NO. COMPONENT  PORT   WHAT      .     COMPONENT  PORT   WHAT
 1    31111      3   FLOW RATE VS.     31111      1   PRESSURE

THE OUTPUT FORMAT CHOSEN IS PLOTS  .

1   SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2   DEFINE A PLOT OR TABLE
3   DELETE A CHOSEN PLOT OR TABLE
4   RETURN TO MAIN MENU
OPTION?
4
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
n

DO YOU WANT THE INPUT DATA TO BE PRINTED OUT ?
ENTER YES OR NO
n

DO YOU WANT A HARDCOPY OF THE RUN?
ENTER YES OR NO.
n

                    *********************
                    * SIMULATION RESULT *
                    *********************
UNITS USED:
        PRESSURE         : POUND PER SQUARE INCH (PSI)
        FLOW RATE        : CUBIC INCH PER SEC
        FORCE            : POUND (LBF)
        VELOCITY         : INCH PER SECOND
        TORQUE           : INCH - POUND
        ANGULAR VELOCITY : RADIANS PER SECOND
        TIME             : SECOND

     COMPONENT  : 3                          COMPONENT  : 3

     I.D. NO.   : 31111                      I.D. NO.   : 31111
                            * VERSUS *
     PORT NO.   : 1                          PORT NO.   : 3

     PERFORMANCE: PRESSURE                   PERFORMANCE: FLOW RATE
       (X-AXIS)                                 (Y-AXIS)
*******PLOT UNNEEDED STRAIGHT LINE********

X VALUE IS:      0.111E+04
Y VALUE IS:      0.201E+02


 ********   END  OF  SIMULATION   ********


(1) RERUN THIS PROGRAM AFTER MAKING PARAMETER CHANGES.
(2) STOP SIMULATION.
CHOOSE ONE OF THE ABOVE:
1
ARE THERE ANY CHANGES YOU WISH TO MAKE IN THE FOLLOWING
```

```
1 - THE VALUES OF FLUID PROPERTIES
2 - THE NATURE OF THE SIMULATION FOR THE SYSTEM
3 - SYSTEM COMPONENTS PARAMETERS AND THE NATURE OF MODEL
4 - THE CONNECTOR BETWEEN THE COMPONENTS
5 - THE OUTPUT FORMATING
6 - REMODEL THE ENTIRE SYSTEM
CHOOSE ONE OF THE ABOVE:
5
                SPECIFY DESIRED OUTPUT
                     CURRENT SETUP
            Y-AXIS                             X-AXIS
NO. COMPONENT  PORT   WHAT            COMPONENT  PORT   WHAT
 1    31111      3   FLOW RATE VS.      31111      1   PRESSURE

THE OUTPUT FORMAT CHOSEN IS PLOTS    .


1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
1


SELECT OUTPUT FORMAT
1 - TABLES
2 - PLOTS
OPTION?
1
                SPECIFY DESIRED OUTPUT
                     CURRENT SETUP
            Y-AXIS                             X-AXIS
NO. COMPONENT  PORT   WHAT            COMPONENT  PORT   WHAT
 1    31111      3   FLOW RATE VS.      31111      1   PRESSURE

THE OUTPUT FORMAT CHOSEN IS TABLES    .


1 - SELECT OUTPUT FORMAT (TABLES OR PLOTS)
2 - DEFINE A PLOT OR TABLE
3 - DELETE A CHOSEN PLOT OR TABLE
4 - RETURN TO MAIN MENU
OPTION?
4
ARE THERE ANY OTHER PARAMETERS CHANGES YOU WISH TO MAKE?
ENTER YES OR NO
n


DO YOU WANT THE INPUT DATA TO BE PRINTED OUT ?
ENTER YES OR NO
n


DO YOU WANT A HARDCOPY OF THE RUN?
ENTER YES OR NO.
n


                          **********************
                          * SIMULATION RESULT *
                          **********************

UNITS USED:
          PRESSURE          : POUND PER SQUARE INCH (PSI)
```

```
FLOW RATE        : CUBIC INCH PER SEC
FORCE            : POUND (LBF)
VELOCITY         : INCH PER SECOND
TORQUE           : INCH - POUND
ANGULAR VELOCITY : RADIANS PER SECOND
TIME             : SECOND

COMPONENT   : 3                       COMPONENT   : 3

I.D. NO.    : 31111                   I.D. NO.    : 31111
                        * VERSUS *
PORT NO.    : 1                       PORT NO.    : 3

PERFORMANCE: PRESSURE                 PERFORMANCE: FLOW RATE
  (X-AXIS)                                 (Y-AXIS)

    1110.3                                20.060


********   END  OF  SIMULATION   ********


(1) RERUN THIS PROGRAM AFTER MAKING PARAMETER CHANGES.
(2) STOP SIMULATION.
CHOOSE ONE OF THE ABOVE:
2

EXITING.....
READY
```

END

FILMED

2-85

DTIC