MARINTEK REPORT

MARINTEK

Norwegian Marine Technology Research Institute

Postal address:
P.O.Box 4125 Valentinlyst
NO-7450 Trondheim, NORWAY

Location:
Marine Technology Centre
Otto Nielsens veg 10

Phone: +47 7359 5500
Fax: +47 7359 5776

http://www.marintek.sintef.no

Enterprise No.: NO 937 357 370 MVA

SINTEF

TITLE

SIMO - User's Manual Version 3.6

AUTHOR(S)

SIMO project team

CLIENT(S)

FILE CODE	CLASSIFICATION	CLIENTS REF.	
MT51 F93-0184	Confidential		
CLASS. THIS PAGE	**ISBN**	**PROJECT NO.**	**NO. OF PAGES/APPENDICES**
		516412	38

REFERENCE NO.		PROJECT MANAGER (NAME, SIGN.)	VERIFIED BY (NAME, SIGN.)
2001-516412-01131560 P		Harald Ormberg	Halvor Lie
REPORT NO.	**DATE**	**APPROVED BY (NAME, POSITION, SIGN.)**	
516412.00.04	2007-11-16	Frode Meling, Research Director	

ABSTRACT

KEYWORDS	ENGLISH	NORWEGIAN
GROUP 1	Hydrodynamics	Hydrodynamikk
GROUP 2	Simulation	Simulering
SELECTED BY AUTHOR		

TABLE OF CONTENTS

1. INTRODUCTION

SIMO is a computer program for simulation of motions and station-keeping behaviour of a complex system of floating vessels and suspended loads. Essential features are:

- Flexible modelling of multibody systems
- Non-linear time domain simulation of wave-frequency as well as low-frequency forces.
- Environmental forces due to wind, waves and current.
- Passive and active control forces.
- Interactive or batch simulation.

The user documentation consists of the following parts:

- **General description**

  A short description of the most important system functions and features.

- **Theory manual**

  A detailed description of the theory on which the program is based.

- **User's manual**

  All information necessary for correct and efficent use of the program.

- **Implementation and testing manual**

  Complete run examples through all modules are described there. The manual will be of help to new or unexperienced users

- **Verification manual**

  Results from the verification study.

1.1 Program Modules

The program is divided into 6 separate modules:

INPMOD	-	manipulation of input data
STAMOD	-	initial condition and static equilibrium
DYNMOD	-	dynamic response calculation
OUTMOD	-	output module
S2XMOD	-	export of time series to various file formats
PLOMOD	-	plotting module (common with the Riflex program system)

2. PRINCIPLES FOR USE

2.1 Interactive Communication

The program is basically an interactive program, but can also be run in batch mode. The input from the terminal can be divided into five types:

- Selection among a number of options that are presented on the screen.
- Giving parameters to a command.
- Giving text strings for identification
- Yes/no answers.
- Waiting on user response.

Common to all types of input is that they allow definition of a macro file, they allow a macro file to be executed and they allow temporary switching of control to the operating system.

The program uses "MAIS - MArintek Input System" for handling the interactive communication. This system is described in appendix B. Useful features with this system are :

- Numerical values have default values and allowable range. Range check can be deactivated by 'range off'.
- Lower-case and upper-case letters are accepted as input.
- The program will never stop because of wrong input.
- Switch commands (print, file) may be given anywhere.
- User input may be logged on macro file for documentation or later input.

2.1.1 Macro System

Macro system commands (starting with @) can be used anywhere where the program waits for input. The commands must contain enough characters so that they are unique. When the user writes @ HELP the a short description is written at the terminal.

The following macro commands are available:

@ HELP - Writes a list of all macro system commands.

@ OPEN filename - The command opens a macro file for writing. All questions and answers until @ CLOSE is given will be stored on the macro file. If the command is given a second time before @ CLOSE, ``@ DO filename" is stored at the macro file.

@ CLOSE - The command closes the macro file which is opened for writing.

@ DO filename n - Input is read from the specified file. The file is read n times, n may be omitted.

@ LIST filename - The contents of the file is listed on the terminal.

@ SPAWN @ PAUSE - The program is set in a ``waiting condition". A new process is created. This means that all operating system commands can be given.

@ WAIT ON/OFF - Changes the program switch IWAIT to 1/0 resp. This switch is used internally in the program to check whether ``press return to continue" shall be written on the terminal

@ DISPLAY ON/OFF	-	Changes the program switch IDISP to 1/0 respectively. This switch is used by the program to enable/supress output to the terminal when macro files are executed interactively.
@ FILE ifile	-	Changes the program switch IFILE. Allowable range is 0-5. This switch is used by the program to change the amount of print to file.
@ RESULT iresu	-	Changes the program switch IRESU. Allowable range is 0-5. This switch is used by the program to change the amount of information to the terminal.
@ STATUS	-	Lists the values of the macro switches.
*****	-	The parameter parameter that is currently read shall be read from terminal when a macro file is executed.

Assume that the user wants to undertake several batch runs. To prepare an input file the user first runs the program in demand and logs the terminal input at file FILA.MAC. This is done by giving the command

@OPEN FILA.MAC

This causes the file FILA.MAC to be opened. All input to the program that is given from terminal will be stored at this file. When this first run is finished FILA.MAC may be duplicated and modified if wanted.

When the program waits at the same question where recording of macro file started, the commands can be repeated by:

@DO FILA

2.2 Selection of execution mode

Each "SIMO" program may be run in two different ways,

- interactive mode and
- batch mode.

Selection between interactive and batch mode is done within the modules. A symbolic file with name "SIMO_DEF.DAT" is searched for at the default directory. The file should contain 3 lines with a character variable on each line:

```
CHMODE
CHSYS
CHINI
```

If the file is not found, the execution mode will be interactive, and both CHSYS and CHINI will be blank. Batch mode is selected only if CHMODE has the value >batch= or 'BATCH'. CHSYS and CHINI are used for generation of default file names as explained in Section 3.6.

Batch mode is useful when several simulations shall be undertaken. The necessary input for running the program may prepared with a text editor, but it is recommended that the user prepares the input "run-file" by running the program in demand mode and logging the input at a macro file. If necessary the macro file can be edited.

Batch mode and interactive mode are different only in the way they respond to detected errors in the program execution and the amount of output presented at standard output device (screen).

Details about running the program on different computers are found in the implementation and testing manual.

2.3 Units and physical constants

All input data must be given in a consistent set of SI units, which are defined as a part of the input data set.

	Property	Notation	Example
Basic units	Length	L	*m*
	Time	T	*s*
	Mass	M	*Mg = tonne*
Derived units	Force	$F = ML/T^2$	Mgm/s^2
	Pressure	$P = F/L^2$	$kN/m^2 = Pa$
	Velocity	V = L/T	*m/s*
	Acceleration	A = V/T	m/s^2

Angles are given in degrees, (*deg*), and coefficients which are multiplied with rotational state variables are given for rotations in radians. Angular velocity is given in radians per time unit, for instance (*rad/s*). Thruster speed and rotation are given in revolutions per minute, (*RPM*). Internally in the program, all angles and rotations are converted to radians.

