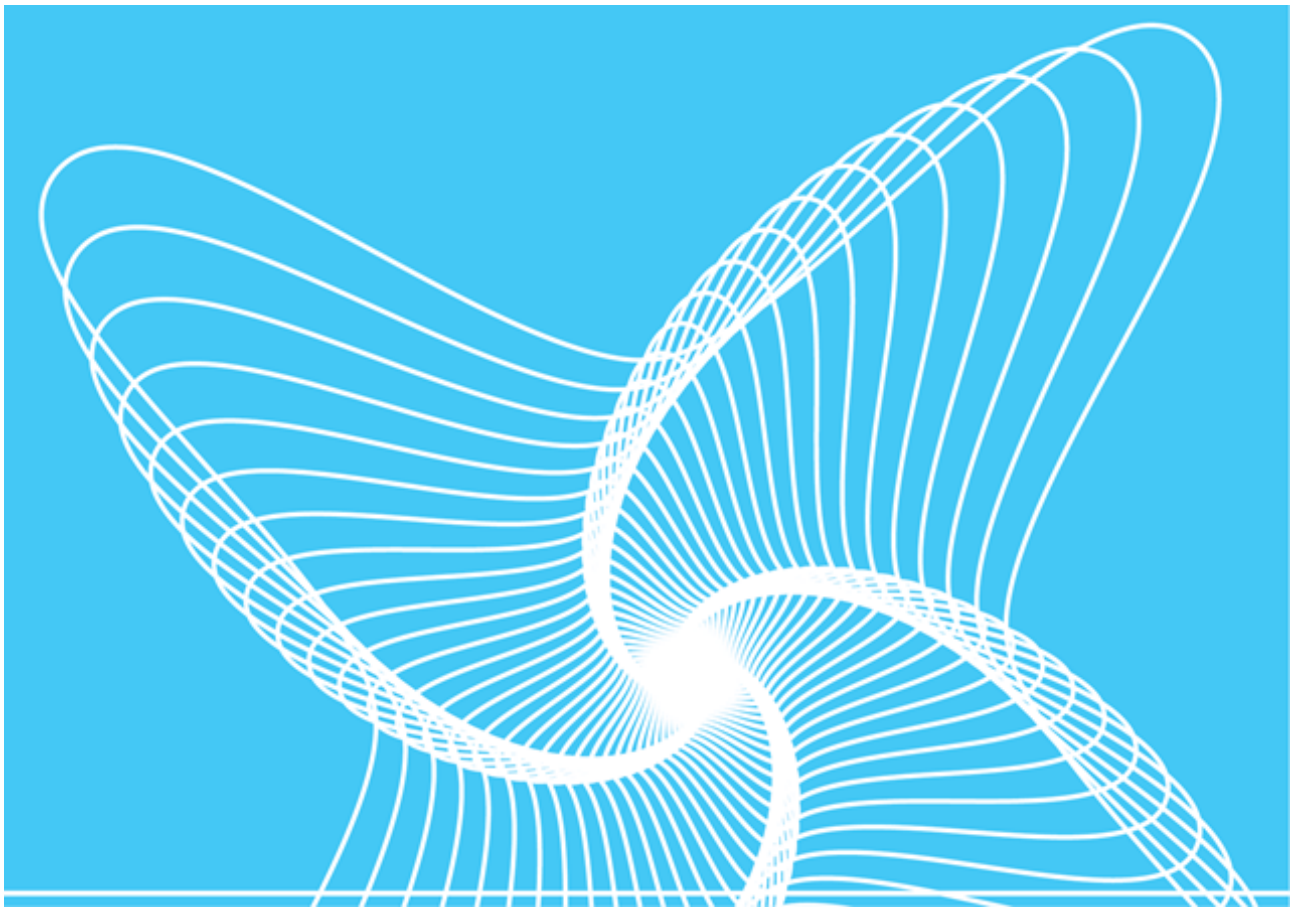
DNV·GL

SESAM USER MANUAL

Prepost

Sesam Utility Program for Data Conversion

Valid from program version 8.4



SAFER, SMARTER, GREENER

Sesam User Manual

Prepost

Date: 17 November 2017

Valid from program version 8.4

Prepared by DNV GL - Software

E-mail support: software.support@dnvgl.com

E-mail sales: software@dnvgl.com

Table of contents

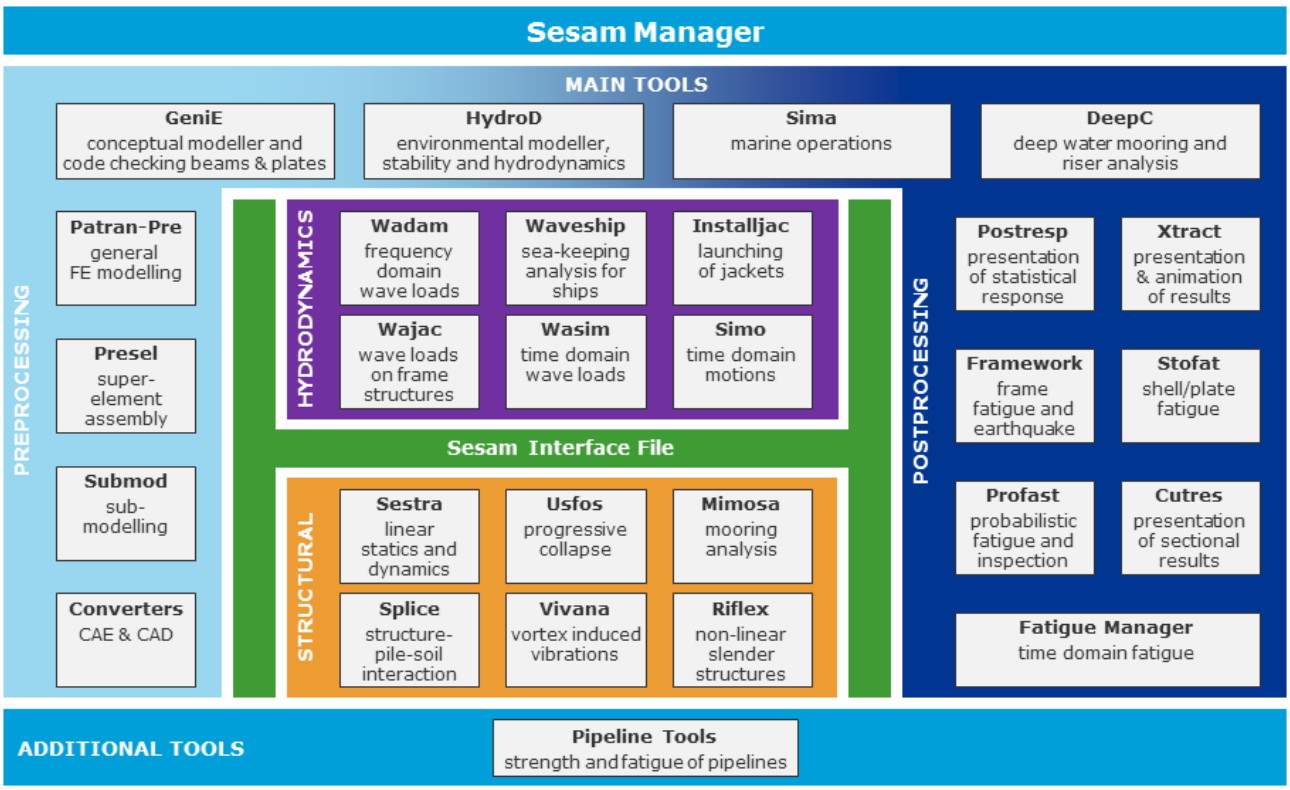


Figure 1.1: Sesam overview

1 Introduction

1.1 Prepost – Utility Program for Data Conversion

Prepost is Sesam's utility program for data conversion. The program offers:

- Conversion between different formats (SIF, SIU and SIN) of the Results Interface File
- Extracting data from the Results Interface Files and creating a Hydrodynamic Results Interface File
- Merging and copying data between Results Interface Files
- Creating combinations of result cases on Results Interface File
- Import of finite element data from other systems to Sesam
- Export of finite element data from Sesam to other systems

Prepost has an interactive user interface but no graphic features.

1.2 Prepost in the Sesam System

Sesam is comprised of preprocessors, environmental analysis programs, structural analysis programs and postprocessors. An overview of Sesam is shown in Figure 1.1.

Prepost, which is not shown in the overview, operates on the various elements of the Sesam Interface File.

1.3 How to read the Manual

Section 2: Features of Prepost contains descriptions of important program concepts and features. The new user should read this section first.

Section 3: User's guide to Prepost contains practical examples illustrating the use of the features explained in Section 2.

Section 4: Execution of Prepost contains practical information on how to start the program. It also describes size-related limitations within Prepost.

Section 5: Command description provides a description of the Prepost commands.

1.4 Status List

There exists for Prepost as for all other Sesam programs, a Status List providing additional information. This may be:

- Reasons for update (new version)
- New features
- Errors found and corrected
- Etc.

To look up information in the most updated version of the Status List go to the support page of our website, click the Sesam Status Lists link and log into this service. Contact us for log-in information.

1.5 Prepost Extensions

Access to a program extension is subject to agreement and a proper license file. Prepost includes the following extensions, each corresponding to a command as listed below:

ANSS	READ ANSYS	...	Ansys to Sesam translation
SANS	WRITE ANSYS	...	Sesam to Ansys translation