2.4 Error Messages

An error is reported where it is detected. The string to be searched for on the print file is:

"*** ERR"

Error flags and messages are transferred towards the root of the call tree. In interactive mode, the user is given control at a high level and can decide whether to continue or terminate the program. In batch mode, the program is terminated at the top level.

3. PROGRAM LAYOUT

The program system consists of five programs or modules communicating by a file system as shown below.

<table>
<tr><td>INPMOD</td><td rowspan="6">File system for communi-cation between modules</td><td>Read and manipulate system description</td></tr>
<tr><td>STAMOD</td><td>Read system description, static analyses, define initial condition.</td></tr>
<tr><td>DYNMOD</td><td>Dynamic analyses, generation of time series</td></tr>
<tr><td>OUTMOD</td><td>Post-processing of time series</td></tr>
<tr><td>S2XMOD</td><td>Export of time series</td></tr>
<tr><td>PLOMOD</td><td>Plotting</td></tr>
</table>

A complete dynamic analysis must include run of the modules STAMOD and DYNMOD. Post processing or export of results is done by OUTMOD or alternatively S2XMOD. However, an efficient data base system simplifies the work during the analysis by storing input data and intermediate results.

In addition, graphic presentation will be possible by use of the interactive plotting program PLOMOD. This program communicates with SIMO by a file, PLOFIL.

The layout of the program system is presented in Figure 3.1. Each module will be further detailed in the following.

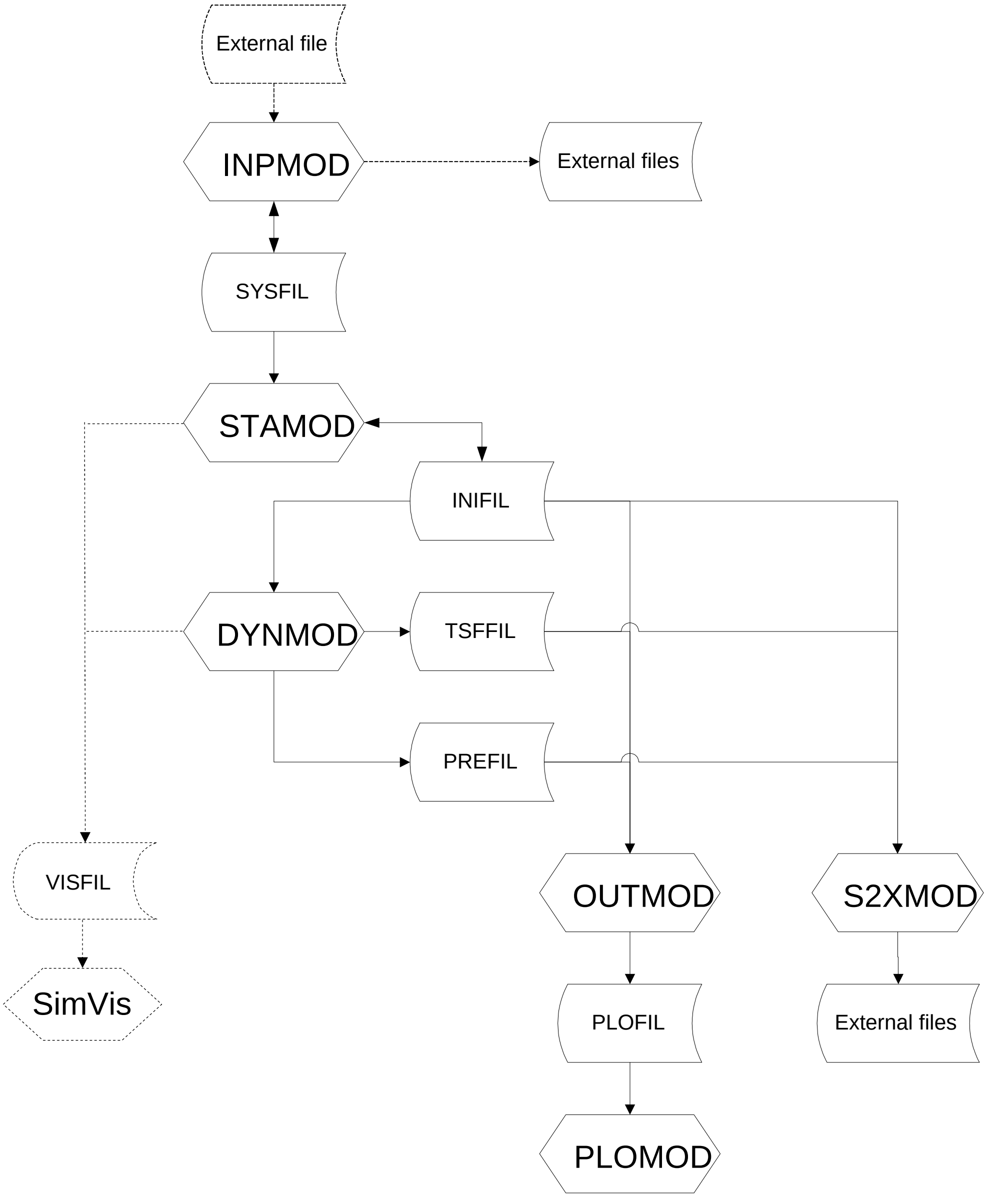

Figure 3.1 Layout of the SIMO program system and file communication between modules

3.1 INPMOD, input data manipulation

The purpose of the module is to provide interfaces to external input data sources, for example hydrodynamic programs, and to modify the system description file, SYSFIL.

3.2 STAMOD, initial condition and static equilibrium

The purpose of the module is to define the initial condition for the dynamic simulation. The description of the system to be simulated is read from a file, SYSFIL.

Selection between environmental conditions can be done.

A static equilibrium position may be calculated with average environmental forces applied. Natural periods and oscillation modes of the system may be calculated.

The initial conditions is written to the file INIFIL for use by DYNMOD.

3.3 DYNMOD, dynamic response calculation

The purpose of the module is to calculate responses in the time domain. Before starting time integration of the equation of motion, the various simulation parameters must be initialized.

3.4 OUTMOD, output module

The purpose of the module is to read time series files generated in the DYNMOD module, generate print and plot of time series and statistical parameters.

3.5 S2XMOD, export of time series

The purpose of the module is to export result from dynamic analysis, i.e. time series to variuos file formats. In present verion export to MATLAB ("m"-file), export to ASCII-file and export to DIRECT ACCESS file are available. In addition the module provides siple statistics and plot of time series.

3.6 PLOMOD, plotting module

The purpose of the module is to plot results generated by OUTMOD.

3.7 SimVis, 3D visualization program

Stand-alone program for 3-dimensional visualization of static and dynamic analysis, based on GLView.

3.8 File system

When running the SIMO modules, different kind of files are needed. Each module communicates to terminal (or read a macro file) and generates output on the print file.

The system description file, SYSFIL, contains a description of bodies, couplings and environmental data. It is a symbolic data file which is read by STAMOD. The file is also read by OUTMOD in order to find geometry data for static configuration plots and snapshots.