SACS	CONVERT SACS-TO-FEM	...	SACS to Sesam translation
INTS	READ INTERGRAPH	...	Intergraph to Sesam translation
SINT	WRITE INTERGRAPH	...	Sesam to Intergraph translation

The extension INTS and SINT are not actively maintained, but provided "as-is" to support users with existing files in the formats. In addition Prepost used to have extension called PATS and SPAT for handling Patran files, they are obsolete.

2 Features of Prepost

2.1 Sesam Interface File

A Sesam Interface File is an Input Interface File, containing the finite element model and the loads, or a Results Interface File, containing results. Most of the content of an Input Interface File is copied into the Results Interface File as part of an analysis in Sestra.

A construction can be modelled as one structure or several sub-parts, also called-sub structures, which is assembled into one model. In Sesam the sub-structures are called superelements. Each superelement is one Input Interface File. It is a formatted file and the file name has the pattern:

```
<prefix>T<superelement number>.FEM
```

where the superelement number is an integer. The loads can be in this file or on a Loads Interface File. Here the file name has the pattern:

```
<prefix>L<superelement number>.FEM
```

The Results Interface File, often referred to as the SIN file, uses the file name pattern:

```
<prefix>R<superelement number>.SIN
```

The Result Interface File has three different formats, all with the same content. The format is reflected by the file extension:

- SIF extension: sequential formatted file
- SIU extension: sequential unformatted file
- SIN extension: database format

For lower level result files with SIF extension the filenames uses the pattern:

```
<prefix>R<superelement number>H<file reference>.SIF
```

Finally, there is the Hydrodynamic Results Interface File:

```
<prefix>G<superelement number>.SIF
```

2.2 READ and WRITE

The READ and WRITE commands operate on Sesam Interface Files.

The OPEN command opens a Results Interface File with extension SIN. The command also creates a new and empty SIN file ready for data to be read into it. The READ command reads files with extensions SIF and SIU into an empty SIN file. The WRITE command writes a SIN file to a SIF or a SIU file.

2.3 MERGE and COPY

The same model may be analysed in several Sestra runs, resulting in multiple Results Interface Files. Prepost provides two facilities for combining such analyses into one Results Interface Files.

The MERGE facility operates on two Results Interface Files by appending results of the second to the first. This requires the first file to be SIN and the second to be SIF or SIU.

The COPY facility is used to select results from one analysis and either append or overwrite the selected data to the second analysis. It may for example be used to "repair" analyses containing some erroneous results by re-running only selected loadcases. The COPY facility allows renumbering resultcases as they are copied.

2.4 Result combination

This facility allows creating result combinations on the Results Interface File. The facility also includes features for transforming complex, i.e. real and imaginary data, into real data and vice versa.

2.5 Ansys conversion

This facility is for converting Ansys models to Sesam models and vice versa. It supports a subset of commands in the Ansys .cdb files.

2.6 Sacs conversion

This is used to convert SACS data into a Sesam Input Interface File. It contains geometry, material and physical properties, as well as nodal and element loads.

In addition, the hydrodynamic loading data is written to a Wajac input file. The input file may not be complete and may require additional data from the user.

2.7 Intergraph translation

This facility provides a two-way data communication tool between the Intergraph neutral file and the Sesam Input Interface File and the Sesam Results Interface File.

2.8 Patran conversion

This facility was used to convert from Patran to Sesam and vice versa. This facility is obsolete.

2.9 Hydrodynamic interface

Provided that the Results Interface File contains complex (real and imaginary) results from a frequency domain analysis the CREATE HYDRODYNAMIC-INTERFACE command can be used to extract results for selected positions/points in selected elements and transfer these to a Hydrodynamic Results Interface File.

3 User's guide to Prepost

3.1 Sesam Interface File facilities

Reformatting Sesam Interface Files

Below is shown which Prepost commands to use for converting Sesam Interface Files from one format to another.

Input format	Output format	Prepost commands
SIF	SIU	READ SIF ... WRITE SIU ...
SIU	SIF	READ SIU ... WRITE SIF ...
SIF/SIU	SIN	OPEN SIN ... NEW READ SIF/SIU ...
SIN	SIF/SIU	OPEN SIN ... OLD WRITE SIF/SIU ...

Using the MERGE facility

The principle of the MERGE process is to merge SIF/SIU files into a SIN file, as shown in case 1. If none of the files is SIN, then an empty SIN file is created and SIF/SIU files are read and merged into it as shown in case 2.

Case	Prepost commands	
1	OPEN SIN ... OLD WRITE-PERMITTED	Run 1
	MERGE SIU ...	Run 2
	MERGE SIF ...	Run 3
	EXIT	
2	OPEN SIN ... NEW	
	READ SIF ...	Run 1
	MERGE SIF ...	Run 2
	EXIT	

Using the COPY facility

The COPY facility allows copying model and/or results data from one SIN-file to another. This is useful for the case when a large analysis contains some erroneous results. Assume for instance the following situation:

- Original analysis, results file is `OriginalR1.SIN`.
  A model with superelement number 1 and 50 static resultcases has been analysed. Resultcase 23 and 44 must be re-calculated.
- Correction analysis, results file is `CorrectionR1.SIN`.
  A new analysis containing only cases 23 and 44 is performed. The loads are numbered 1 and 2, respectively, in the correction analysis.
- Integrating correction analysis into the original analysis, using COPY.
  The COPY facility is used to replace the unwanted results of the original analysis by reading data from the correction analysis.

  The following Prepost commands are used. Exclamation mark is used for comments.

```
OPEN SIN-DIRECT-ACCESS ORIGINAL R1 OLD WRITE-PERMITTED
!
! Define the COPY source file.
SET COPY SOURCE-FILE CORRECTION R1 END
!
! Release protection of the erroneous resultcases.
```

```
! Each resultcase is identified by two numbers, case number and run number.
! Run number is normally 1. The third number is 0 and not used in static analysis.
SET COPY PROTECTION 23 1 0 UNPROTECTED
                    44 1 0 UNPROTECTED
                    END END
!
! Define source & destination resultcases.
SET COPY RESULTCASES 1 1 0 23 1 0
                     2 1 0 44 1 0
                     END END
! Copy results.
COPY ALL RESULTS
EXIT
```

Resultcase identification

A resultcase is identified in Sesam by three numbers:

- Resultcase number
- Run number
- Occurrence number

The resultcase number is normally the same as the loadcase number defined for the model. The run number is normally 1 but when merging an analysis into another the added resultcases will be given run number 2. The occurrence number is not used for static analysis results and the value 0 is entered in commands. In a frequency domain analysis the occurrence number identifies the various frequencies of the same wave direction. In a time domain analysis the occurrence number identifies the time steps. For further details see Resultcase identification.

Creating combination resultcases

The table below shows commands for creating combined resultcases. To do so a SIN file must be opened with permission to write on it.

Case	Prepost command used
1	CREATE RESULT-COMBINATION 11 2 REAL 1 1 0 1.5 0.0 2 1 0 2.0 0.0 3 1 0 2.5 0.0 END END
2	CREATE RESULT-COMBINATION 12 2 COMPLEX 1 1 0 1.0 0.0 2 1 0 1.0 90.0 END END
3	CREATE RESULT-COMBINATION 13 2 REAL 1 1 0 1.5 0.0 6 1 1 2.0 45.0 END END

In case 1, resultcase 11 with run number 2 is defined as a combination of cases 1, 2 and 3 with factors 1.5, 2.0 and 2.5, respectively. The third value (0) and last value (0.0) are occurrence and phase shift, and are dummy in this case.

In case 2, resultcase 12 with run number 2 is defined as a complex combination using resultcase factors 1.0 and 1.0, phase shifts 0.0 and 90.0 degrees, of static resultcases 1 and 2 respectively. In this way, two static

resultcases are merged into one complex resultcase where the first static resultcase becomes the real part and the other the imaginary part.

In case 3, resultcase 13 with run number 2 is defined as a real combination of the real resultcase 1 and the complex resultcase 6, occurrence 1, using the factors 1.5 and no phase shift, and 2.0 and phase shift equal 45 degrees. In this way, real and complex results are combined by selecting a phase angle for the complex results.