The initial condition file, INIFIL, is generated by STAMOD. If only modifications to an already defined initial condition is required, STAMOD can also read INIFIL . DYNMOD reads INIFIL to fetch the system to be simulated, and OUTMOD reads INIFIL to get initial positions and key data.

The pregenerated data file, PREFIL is used for internal storage of simulation parameters in DYNMOD. It is also read by OUTMOD to fetch information on the channels that are stored on TSFIL.

The time series file, TSFIL, contains all time series. It is generated by DYNMOD. OUTMOD reads the time series to be processed from TSFIL.

The plot file, PLOFIL, is generated by OUTMOD and contains plot information to be read by PLOMOD.

The default file name system utilizes the **system identifier** and **initial condition identifiers**, *chsys* and *chini*. Both contain up to 8 characters. At the beginning of each module, it is asked if default file names should be used. The default file names will be:

File type	Short name	File name syntax	Example *chsys* = s1 *chini* = i1
System description file	SYSFIL	sys[-*chsys*].dat	sys-s1.dat
Initial condition file	INIFIL	ini[-*chsys*[-*chini*]].sam	ini-s1-i1.sam
Print from INPMOD	PRIFIL	pri[-*chsys*].lis	pri-s1.lis
Print from STAMOD	PRSFIL	prs[-*chsys*].lis	prs-s1.lis
Print from DYNMOD	PRDFIL	prd[-*chsys*[-*chini*]].lis	prd-s1-i1.lis
Print from OUTMOD	PROFIL	pro[-*chsys*[-*chini*]].lis	pro-s1-i1.lis
Plot file from OUTMOD	PLOFIL	plo[-*chsys*[-*chini*]].ffi	plo-s1-i1.ffi
Pregenerated data file	PREFIL	pre[-*chsys*[-*chini*]].sam	pre-s1-i1.sam
Time series file	TSFFIL	tsf[-*chsys*[-*chini*]].ffi	tsf-s1-i1.ffi

3.9 Running SIMO

SIMO is normally started by typing **simo** at the command line prompt in the operating system. The system then typically responds by:

```
********************************************************
*   WELCOME TO                                         *
*                                                      *
*    SSSSSSSSSS    III   MMMMM     MMMM     OOOOOOOO    *
*   SSSSSSSSSSSS   III   MMMMM   MMMMM    OOOOOOOOOO   *
*   SSS            III   MMMMMM MMMMMM   OOO      OOO  *
*   SSS            III   MMM MMMMM MMM   OOO      OOO  *
*   SSSSSSSSSS     III   MMM  MMM  MMM   OOO      OOO  *
*   SSSSSSSSSSS    III   MMM   M   MMM   OOO      OOO  *
*           SSS    III   MMM       MMM   OOO      OOO  *
*           SSS    III   MMM       MMM   OOO      OOO  *
*   SSSSSSSSSSS    III   MMM       MMM    OOOOOOOOOO   *
*    SSSSSSSSS     III   MMM       MMM     OOOOOOOO    *
*                                                      *
*   SIMULATION OF COMPLEX MARINE OPERATIONS            *
********************************************************

Enter system identification > sysa
Enter initial condition identification > inib

I-> SELECTION OF PROGRAM MODULE
I
I      INP  - SIMO INPMOD
I      STA  - SIMO STAMOD
I      DYN  - SIMO DYNMOD
I      OUT  - SIMO OUTMOD
I      PLO  - SIMO/RIFLEX PLOMOD
i      S2X  - SIMO S2XMOD
I
I      PINP – Print postscript inpmod/stamod/outmod file (simo.plt)
I      PPLO - Print postscript plomod file (pscr-meta.dat)
I
I      DEF  - Redefine default files (for default file names)
I      LIS  - List identifiers (for default file name)
I
I      TERM - Terminate
I
Select option >
```

The user is first asked to specify the system identifier and initial condition identifier. They are described in previous sections, and are used to transfer default file names between the modules.

The user has the following options:

- switch between the program modules
- send print files to printer
- change default file names
- terminate SIMO program system

If for example the INPMOD module is selected, the system will respond with the following:

```
**********************************************
*                                            *
*   III   N   N  PPPP   M   M   OOO   DDDD   *
*    I    NN  N  P   P  MM MM  O   O  D   D  *
```

```
        *    I    N N N  PPPP   M M M  O    O  D    D  *
        *    I    N  NN  P      M   M  O    O  D    D  *
        *   III   N   N  P      M   M   OOO    DDDD    *
        *                                              *
        *     Version     2     Release     2          *
        *                                              *
        *     Module linked at 27-OCT-1994 11:02       *
        ************************************************

        System identifier . . . . . . . : sysa
        Initial condition identifier. . : inib

    Use default file names ? (Y) :
```

The module name, version and release number and the link date/time, the system and initial condition identifiers appears on the screen. It is strongly recommended to use default file names.

```
   +--------------------------------------------------------------------+
   !   File : pri-sysa.lis                                              !
   !   has been opened as PRINT FILE                                    !
   +--------------------------------------------------------------------+

    The module allows test print from maximum  10 routines

    Specify number of routines           (               0 ) :

        The following default plotting parameters are used:
        ---------------------------------------------------

        Screen device                 :   72
        Plotting device               :   90
        Plot file                     : simo.plt

    Use default values ? (Y)                                        :
```

The file name for printed output from the module is displayed. Error massages will be written on this file. There is an option for output of test print from the different program modules. It can be used for debugging purposes in cooperation and on instruction from program system aministrator, and normally the number of routines for test print should be set to 0 (default value).

All modules except DYNMOD generate plots. These can either be seen on screen or sent to postscript files. The default names are simo.plt or pscr-meta.dat (PLOMOD). These files can be opened in i.e. Ghostview and then copied directly into i.e. Microsoft Word.

The following plotting devices are available:

Screen	72	X11 (Windows, X-terminals or equivalent)
Plot fil	80	HPGL
	90	Postscript

4. USE OF INPMOD

The purpose of the module is to provide interfaces to external input data sources, for example hydrodynamic programs, and to modify system description file.

```
I->  INPMOD MAIN MENU
I
I       1 : Body data manipulation
I       2 : Not implemented - Coupling data manipulation
I       3 : Environment data manipulation
I      99 : Terminate
I
I->  Select option :
```

4.1 Body data manipulation

```
I->  INPMOD - body data
I
I       0 : Return
I       1 : Define body type and name
I       2 : Read body data
I       3 : Not implemented - Define body data
I       4 : Manipulate body data
I       5 : Status body data
I       6 : Present body data
I       7 : Write body data
I
I->  Select option (  1 ) :
```

4.1.1 Define body type and name

Body type and name may be defined

4.1.2 Read body data

Body data may be read from a SIMO system description file, or results from hydrodynamic programs such as WAMIT, MULDIF, WADAM or SWIM may be read.

4.1.3 Define body data

Not implemented in present version.

4.1.4 Manipulate body data

The following options exist:

- retardation functions may be calculated from added mass and damping
- added mass and damping may be calculated from retardation functions
- first order motion transfer functions (RAO) may be calculated. Linear damping (if defined) is then used in addition to potential damping

4.1.5 Status body data

A list of data groups defined for a body is presented.