NOTE:

The command CREATE RESULT-COMBINATION does not perform the actual combination. Only the combination definition is stored. The actual combination is performed in the preprocessor, i.e. in Xtract, Framework, etc.

3.2 Sesam data import/export facilities

Sacs to Sesam conversion

The output from Sacs is converted to a Sesam finite element model and a Wajac input file:

```
CONVERT SACS-TO-FEM name METER NEWTON prefix 1
```

For further information see the CONVERT command.

Ansys to Sesam translation

The output from Ansys is one or more files. Typically, one cdb file containing model data and one or more load step files with load data. Note that the load files must be .s01, .s02, etc. The Ansys filer are read using the READ ANSYS command. Below is an example of translating an Ansys model without load files into a Sesam Input Interface File:

```
READ ANSYS prefix name 0 1
WRITE FEM-FORMATTED prefix name
```

Sesam to Ansys translation

The commands below are used to read a Sesam Input Interface File and write a Ansys cdb file:

```
READ FEM-FORMATTED prefix name ALL
WRITE ANSYS prefix name 1 1
```

For further information see the WRITE ANSYS command.

Intergraph to Sesam translation

An Intergraph neutral file is read using the READ INTERGRAPH command. The commands below are an example of translating an Intergraph neutral file into a Sesam Input Interface File.

```
READ INTERGRAPH MODEL- T12 12 DIRECT SPECIFIED
   GROUP 1 20 1 NORMAL MID
   GROUP 21 22 1 TEMPERATURE
   END
```

These commands translate loadcases 1 to 20 as node and element loads, and 21 and 22 as temperature loads.

Note that to increase efficiency, the GROUP instead of the ALL option should be used. The same Intergraph loadcase may be referred to several times.

The superelement created may be written as a Sesam Input Interface File using the command:

```
WRITE FEM-FORMATTED MODEL_ TI2 1 END
```

This file will have the name `MODEL_T12.FEM`. The Sesam superelement number is 12.

Sesam to Intergraph translation

The commands below are an example of translating a Sesam Input Interface File into an Intergraph neutral file.

```
READ FEM-FORMATTED MODEL_ T12 1 END
WRITE INTERGRAPH MODEL_ T12 1 0 END
```

This is based on the following assumptions/limitations:

1. The WRITE INTERGRAPH command assumes that no additional nodes shall be created at beam element ends (last command parameter).
2. Note the limitations on load translation listed under the WRITE INTERGRAPH command in Chapter 5.

4 Execution of Prepost

The preferred way of running Prepost is through Sesam Manager.

4.1 Program environment

Prepost produces two files, a journal file logging the given commands, and a file logging the execution of Prepost. The former is named `PREPOST.JNL` and is frequently referred to as the journal file, the latter is named `PREPOST.MLG`. In a successful run of Prepost the last line in `PREPOST.MLG` should say `EXIT`. The journal file can be used as an input file in a later execution of Prepost.

Starting Prepost from a Command Window

Prepost may be started in graphics mode using the command:

```
<path>prepost.exe /STATUS=NEW
```

The command `/STATUS=NEW` makes a new journal file. If `/STATUS=OLD`, or the command is absent, and a journal file is found the new commands are appended to the file.

An input file, e.g. `commands_in.jnl`, can be read into a batch mode execution by:

```
<path>prepost.exe /STATUS=NEW /COMMAND=commands_in.jnl /INTERFACE=LINE /FORCED-EXIT
```

Here `/INTERFACE=LINE` runs Prepost without the user interface and `/FORCED-EXIT` ends Prepost when all commands in the input file have been executed.

Note that commands can be abbreviated if they are unique, e.g. `/STA=NE` instead of `/STATUS=NEW`.

5 Command description

The hierarchical structure of the commands and numerical data is documented in this chapter by use of tables. How to interpret these tables is explained below. Examples are used to illustrate how the command structure may diverge into multiple choices and converge to a single choice.

In the example below command A is followed by either of the commands B and C. Thereafter command D is given. Legal alternatives are, therefore, A B D and A C D.

A	B C	D

In the example below command A is followed by three selections of either of commands B and C as indicated by *3. For example: A B B B, or: A B B C, or A C B C, etc.

A	B C	*3

In the example below the three dots in the left-most column indicate that the command sequence is a continuation of a preceding command sequence. The single asterisk indicate that B and C may be given any number of times. Conclude this sequence by the command END. The three dots in the right-most column indicate that the command sequence is to be continued by another command sequence.

. . .	A	B C	*	. . .
		END		

In the example below command A is followed by any number of repetitions of either of the sequences B D and C D. Note that a pair of braces ({ }) is used here merely to define a sequence that may be repeated. The braces are not commands themselves.

A	{	B C	D	}

The characters A, B, C and D in the examples above represent parameters being COMMANDS (written in upper case) and numbers (written in lower case). All numbers may be entered as real or integer values. Brackets ([ ]) are used to enclose optional parameters.

Note: The command END is generally used to end repetitive entering of data. Using double dot (..) rather than END to terminate a command will, depending on at which level in the command it is given, save or discard the data entered. Generally, if the data entered up to the double dot is complete and self-contained the double dot will save the data. If in doubt, it is always safest to leave a command by entering the required number of END commands.

Alphabetic List of Commands

CHANGE	Change the numbering of the load cases.
CONVERT	Convert data from Sacs and StruCAD*3D to Sesam Input Interface File.
COPY	Copy data from one SIN-file to another.
CREATE	Process results in the Results Interface File.
EXIT	Terminate a Prepost session.
HELP	Print help in message window.
MERGE	Merge results from different analyses.
OPEN	Open a SIN database.
PRINT	Print result file information to the message window.
READ	Read file(s) into the Prepost working file.
SET	Set defaults.
TRANSFORM	Transform stresses to a local coordinate system.
WRITE	Write the current Prepost working file to a file.
#	Execute the given number of commands from a command input file.
`<select superelements>`	Sub-command for selecting superelements.