4.1.6 Present body data

The following types of data may be plotted:

- First order wave excitation forces
- Motion transfer functions (RAO)
- Drift force coefficients
- Added mass, damping, retardation functions
- Wind force coefficients
- Linear current drag coefficients
- Quadratic current drag coefficients
- Frequency diagonal of quadratic transfer functions

4.1.7 Write body data

A system description file may be generated and updated with the data presently defined.

5. USE OF STAMOD

The main purpose of STAMOD is to read and check the system description from SYSFIL and to write those data to a direct access file, INIFIL, which will be used in DYNMOD and OUTMOD. This file contains a complete system description, definition of the environment to be applied and the initial positions for dynamic simulation.

```
I->  STAMOD main menu
I
I      1 : Read system description file
I      2 : Read initial condition file
I      3 : Modify/present system
I      4 : Print static condition
I      5 : Equilibrium calculation
I      6 : Eigen value calculation
I      7 : Mooring system optimization
I      8 : Write initial condition file
I      9 : Write file for visualisation
I     99 : Terminate
I
I->  Select option >
```

5.1 Read system description file

The SYSFIL is an ASCII file containing a complete description of the system to be analysed. For a brief describtion of the sys-file layout and file syntax, see section 8. A detailed description of the file contents is given in Appendix A.

The SYSFIL is read twice, the first time only to find dimensioning parameters for the work array, the second time to put all data into the work array.

If an error occurs, messages will be written both on the print file and in some cases also on the terminal. Error messages on the print file are found by searching for the string ``*** ERROR" from the beginning of the file. If a non-fatal error is encountered, reading of SYSFIL will continue until the maximum number of errors are found.

If errors are found during the first scan of SYSFIL and the print file does not give any meaningful indications on the problem, the amount of ouput to the printfile can be increased by the MAIS-command ``@ FILE 5".

Symmetry codes

Wind force coefficients, current force coefficients, wave drift force coefficients and transfer functions are given as functions of directions relative to the body.

In the calculations, extrapolation on directions are not allowed. In order to avoid problems with program termination due to attempted extrapolation, it is advised that both 0 degrees and 360 degrees should be represented in the range of directions on the input file.

If symmetry codes are used, this will automatically be taken care of. For example, if symmetry code is 2 and the directions 20 degrees and 70 degrees are specified, they will be mirrored to -20, 20, 70, 110, 160, 200, 250, 290, 340 and 380 degrees.

NB: It is strongly advised that values are specified for the symmetry lines. For the example above, 0 deg and 90 deg should also be included as input.

5.2 Read initial condition file

An INIFIL that has been written by STAMOD during the present or a previous run may be read. If no changes have been made to the system as defined on SYSFIL, this is an alternative which is much faster than reading SYSFIL.

5.3 Modify / Present system

The menu for system modification is:

```
I->  STAMOD modify/present system
I
I      0 : Return
I      1 : Select environment
I      2 : Initial positions
I      3 : Eliminate degrees of freedom
I      4 : Positioning system
I      5 : Restoring force (incl. environmental forces)
I      6 : Dynamic positioning
I
I->  Select option >
```

5.3.1 Select environment

Several regular and irregular wave conditions, current conditions and wind conditions may be defined on SYSFIL (or INIFIL). The default wave condition will be the present condition. If no condition has been selected, the first one defined on SYSFIL will be the default. Waves, current and wind can be removed individually.

5.3.2 Initial positions

New initial positions for all bodies may be specified. Present positions will be default.

5.3.3 Eliminate degrees of freedom

Elimination of degrees of freedom is implemented by setting the corresponding local acceleration components to zero during time domain simulation (DYNMOD) and equilibrium iteration (STAMOD).

5.3.4 Positioning system

The initial tension may be changed. This is done by moving the anchor position. Print of positioning element locations and plot/print of line characteristics are available.

5.3.5 Restoring forces

The horizontal restoring force for translation in any direction or rotation may be specified. Output is presented on plots and tables.

5.3.6 Dynamic positioning

The following data may be modified:

- Reference position and heading, including the point on the body to be positioned
- Start value on bias forces that are not measured. If static forces have been calculated, the default values will be updated.
- Kalman filter data or PID controller data

5.4 Print static condition

Initial positions for all bodies will be written to the terminal (standard output) and, if selected also to the print file.

Static forces acting on bodies are written to the terminal and optionally to the print file. The following forces are presented:

- Average wave drift force calculated for the present heading, including wave-current interaction if specified.
- Wind force (linear interpolation between directions)
- Current drag force based on linear and quadratic current coefficients (linear interpolation between directions)
- Munk moment caused current velocity and different magnitude of added mass specified in individual degree of freedom
- Positioning element forces
- Thruster forces
- Coupling element forces
- Foreces from general line systems
- Hydrostatic stiffness force
- Gravity and bouyancy forces, also including small volume hydrodynamic force with wave particle elocities and accelerations set to zero. The force contain contributions from current, gravity force and any static soil reaction forces.
- Gravity forces due to time dependend added mass at time zero
- Gravity and buoyancy force on slender elements and fixed body elements
- Current drag forces acting on slender eements and fixed body elements
- Specified forces at time equal to zero
- External forces, i.e. forces from any special force subroutines linked into the program

If a dynamic positioning system is defined, thrust demand will be calculated according to equation (5.3).

5.5 Equilibrium calculation

The following parameters are specified interactively:

T - Typical (maximum) natural period

δx - Position tolerance

$\delta \alpha$ - Direction tolerance

δt - Time step

From position and direction tolerances, the following tolerances for velocities and acceleration are calculated:

$$\omega = 2\pi/T$$
$$\delta\dot{x} = \omega\delta x$$
$$\delta\dot{\alpha} = \omega\delta\alpha \qquad \textbf{(5.1)}$$
$$\delta\ddot{x} = \omega^2\,\delta x$$
$$\delta\ddot{\alpha} = \omega^2\,\delta\alpha$$

By default, critical damping for each degree of freedom can be calculated:

$$c_i = 2\,M_i\,\omega\ ,i = 1,6 \qquad \textbf{(5.2)}$$

where M_i are the diagonal elements of the total mass matrix.

The equilibrium condition is found by stepping the equations of motion. The following force models will be included (if they are defined)

- "Critical damping" force
- Modified retardation function force, $F_{ij} = 4\,\Delta t\,h_{ij}(0)\,\dot{x}_j$
- Hydrostatic stiffness force
- Linear damping force
- Quadratic damping force
- Wind force (linear interpolation between directions)
- Linear and quadratic current drag force (linear interpolation between directions)
- Average wave drift force. The force will be calculated for the instantaneous heading the first and last time step. For other steps, corrections for instantaneous heading is done utilizing 1. derivatives.
- Small volume hydrodynamic force, wave particle velocities are set to zero.
- Coupling element forces
- Specified forces for t = 0.
- Positioning element forces
- Thruster forces.

If a dynamic positioning system is defined, the thruster forces are calculated from the thruster allocation algorithm described in the theory manual. The thrust demand is

$$F_{d,i} = \delta\,x_i\,G_{i1}\,F_{wi,i}\,F_{wa,i}\,F_{cu,i}\,F_{li,i}\,F_{co,i}\ \ ,i = 1,2,6 \qquad \textbf{(5.3)}$$

where

$\delta\,x_i$ - Positioning error in local coordinate system

$G_i 1$ - Position and heading gains in linear controller matrix

F_{wi} - Static wind force

F_{wa} - Static wave drift force

F_{cu} - Current force

F_{li} - Positioning element force

F_{co} - Coupling force

The equations of motion are stepped the specified number of steps with the modified Euler algorithm. At the last step, relative position and direction "errors" are calculated as:

$$\varepsilon_1 = \max\left[\left| \frac{\dot{x}_i}{\delta \dot{x}} \right|, \left| \frac{\ddot{x}_i}{\delta \ddot{x}} \right| \right], i = 1,2,3 \quad \textbf{(5.4)}$$

$$\varepsilon_2 = \max\left[\left| \frac{\dot{x}_i}{\delta \dot{\alpha}} \right|, \left| \frac{\ddot{x}_i}{\delta \ddot{\alpha}} \right| \right], i = 4,5,6 \quad \textbf{(5.5)}$$

"Equilibrium position" is found when both ε_1 and ε_2 are less than 1.

Note that initial positions are updated when calculation of equilibrium is selected, even if equilibrium position is not found.

It is advised to check the equilibrium position by calculating static forces and printing new initial positions.

5.6 Eigenvalue calculation

Natural periods and eigen vectors can are calculated for selected modes. The calculation is based on a linearized model of the system. Mass and stiffness matrices are calculated, and the system solved by a standard acobian solver. Effect of damping is not included in the analysis.

The mass matrix is built from the virtual mass matrix, including both structural and hydrodynamic mass. The frequency dependency of the latter is **not** accounted for in the analysis as asymptotic values are used. For bodies of type 1 (see appendix A, section 2: Body Data Specification), this means added mass at infinite frequency, while the added mass at zero frequency is used for bodies of type 2.

The secant stiffness for the user selected modes is calculated at the initial position. The user may choose to specify the secant excursion used in the calculation for each mode. The stiffness matrix is made symmetric.

For each natural period, the eigen vectors expresses the relative contribution from each mode. Translations are given in the length unit (L) and rotations in degrees (deg).

In the interpretation of the results, it should be kept in mind that damping is not included in the analysis. Modes that are heavily damped, may thereby not be recognized in time domain simulation results.

Observe also that environmental forces are not included. Thus, the yaw motion of a turret moored or a single point moored vessel can not be checked by an eigenvalue analysis.

5.7 Mooring system pretension optimization

By this option it is possible to adjust line lengths (or move anchor position) in an optimal way in order to counteract average environmental forces. The sum of squared differences between pretension and actual tension in each line is minimized.

5.8 Write initial condition file

After all modifications have been made with the system, the initial condition must be saved. The initial condition file will be read by DYNMOD. It can also be read by STAMOD if more modifications are required.

5.9 Write file for visualization

A file for 3D visualisation by the stand-alone program SimVis can be written whenever the system description file has been read successfully. This means that the system can be visualized before or after equilibrium condition calculation has been made. Visualisation directly after reading the system description file can be advantageus as a preliminar control of the system geometry.

6. USE OF DYNMOD

The purpose of DYNMOD is to perform time domain simulation of the system with initial condition as defined by INIFIL. Main analysis parameters, method parameters, storage parameters and integration parameters must be specified. The different options in the main menu must be selected in increasing order.

```
I->  DYNMOD main menu
I
I      1 : Read initial condition file
I      2 : Set simulation parameters
I      3 : Initialize simulation
I      4 : Time domain simulation
I     99 : Terminate
I
I->  Select option (  1 ) >
```

6.1 Read initial condition file

An INIFIL prepared by STAMOD will be read.

6.2 Set simulation parameters

Default values representing the "most common used" values are assigned to all parameters. Within each group default values can be selected for all parameters. Specification can thus be done with a minimum of keystrokes. Otherwise default values are obtained by pressing ``Return" during the specification.

6.2.1 Main analysis parameters

Motion mode for station-keeping forces

If any of the bodies included in the simulation is of type 2, i.e. the motion is separated in HF and LF motions, the station-keeping forces may be calculated due to LF response or due to the total HF+LF response, which is the default.

Wave generation method

The time series of wave responses are generated by superposition of harmonic components with uniformly distributed phases by means of pre-generation by the Fast Fourier transform (FFT) or by time domain summation of the harmonic components.

Due to these time-consuming calculations, a combination of pre-generated time series and cosine series in the time domain is made possible. For such a combination it is also possible to ensure that the same realization of wave components is used for both options for wave response calculation.

```
   Specification of calculation method for waves:
    ----------------------------------------------

    I->  Select method for calculation of waves
    I
    I      1 : No waves to be calculated
    I      2 : FFT only
    I      3 : FFT and Cosine series combined
    I      4 : Cosine series only
    I      5 : Visualization (cosine series)
 I->  Select option (  2 ):
```

Wind gust generation method

The time series of wind gust are generated either by superposition of harmonic components with uniformly distributed phases by means of pre-generation by the Fast Fourier transform (FFT) or by a time domain state-space model driven by white noise.

```
Specification of calculation method for wind gust:
 -------------------------------------------------

 I->  Select method for calculation of wind
 I
 I       1 : No wind gust to be calculated
 I       2 : FFT
 I       3 : State Space model
```

Random seed for waves and wind

A seed for generation of random phase angles must be specified. With different seed numbers, different time series of wind and waves will be generated.

It is also possible to read wind and waves from an ASCII file.

Each body may experience the same wind time series; alternatively different wind time series will be generated for each body.

Time series of wind force will be calculated for those bodies that have defined wind force coefficients.

FFT parameters, simulation length

If waves or dynamic wind responses are included in the simulation, time series can be generated by FFT. The number of time steps in the pre-generated series must be an integer power of 2. For example 2^{11} steps equals 2048 steps.

Wave generation by summation of harmonic components (cosine series) in the time domain

The time series generated (equvidistant frequencies), will repeat them-selves with a period $T_p = 2\pi / \Delta\omega_{k\min}$, where $\Delta\omega_{k\min}$ is the smallest frequency increment. For user-given number of wave components for wind sea and swell, it is important that the number of components is sufficiently high to avoid a repeating wave response time series for the longest duration of the simulated operation that is close to stationary.

Stochastic amplitudes may be specified to be used for time domain wave response generation. This is a conflict with the assumption of ergodic processes, and analyses will often require simulations with multiple wave realizations in order to be able to give reliable extreme value estimates.

Time step

The time step for the retardation functions is used as the default time step in the simulation. There is a possibility of subdividing each time step into sub steps. Sub steps will not be in stored on the time series file. Care should be used when using a large number of sub steps, particularly when retardation functions are specified.