CHANGE

CHANGE	DUPLICATE-RDSTRESS		
	RESULTCASE-NAME	<select superelements>	. . .
	RESULTCASE-NUMBER		

. . .	{	<source-resultcase>	<destin-resultcase>	}
	END			

PURPOSE:
The command changes the numbering of result cases.

PARAMETERS:

DUPLICATE-RDSTRESS	This option is obsolete.
RESULTCASE-NAME	Change the names of result cases
RESULTCASE-NUMBER	Change the numbering of result cases
<select superelements>	Select superelements, see select superelements for a description.
<source-resultcase>	Unique identification of an existing result case. The identification consists of three or four parameters as detailed for command SET COPY RESULTCASES
<destin-resultcase>	Unique identification of the new result case number. The identification consists of three or four parameters as detailed for command SET COPY RESULTCASES

See Also:

select superelements

SET COPY RESULTCASES

CONVERT

CONVERT	SACS-TO-FEM STRUCAD-TO-FEM	inp-fil	length-unit	force-unit	prefix	selno

PURPOSE:
The command converts data from SACS and StruCAD*3D to Sesam's Input Interface File.

Note: The converter is one way from SACS/StruCAD*3D to Sesam only.

PARAMETERS:

SACS-TO-FEM	Convert SACS data into an Input Interface File, i.e. a T*.FEM file, containing all principal structural data: geometry, material and physical properties as well as nodal and element loads, Pile, soil and weight is not converted. Note: Do not mix Metric and English units when selecting output units. In addition to the T*.FEM a Wajac analysis control data (input) file is created containing key hydrodynamic loading data like wave and current specifications and flooded members. The Wajac input file is a template only containing main sea data found on the SACS input file (SPEC, FLOO, CRNT and SEA cards). Additional data should be entered by editing the Wajac input file
STRUCAD-TO-FEM	Convert StruCAD*3D structural model data into a T*.FEM file. It will contain all principal structural data: geometry, material and physical properties as well as nodal and element loads In addition to the T*.FEM a Wajac analysis control data (input) file is created containing key hydrodynamic loading data like wave and current specifications and flooded members. The Wajac input file is a template only containing main sea data found on the StruCAD*3D input file (SPEC, FLOO, CRNT and SEA cards). Additional data should be entered by editing the Wajac input file
inp-fil	SACS/StruCAD*3D file name exclusive the file extension INP and S3D, respectively
length-unit	Length unit of output data (millimetre, metre, inch or foot)
force-unit	Force unit of output data (Newton, kiloNewton or kilo pound force)
prefix	Input Interface File prefix
selno	Input Interface File superelement number

NOTES:

1. For further details related to the conversion from Sacs, see the Sacs import manual distributed with Prepost.

COPY

COPY	<source key> <destination key>	. . .	LOADS
	ALL		MODEL
			RESULTS
			STRUCTURE
			ALL

PURPOSE:
To copy model or result data from one SIN-file to another. The file COPY reads from is called the "COPY source file". The file COPY writes to is called the "COPY destination file". Note that source and destination files cannot be the same physical file. The COPY destination file is defined by use of the OPEN command. The COPY source file is defined by use of the SET COPY SOURCE-FILE command. Note that the MERGE command is sometimes an alternative to COPY.

PARAMETERS:

<source key>	Superelement identification key in source file, defining the superelement in the source file from which data shall be copied in the case where only a specific superelement is copied. A superelement identification key is simply an integer number in the range 1 to number of 1st level superelements, see PRINT SUPERELEMENTS for further details.
<destination key>	Superelement identification key in destination file, defining the superelement in the destination file to which data shall be copied. In most cases, the superelement hierarchies within the source and destination analyses will be identical, and then <destination-key> will be identical to <source-key>.
ALL	Take data from all superelements in the source analysis and copy to the corresponding superelements in the destination analysis. This option requires that the source and destination files have identical superelement hierarchies.
LOADS	Only load data shall be copied. Load data can be copied several times for each superelement in the destination file.
MODEL	Only model data shall be copied. Model data can be copied only once for each superelement. If model data already exists, a model comparison will be performed between source and destination models.
RESULTS	Only result data shall be copied. Result data can be copied several times for each superelement in the destination file. The program will always perform a model comparison between source and destination models, to verify that it will be meaningful to copy the specified result data. Note that the resultcases to be copied must be previously defined by the command SET COPY RESULTCASES.
STRUCTURE	Only structure data, i.e. the hierarchy structure of the superelements, shall be copied. The structure data can only be copied once.
ALL	Both model and result data shall be copied See details above (MODEL and RESULTS).

See Also:

SET COPY

MERGE

OPEN

PRINT SUPERELEMENTS

CREATE

CREATE	GRIF-INTERFACE	. . .
	HYDRODYNAMIC-INTERFACE	
	POSTFEM-DATABASE	
	POSTFRAME-DATABASE	
	RESULT-COMBINATION	
	SERIES	

PURPOSE:
The command processes in various ways the results stored on the results file.

PARAMETERS:

GRIF-INTERFACE	This option is obsolete.
HYDRODYNAMIC-INTERFACE	Create a Hydrodynamic Results Interface File for further processing in Postresp.
POSTFEM-DATABASE	This option is obsolete.
POSTFRAME-DATABASE	This option is obsolete.
RESULT-COMBINATION	Create a result case combination.
SERIES	This option is obsolete.

CREATE HYDRODYNAMIC-INTERFACE

. . .	HYDRODYNAMIC-INTERFACE	<prefix>	<filename>	<select superelements>	. . .

. . .	ELEMENT-FORCES	<element number>	<integration point>	<component>
	ELEMENT-STRESSES		ALL	
	NODAL-DISPLACEMENTS	<node number>	<component>	
	END			

PURPOSE:
The command creates a Hydrodynamic Results Interface File from data on a Results Interface File. The data on the file may be read by Postresp for statistical analysis, display of transfer functions and printing. Prepost cannot read a Hydrodynamic Results Interface File.

PARAMETERS:

<prefix>	File prefix
<filename>	G*.SIF/SIN/SIU file name
<select superelements>	Select superelements, see select superelements
ELEMENT-FORCES	Element forces are written to the G*.SIF/SIN/SIU file
ELEMENT-STRESSES	Element component stresses are written to the G*.SIF/SIN/SIU file
<element number>	External basic element number
<integration point>	Element integration point number, or ALL points
<component>	Stress or force component name. A menu of component names will be available to the user. This menu will contain the names of result components stored for the actual element or node. The range of components available will depend on the actual analysis program used
NODAL-DISPLACEMENTS	Nodal displacements are written to the G*.SIF/SIN/SIU file
<node number>	External basic node number
<component>	Displacement component name. A menu of component names will be available to the user. This menu will contain the names of the result components stored for the actual element or node. The range of components available will depend on the actual analysis program used

CREATE RESULT-COMBINATION

. . .	RESULT-COMBINATION	. . .

. . .	<destination resultcase> <destination run number>	REAL	. . .
		COMPLEX	

. . .	<basic resultcase><basic run number><basic occurrence number><factor><phase>
	END

PURPOSE:
This command is used for creating combination resultcases on the current working file. The actual result combination will NOT be performed when executing this command, but sufficient information will be stored for later use, e.g. by post processing programs.

The combination definition is stored for all superelements in the current working file. It is assumed that all superelements are present, and that all superelements have the same resultcases. The program reads resultcase data from the first superelement to verify the user-input of basic resultcases.

PARAMETERS:

<destination resultcase>	Resultcase number of the combined destination resultcase to be created.
<destination run number>	Run number of the combined resultcase to be created, for further details see Resultcase identification.
REAL	The destination resultcase will be real.
COMPLEX	The destination resultcase will be complex.
<basic resultcase>	Resultcase number of a basic resultcase input to the combination. This resultcase cannot be a previously defined combined resultcase.
<basic run number>	Run number of the basic resultcase input to the combination.
<basic occurrence number>	Occurrence number of basic resultcase, for further details see Resultcase identification.
<factor>	Basic resultcase factor.
<phase>	Basic resultcase phase shift angle in degrees.

EXIT

EXIT

PURPOSE:
This command is used for terminating a Prepost session. All open files will be closed. If the current working file is a scratch file, the contents of it will be lost.

HELP

HELP	GENERAL-SYNTAX
	SPECIAL-KEYS
	STATUS-LIST
	SUPPORT

PURPOSE:
Print help in message window.

PARAMETERS:

GENERAL-SYNTAX	Command syntax.
SPECIAL-KEYS	List of special keys.
STATUS-LIST	This option is obsolete.
SUPPORT	Support contact information.

MERGE

MERGE	SIF-FORMATTED SIU-UNFORMATTED	<prefix>	<file name>	<select superelements>

PURPOSE:
This command is used for merging results from different analysis runs. The file(s) specified explicitly and implicitly in the MERGE command are merged with the current working file, defined by use of OPEN or READ commands. The resultcase run numbers, see CREATE RESULT-COMBINATION, from the result file specified in the MERGE command will be updated according to the formula:

$$\text{RN2merg} = \text{RN2orig} + \text{RNlmax}$$

where,

$$\text{RNTlmerg} = \text{New run number of second analysis in in merged file}$$
$$\text{RN2orig} = \text{Original run number of second analysis}$$
$$\text{RNlmax} = \text{Max run number of first analysis}$$

This means that if both analyses have original run numbers = 1, then the merged file will contain resultcases with run numbers 1 and 2 respectively. Note that it is the resultcase run numbers, and not the resultcase numbers themselves that are that are changed.

PARAMETERS:

SIF-FORMATTED	Merge a sequential formatted file.
SIU-FORMATTED	Merge a sequential unformatted file.
<prefix>	File prefix of file(s) to merge with working file.
<file name>	Name of file to merge with working file. In case of a superelement analysis, this must be the top level file of the second analysis. Each of the first level superelement files from the second analysis will be read automatically and merged with the data of the first analysis.
<select superelements>	Superelement selection, see select superelements.

NOTES:

1. Merge requires that the superelement hierarchies of the two analyses to be merged are identical. Merge also requires the superelements to be identical in terms of nodes and elements and their numberings.

2. The COPY command provides an alternative way of merging results from different runs.

See also:

OPEN

READ

COPY

CREATE RESULT-COMBINATION

select superelements

OPEN

OPEN	SIN-DIRECT-ACCESS	<prefix>	<name>	NEW	
				OLD	READ-ONLY
					WRITE-PERMITTED
				SCRATCH	

PURPOSE:
Open a SIN database.

PARAMETERS:

<prefix>	Result Interface File prefix.
<name>	Result Interface File name, usually on the form “R<superelement number>”.
NEW	A new file will be created to be used as working file.
OLD	Open an existing Result Interface File.
READ-ONLY	An existing file to be opened with a status that will protect it against writing. This may permit using a file that is protected by the operating system, or belongs to another user. It may also permit several users to read from the file simultaneously.
WRITE-PERMITTED	The file is opened with a status that permit the file to be modified. This is needed when you are going to perform COPY or MERGE commands. Or use CREATE RESULT-COMBINATION commands.
SCRATCH	Temporary SIN database.

See also:

COPY

MERGE

CREATE RESULT-COMBINATION

PRINT

PRINT	HISTORY		<select superelements>
	LOADCASES		
	RESULTCASES		
	SET-NAMES		
	RECORD	<record name>	
	SUPERELEMENTS		
	POSTFEM-SELECTION		

PURPOSE:
PRINT is used for printing result file information, e.g. PRINT SUPERLEMENTS or PRINT RESULTCASES, and current Prepost program settings.