6.2.2 Method parameters

Possible wave responses

- The following wave responses may be calculated:
- Wave elevation in origo
- First order motion

- Wave particle motion at body initial position for those bodies that have defined small body hydrodynamic forces.
- Wave particle motion for slender structures with distributed hydrodynamic forces
- 1st order wave forces
- 1st order diffracted wave kinematics at selected points around large volume bodies
- Wave drift forces
- Wave drift damping forces
- Full 2.order wave forces
- Ringing forces

Wave responses by FFT, corrections due to heading changes

The following possibilities exist:

- No correction, i.e. responses calculated for initial heading is used throughout the simulation.
- Interpolation in time domain. The response is initially calculated for a number of different headings, and interpolation between these is preformed in time domain. The maximum heading change compared to initial heading must be specified, and simulation stops if this is exceeded.
- For wave drift forces only: the first (and optionally also the second) derivative of average drift forces is applied to modify the pre-generated forces in time domain.

Calculation methods for wave drift forces

The following methods exist

- Average force will be calculated for each time step, calculation in the time domain
- Use initial average force, and make corrections in time domain using the derivative of average force, calculation by FFT
- Newman model, calculation by FFT or in the time domain (cosine series)
- Extend drift force coefficients to full quadratic transfer functions, by using the Newman approximation, calculation by FFT or in the time domain (cosine series)

Calculation methods for 2nd order forces (QTF)

Quadratic transfer functions for both sum and difference frequencies and optionally directional seas may be used as input. For difference frequencies the following methods exist.

- Using the frequency diagonal as wave drift force coefficients and apply the Newman model.
- Using the frequency diagonal and expand to off-diagonal terms by Newman's approximation.

Calculation of these forces in the time domain is not yet implemented.

Wave elevation

Normally, the resultant wave elevation is stored at the time series file. Alternatively, time series for each direction may be stored separately.

Wind velocities

Wind velocities will be calculated for the wind propagation direction, but also transverse gust speed may be specified:

```
Wind velocity:
--------------

I->  Wind velocity, body-id
I
I       1 : One dimensional
I       2 : Two dimensionaø
I
I->  Select option ( 2 ) :
```

Wind forces

Three force models are available:

- Calculation of static force (due to average wind velocity).
- Force due to total wind velocity
- Force due to relative wind velocity.

Interpolation of wind force coefficients between directions may be performed by either linear or spline interpolation.

Current forces

Two force models are available:

- Calculation of static force
- Force due to relative current velocity.

Interpolation of current force coefficients between directions may be performed by either linear or spline interpolation.

The methods to be selected apply for both linear and quadratic current force coefficients.

6.2.3 Storage parameters

The following results from time domain simulation can be stored on the time series file:

- Wind speed
- Wind force
- Wave elevation. For short-crested sea, either each wave component or for resultant wave may be stored.
- Wave particle velocities and accelerations, either for each directional wave component or for resultant wave.
- Diffracted wave elevation and wave particle velocities and accelerations
- 1^{st} order wave force for degrees of freedom where transfer functions have been read. Either for each directional wave component or for resultant wave.
- 2^{nd} order wave force for degrees of freedom where transfer functions are read. If 2^{nd} order wave forces are modified due to heading changes, the force on the file will be updated, and the original value is lost.
- Total force.
- Retardation force.
- Hydrostatic stiffness force.
- Linear damping force.

- Quadratic damping force.
- Linear current drag force.
- Quadratic current drag force.
- Small body hydrodynamic force.
- Total force from all elements of a body. In addition, the total force and the components of each element can be stored.
- Total force from all couplings of a body. In addition, the total force and components in global and local system can be stored for each coupling element.
- Total force from all thrusters of a body. In addition, force and direction of individual thrusters can be stored.
- Dynamic positioning estimators.
- Specified forces.
- Global positions.
- Global accelerations.
- Local velocities.

6.3 Initialization of time domain simulation

Time series of wind speed, wave elevation, wave particle motion, 1st order wave forces and 2nd order wave forces will be generated at the initial positions of the bodies.

If regular waves are specified, these are also generated in the same way as irregular waves. The frequencies of the harmonic components are multiples of the frequency increment, $\Delta\omega$.

$$\Delta\omega = \frac{2\pi}{N_t \Delta t} \qquad (6.1)$$

where N_t is the number of time steps and Δt is the time increment. The obtained wave periods will normally be slightly different from the specified ones since the selected wave frequency is a multiplum of $\Delta\omega$.

If wave periods are carefully selected, the specified wave periods will be obtained. The condition for this is that the specified simulation duration is an integer number of wave periods:

$$\frac{\omega}{\Delta\omega} = \frac{N_t \Delta t}{T} \quad \text{is an integer} \qquad (6.2)$$

where T is the specified wave period.

6.3.1 Reading wind time series from file

Optionally, it is possible to read wind velocity or wind velocity and direction (for each body) from an ASCII file.

```
Read wind series at body-id from file? (N)          : YES

Name of wind-file
File name may be DROP

File name
        ---------------------------------------------
Old    (wind.asc                                     )
New   :

Scaling factor:
```

In case one dimensional wind velocity has been specified, the input file should contain the time series of wind velocity only. The propagation wind direction for the present wind case is used (chosen in STAMOD, specified on SYSFIL). If two-dimensional wind velocity, the input file must contain series of both wind velocity and wind direction (in degrees). In this case the read vind time series is transformed to the propagation direction and transverse direction of the present wind case (chosen in STAMOD, specified on SYSFIL). This will not have any influence the results from the simulation. It will only have an effect on the stored wind time series, stored as components in propagation and transverse directions.

Both the read wind velocity and time step will be scaled by the square root of the scaling factor.

The time step given on the ASCII file may be different from the time step used in the simulation. The specified wind time series will be re-sampled using the time step applied in the simulation basefd on linear interprolation. Further the velocities will be scaled by the square root of the scaling factor.

If the read time series is shorter than the length of the simulation, the wind velocity in the last part of the simulation will be set to zero.

It is assumed that the reference height for wind velocity coincide with the height for which wind force coeficients are given. Further, no filtering (application of admittance function) is done on the read wind times series.