Notes:

1. Print is directed to the message window and the Prepost.MLG file.

PARAMETERS:

HISTORY	This command will print the DATE records stored for the selected superelements.
LOADCASES	This command prints and overview of the load cases for the selected superelements in the current database.
RESULTCASES	This command prints an overview of resultcases for the selected superelements in the current results file. The resultcases can be listed including or excluding resultcase attributes, i.e. time instants, frequencies, wave heights etc.
SET-NAMES	This command prints and overview of the set names for the selected superelements in the current database.
RECORD	Print named records for the specified superelements. The <record name> can be one of BELOAD1, BGRAV, BNBCD, BNDISPL, BNDOF, BNINCO, BNLOAD, BNMASS, BNTEMP and BNTRCOS.
SUPERELEMENTS	Reads the superelement hierarchy information found on the current results file, and subsequently prints the information on all first level superelements (note that information on higher level superelements is not printed). This command requires that an OPEN or READ command has been entered first. The list of superelements will be as shown in the following example, where four first level superelements have been found.
POSTFEM-SELECTION	This option is obsolete.
<select superelements>	Select the relevant superelements, see select superelements.

See also:

OPEN

READ

select superelements

PRINT RESULTCASES

...	INCLUDING-ATTRIBUTES	<select superelements>
	EXCLUDING-ATTRIBUTES	

PURPOSE:
This command prints an overview of resultcases stored on the current results file. The resultcases can be listed including or excluding resultcase attributes (i.e. time instants, frequencies, wave heights etc.). The resultcases can be printed for all or a selection of superelements.

PARAMETERS:

INCLUDING-ATTRIBUTES	Resultcases are printed with resultcase attributes included.
EXCLUDING-ATTRIBUTES	Resultcases are printed without resultcase attributes.
<select SUPERELEMENTS>	Superelement selection sequence, see select superelements.

EXAMPLE:
Printout of six basic resultcases and one combination (created by use of CREATE RESULTCOMBINATION)

```
PRINT SUPERELEMENTS
------------------------------------------------------------------------
SUPERELEMENT     ! FILE ! RETRACKING ROUTE OR
KEY ! TYPE ! INDEX ! REF. ! DESCRIPTIVE TEXT
------------------------------------------------------------------------
1 !    1 !     1 !    4 ! + 1 1
2 !    2 !     1 !    5 ! + 1 1
3 !    1 !     2 !    9 ! + 1 1
4 !    2 !     2 !   10 ! + 1 1
------------------------------------------------------------------------

          RESULT ! RUN ! TYPE    ! ANALYSIS TYPE
           CASE  ! NO. !         !
   ---------------------------------------------------------
   BASIC:       1 !   1 ! REAL    ! STATIC LINEAR
   BASIC:       2 !   1 ! REAL    ! STATIC LINEAR
   BASIC:       3 !   1 ! REAL    ! STATIC LINEAR
   BASIC:       4 !   1 ! REAL    ! STATIC LINEAR
   BASIC:       5 !   1 ! REAL    ! STATIC LINEAR
   BASIC:       6 !   1 ! REAL    ! STATIC LINEAR
   COMB:       10 !   1 ! COMPLEX ! QUASI-STATIC LINEAR
```

See also:

select superelements

PRINT SUPERELEMENTS

PURPOSE:
PRINT SUPERELEMENTS will read the superelement hierarchy information found on the current results file, and subsequently print the information on all first level superelements (note that information on higher level superelements is not printed).

This command requires that the READ command has been entered first. The list of superelements will be as shown in the following example, where four first level superelements have been found.

EXAMPLE:

```
PRINT SUPERELEMENTS
------------------------------------------------------------------------
   SUPERELEMENT     ! FILE ! RETRACKING ROUTE OR
 KEY ! TYPE ! INDEX ! REF. ! DESCRIPTIVE TEXT
------------------------------------------------------------------------
   1 !    1 !     1 !    4 ! + 1 1
   2 !    2 !     1 !    5 ! + 1 1
   3 !    1 !     2 !    9 ! + 1 1
   4 !    2 !     2 !   10 ! + 1 1
------------------------------------------------------------------------
```

The first column contains the superelement KEY, which is a number in the range from 1 to number of first level superelements in the analysis. The superelement key is used elsewhere in Prepost (e.g. in CREATE, READ and WRITE) for selection of superelements to be processed.

The superelement numbers are printed in the TYPE column. The superelement repetition indices are shown in the INDEX column. Note that some analysis programs do not always write the superelement index, therefore a zero index may appear. In the above example, superelement numbers 1 and 2 are both repeated, i.e. used twice, and each usage is identified by INDEX equal to 1 and 2, respectively.

The FILE REF. column is showing the so-called hierarchy or file reference number of each superelement. This number is used in the automatically generated file names of the sequential result files. Other than being used to distinguish file names for equal superelement numbers this number is of no significance.

The last column contains the retracking route of each superelement, or other relevant information. When a retracking route is printed in this column, it will be preceded by a plus ("+"), or a minus ("-") sign. A plus sign indicates that the first level superelement is present in the direct-access result file. A minus sign indicates that the first level superelement is absent in the direct-access result file.

READ

READ	SIN-DIRECT-ACCESS	. . .
	SIF-FORMATTED	
	SIU-UNFORMATTED	
	FEM-FORMATTED	
	ANSYS	
	PATRAN	
	INTERGRAPH	
	TIMESERIES-TABLE	
	RES-FORMATTED	
	RES-UNFORMATTED	

PURPOSE:
Read one file, or a set of files, and enter their contents into the Prepost working file. When necessary, the proper data-translation will be performed before storing the data in the working file.

When reading from sequential files, the working file will be a scratch file. Use the OPEN command prior to the READ command to make a permanent SIN file.

PARAMETERS:
See Chapter 2 of this manual for an introduction to the Sesam Interface File concepts, and the terms used in connection with the different file types.