File format (ASCII) for one dimensioal wind velocity

Line 1	:	Number of samples
Line2	:	Time step
Line3	:	Arbitrary text
Line4	:	Arbitrary text
Lines5 – N	:	Time series read in free format (arbitrary number of samples on each line)

Example:

```
 2048       ' Number of samples
 .50        ' Time step
Text 1
Text 2
     20.0000
     20.3923
     20.7822          21.1672
     21.5451          21.9134
  .
  .
```

File format (ASCII) for two-dimensioal wind velocity

File head: (*The file head may contain an arbitrary number of comment lines starting with:* ')

Line	1	:	Number of samples	
Line	2	:	Time step	
Line	3	:	Number of columns (=2)	

Time series:

Line	4	:	velocity	direction
Line	5	:	velocity	direction
.				
.				
.				
Line 3 + N		:	velocity	direction

Example

```
  2048       ' Number of samples \
´ Comment line 1                  \
  .50        ' Time step           \
  2          ' Format type       Head of file
´ Comment line 2                   /
´ Comment line 3                  /
´ Last comment line              /
  20.0000      180.000
  20.3923      180.500
  20.7822      181.000
  21.1672      181.500
  21.5451      182.000
   .
   .
```

6.3.2 Reading wave elevation from file.

Provided that lon-crested wave is specified, it is possible to read wave elavation time series from an ASCII file:

```
Should wave time series be read from fileat body-id from file? (N)           : YES

Name of wave-file
File name may be DROP

File name
        --------------------------------------------
Old    (wave.asc                                    )
New   :

Scaling factor:                   (1.0000) :
```

```
X and Y coord. where wave is given (full scale).

X-coord. (wave)                                  ( 0.00000 ) :
Y-coord. (wave)                                  ( 0.00000 ) :

Water depth in FULL scale                         ( 1000.00 ) :

Cut-off periode for HP filter of wave signal     ( 40.0000 ) :
```

The propagation wave direction for the present sea state is used (chosen in STAMOD, specified on SYSFIL).

The read wave elevation will be scaled by the scaling factor while the time step is scaled by the square root of the scaling factor.

The time step given on the ASCII file may be different from the time step used in the simulation. The specified wave elevation time series will be re-sampled according the time step used in the simulation baseed on linear inteprolation. If the read time series is shorter than the length of the simulation, the wave elevation in the last part of the simulation will be set to zero.

File format (ASCII) for wave elevation

Line	1	:	Number of samples
Line	2	:	Time step
Line	3	:	Arbitrary text
Line	4	:	Arbitrary text
Lines	5 – N	:	Time series read in free format (arbitrary number of samples on each line)

Example:

```
 2048       ' Number of samples
 .50        ' Time step
Text 1
Text 2
     20.0000
     20.3923
     20.7822          21.1672
     21.5451          21.9134
   .
   .
```

A complete example from a simulation is given below.

```
Should wave time series be read from file ? (N)                  : y

 Name of wave-file
 File name may be DROP

 File name
         ----------------------------------------
 Old    (wave.asc                                )
 New   : wave.asc

 Scale factor                        (     1.00000      ) :

 Time step in model scale is 0.0200 sec.
 Each series consists of  131072 points.

 Wave signal in full scale contains    2048 values
 different from zero.

 The wave is transformed to ORIGO according to wave direction
 and coord. of the point where the wave is given.
 Transf. dist. is in FULL scale.
 Deep water is NOT assumed : w*w = g*k*tanh(k*d)
 The wave read from file replaces the wave
 specified on the system input file.
 The wave must be long-crested.
 Wave dir. is taken from the system input file.

 X and Y coord. where wave is given (full scale).
                                     :
 X-coord. (wave)                     (     0.00000      ) : 10
 Y-coord. (wave)                     (     0.00000      ) : 20
 Wave direction   45.00 deg. results in a
 transformation distance of -21.21 m

 Water depth in FULL scale           (     100.000      ) : 200

 Cut-off period for HP filter of wave signal
                                     (     40.0000      ) : 100
```

6.4 Time domain simulation

The simulation time and the integration method must be specified.

Clutch on 1st order wave forces is implemented. The first and second derivatives of the clutch function are zero in both ends. The duration of the clutch function must be specified.

6.5 Work array size in DYNMOD

The size of the DYNMOD work arrays may be specified using the environment variable SIMO_MEM. The variables give the size in MB, i.e. 4 times the number of million words. The minimum size is 4 MB and the maximum size is 800 MB.

The default size of the DYNMOD work array is 4 million words, e.g. 16 MB.

7. USE OF OUTMOD

The purpose of OUTMOD is to prepare plots of static system geometry and to analyse and present results from the time domain simulation.

```
I->  OUTMOD main menu
I
I       1 : Define input and output files
I       2 : List stored data
I       3 : Environmental time series
I       4 : Force time series
I       5 : Position time series
I       6 : Snap shot of bodies
I      99 : Terminate
I
I->  Select option (  1 ) >
```

7.1 Define input and output files

It is possible to use OUTMOD for generation of static geometry only. In that case, the input files will be the initial condition file, INIFIL, and the system geometry file, SYSFIL.

If results from time domain simulation are requested, the input files will be PREFIL, the time series file, TSFIL, and the initial condition file, INIFIL. In this case, the names of TSFIL and INIFIL will be read from PREFIL and can not be changed by the user. If snapshots or static system geometry are requested, SYSFIL must be read as well.

Note that SYSFIL is used in OUTMOD only to read body geometry data. It will be favourable to use a reduced SYSFIL as described in Appendix A.

Results from OUTMOD will be written to the print file. If time series are wanted as output, the time series will normally be written. This can be avoided by decreasing the print switch ``@ FILE".

Plot information will be written to the plot file, PLOFIL. It is possible to append more plots to a PLOFIL generated in a previous OUTMOD run. Plotting and scaling of plots is done by a separate program, PLOMOD.

7.2 List stored data

A summary of channels stored on TSFIL will be presented on the terminal.

7.3 Time series analysis

Any part of the time series can be selected for processing. Position time series can be derivated twice before processing.

7.3.1 Time series

Average value, standard deviation and extreme values are written to the print file and PLOFIL. The time series is written to PLOFIL and may also be written to the print file. It is also possible to preview a time series plot in OUTMOD.

The plots might appear with response type numbers. The following possibilities exist:

1	Wind velocity
2	Wave elevation

3	Wave particle motion at initial position
4	Wave particle motion for distributed hydrodynamic forces
7	Total force
8	Retardation function force (convolution integral)
9	Stiffness force
10	Linear damping force
11	Quadratic damping force
12	1st order wave force
13	Wind force
14	Linear current force
15	Quadratic current force
17	2nd order wave force (wave drift force)
18	QTF (quadratic transfer function) force
19	Small body hydrodynamic force
20	Positioning system force
21	Coupling system force
23	Dynamic positioning parameters
24	Specified force
25	External force
26	Thruster force
27	Distributed hydrodynamic force
28	Wave drift damping force (wave current interaction)
29	Global position from time domain calculations
30	Global acceleration
31	Local position
32	Local velocity
33	Global (total) position

7.3.2 Statistical analysis (Time series statistics)

Input to the statistical analysis is the time series of the response to be analysed. Additional time series for maxima and minima are derived. For all three series, the four first statistical moments are computed:

$$m_i = \frac{1}{n}\sum_{j=1}^{n} x_j^i \qquad (7.1)$$

The results are presented as average value, μ, standard deviation, σ, and coefficients for skewness and excess, γ_1 and γ_2.

$$\mu = m_1 \qquad (7.2)$$

$$\sigma^2 = m_2 - m_1^2 \qquad (7.3)$$

$$\gamma_1 = (m_3 - 3m_1 m_2 + 2m_1^3)/\sigma^3 \qquad (7.4)$$

$$\gamma_2 = (m_4 - 3\, m_2^2 - 4\, m_1 m_3 + 12\, m_1^2 m_2 - 6\, m_1^4 - 3)/\ \sigma^4 \qquad (7.5)$$

For a Gaussian process γ_1 and γ_2 are zero, hence these two parameters describe the deviation from Gaussian distribution. An interpretation of these parameters is shown in Figure 7.1

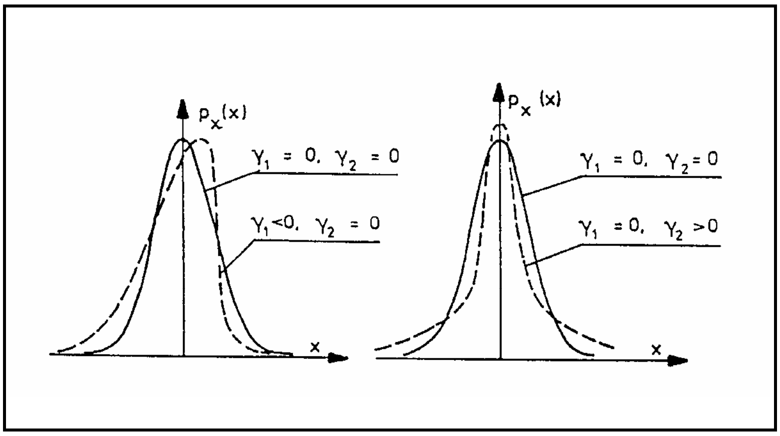

Figure 7.1 Interpretation of skewness and excess coefficients

The distribution densities and cumulative distributions are computed based on simple ranking into a set of classes defined by the user in terms of:

- Number of classes.
- Lower range defined by number of standard deviations below mean value.
- Upper range defined by number of standard deviations above mean value.

The range refers to the initial distribution. Thus the distributions of maxima and minima are computed for the same class discretization as the initial distribution.

No output is presented on PLOFIL.

7.3.3 Spectral analysis

The spectrum estimate is based on frequency averaging ('smoothing') of hamonic components obtained by FFT.

The "Standard deviation", the "Average period" and the "Zero crossing period" are calculated. When the spectral density is given as a function of the frequency in (rad/s), the defintion of these parameters are:

Standard deviation = sqrt (M0)

Average period = T01 = 2*pi * M0 / M1

Zero crossing period = T02 = 2*pi * sqrt (M0 / M2)

M0, M1 and M2 are the zero, first and second order moment of the spectral density.

Spectral moments, spectrum parameters and spectral values are presented on the print file. In addition, spectrum values are written to PLOFIL.

Extreme values are estimated by fitting a Rice distribution to the sample and calculating the expected largest within the time interval to be specified by the user.

7.4 Snap shot of bodies

Both static system geometry and snapshots from dynamic simulation can be generated on PLOFIL. Projections into any of the planes defined by the axes of the global coordinate system can be selected. In addition, 3-dimensional plots can be generated. Rotations of the 3-D plots are specified in PLOMOD.

Wave elevation can be included in the plots. In PLOMOD, the snap shots can be animated.

For 3D plots, results can be written to files for later visualization by a separate program, GLview (not a part of the SIMO program system). The files generated are

ani.geo	-	containing the geometry
ani.txt	-	containing displacements on ascii form.

The latter should be transformed to binary form by the command:

```
vec2one    ani.txt    ani.dis
```

8. USE OF S2XMOD

The main purpose of S2XMOD is to export time series to other file formats than applied by SIMO-DYNMOD. The options provided by S2XMOD are:

- Give an overview of all series generated by SIMO
- Produce statistics of series
- Plot series
- Writes selected series to MATLAB "m"-file format
- Writes selected series to a direct access file

```
I->  S2XMOD main menu
I
I       1 : Time series info
I       2 : List/modify default settings
I       3 :  Statistics of series
I       4 :  Plot of series
I       5 :  Statistics and plot of series
I       6 : EXPORT to one MATLAB "m"-file
I       7 : EXPORT to separate ASCII files + "m"-file
I       8 : EXPORT to DIRECT ACCESS file
I      99 : Terminate
I
I->  Select option (  3 ) :
```

8.1 Time series info

A summary of channels stored on TSFIL will be presented on the terminal.

8.2 List/modify default settings

List and modify default setting for export of time series, statistics and plot:

- Limit for plotting
- Amount of statistics
  - Minimum, maximum, mean
  - Minimum, maximum, mean, skewness, kurtosis
- How to specify time series
  - Specified by: 1) array number or 2) body number, response number and channel
- Number of time steps to be skipped at start and end of time series
- Unit on x-axis for plotting (time or sample number)

8.3 Statistics of series

Gives simple statistics of one or several time series

8.4 Plot of series

Present plot of one or several time series

8.5 Statistics and plot of series

Present statistics and plot of one or several time series

8.6 EXPORT to one MATLAB "m"-file

Export one or several time series to MATLAB .m-file

8.7 EXPORT to separate ASCII files + one MATLAB "m"-file

Export one or several time series to separate ascii files, while information is stored in the MATLAB .m-file

8.8 EXPORT to DIRECT ACCESS file

Export one or several time series to direct access file. Information about the file is written to an ascii- file.

9. SYSTEM DESCRIPTION FILE

The SYSTEM DESCRIPTION FILE is an ASCII file that contains a complete description of the system.

9.1 System description file layout

The input file is read by the free format reader and decoder FREAD.

Each set of data card or data card group is identified with a character string consisting of maximum 3 words. These are identified by up to maximum four characters in each word.

Comment lines in the input set are obtained by giving an accent (') in the fist column.

Data items in a card image are separated by ("space"). Default values of data items are obtained by giving a slash (/).

For missing data items, the default value is used.

Continuation of the card image is obtained by (&).

All numbers given in the input files can be read in free formats, both integer and real numbers. Data types are specified for each input parameter, where (I) means integer type, (R) means real type, (A) means character or hollerith type.

Some parameters may have default values. The default values are obtained simply by giving a slash (/) instead of the input value.

The input data are defined by the following input description frames:

symbolic names of data items			
item	*description*	*format*	*default*
item	*description*	*format*	*default*
Note	*note*		

- *symbolic names of data items* appear in the order they are decoded
- *item* is repeated for all items in the heading
- *description* is used for explaining the parameter
- *format* is either I,R or A
- *default* value may be indicated (if relevant)
- *note* may be used for comments, etc.

The system description file is divided into 5 main data groups:

-	**SYSTEM DESCRIPTION SIMO**	Identifies the file type, always the first data group
-	**HLA COMMUNICATION SPECIFICATION**	Needed only for real-time simulation
-	**BODY DATA SPECIFICATION**	The data group to be repeated for all bodies
-	**HYDRODYNAMIC COUPLING DATA**	The data group to be repeated for each coupling between 2 bodies
-	**GENERAL LINE DATA**	Not to be repeated
-	**COUPLING DATA**	The data group to be repeated for all couplings
-	**ENVIRONMENT DATA SPECIFICATION**	Not to be repeated, environmental conditions must be specified together
-	**END**	Indicates end of the file.

All data within a main data group e.g. BODY DATA SPECifictation, must be given in one sequence. However the different data groups within a main data group may be given in an arbitrary sequence.