SIN-DIRECT-ACCESS	This option is obsolete, use OPEN instead.
SIF-FORMATTED	The input is one R*.SIF file, for a direct analysis, or a set of R*.SIF files, for a superelement analysis. SIF-files are ASCII formatted files.
SIU-UNFORMATTED	The input is one SIU-file, for a direct analysis, or a set of SIU-files, for a superelement analysis. SIU-files are sequential unformatted files.
FEM-FORMATTED	The input is one 1st level T*.FEM-file.
ANSYS	The input is one Ansys cdb-file. The Ansys data is converted before it is stored in the working file.
PATRAN	This option is obsolete.
INTERGRAPH	The input is one INTERGRAPH neutral file (ASCII formatted). The INTERGRAPH data is translated before it is stored in the working file.
TIMESERIES-TABLE	This option is obsolete.
RES-FORMATTED	This option is obsolete.
RES-UNFORMATTED	This option is obsolete.

See also:

OPEN

READ SIF-FORMATTED

. . .	<prefix>	<name>	<select superelement>

PURPOSE:
Read a SIF-file or a set of SIF-files into the current working file. The working file will be a scratch file unless the READ command was preceded by an OPEN command.

The SIF-format is an ASCII file format, which is commonly used when transporting data between different computers and operating systems.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	Interface File name, usually on the form "R<superelement number>". For a direct analysis enter the name of the first level SIF-file, for a superelement analysis enter the name of the top level SIF-file.
<select superelement>	Select superelements, see select superelements for a description. PRFPOST will automatically generate file names and read the selected first level superelements. Files that are not found will be reported, and then ignored.

See Also:

OPEN

select superelements

READ SIU-UNFORMATTED

. . .	<prefix>	<name>	<select superelements>

PURPOSE:
Read a SIU-file or a set of SIU-files into the current working file. The working file will be a scratch file unless the READ command was preceded by OPEN command.

The SIU-format is an unformatted sequential file format. The SIU format is up to 5 times as compact as the SIF-format, and therefore well suited for long-term data-storage.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	Interface File name, usually on the form "R<superelement number>". For a direct analysis enter the name of the first level SIU-file, for a superelement analysis enter the name of the top level SIU-file.
<select superelements>	Select superelements, see select superelements for a description. PRFPOST will automatically generate file names and read the selected first level superelements. Files that are not found will be reported, and then ignored.

See Also:

OPEN

select superelements

READ FEM-FORMATTED

. . .	<prefix>	<name>	<select superelements>

PURPOSE:
Read a first level T*.FEM file into the current working file. The working file will be a scratch file unless the READ command was preceded by an OPEN command.

See Chapter 2 for more general information on FEM-files.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	First level Input Interface File name, usually on the form "T<superelement number>"
<select superelements>	Select superelements, see select superelements for a description. PRFPOST will automatically generate file names and read the selected first level superelements. Files that are not found will be reported, and then ignored.

NOTES:

1. Top or intermediate level T*.FEM files cannot be read.

   Only first level T*.FEM files can be read by this command, i.e. files created by for example GeniE. The files created by Presel cannot be read.

2. Loads Interface Files cannot be read.

   This command handles only first level Interface File containing geometry data.

See Also:

OPEN

select superelements

READ ANSYS

. . .	<prefix>	<name>	<read loadstep file>	<superelement number>

PURPOSE:
Read an Ansys cdb file, and possibly a set of load files, into a Sesam Input Interface File. The working file will be a scratch file. Use the OPEN command to make a permanent SIN file.

PARAMETERS:

<prefix>	Ansys command file prefix.
<name>	Ansys command file name.
<read loadstep file>	0: No, do not read load files. 1: Yes, give number of loadstep files. The files are specified as a range 1-n, where n is a number. The maximum number of loadstep files is 99.
<superelement number>	The Sesamsuperelement number to be assigned to the Ansys model.

See Also:

OPEN

NOTES:

1. For further details related to the conversion from Ansys to Sesam and Sesam to Ansys, see the Sesam-Ansys manual distributed with Prepost.

READ INTERGRAPH

...	<prefix>	<name>	<superelement number>	<options>	...

PURPOSE:
Read an Intergraph neutral file, translate it into Sesam format, and store it the current working file. The working file will be a scratch file unless the OPEN command is used.

This Intergraph to Sesam translation facility is a preprocessor to analysis program interface tool, meaning that the user can use Intergraph to create the finite element model, and perform the structural analysis in Sesam, e.g. using Sestra.

PARAMETERS:

<prefix>	Intergraph neutral file prefix.
<name>	Intergraph neutral file name.
<superelement number>	The Sesam superelement number to be assigned to the Intergraph model.
<options>	Intergraph model translation options. See Appendix I of this manual for a full description of the translation options.

See Also:

OPEN

WRITE FEM-FORMATTED

NOTES:

1. Creating Sesam Input Interface Files from Intergraph models.

   After translating the Intergraph model into the working file, it must be written out to a Sesam Input Interface File before a structural analysis can be executed. That is done by using the WRITE FEM-FORMATTED command.

SET

SET	COMMAND-INPUT-FILE	. . .
	COPY	
	EXTENSION-WORKING-FILE	
	PERMANENT-WORKING-FILE	
	POSTFEM-SELECTION	
	POSTFRAME-SELECTION	
	END	

PURPOSE:
SET is used for

- defining command-input-file.
- defining selections for the COPY command.
- creating a new permanent working-file.

PARAMETERS:

COMMAND-INPUT-FILE	Open a new Prepost command input file.
COPY	Set selections for the COPY command.
EXTENSION-WORKING-FILE	This option is obsolete.
PERMANENT-WORKING-FILE	This option is depreciated, use OPEN.
POSTFEM-SELECTION	This option is obsolete.
POSTFRAME-SELECTON	This option is obsolete.
END	End the SET command.

See also:

COPY

OPEN

SET COMMAND-INPUT-FILE

. . .	COMMAND-INPUT-FILE	<prefix>	<filename>

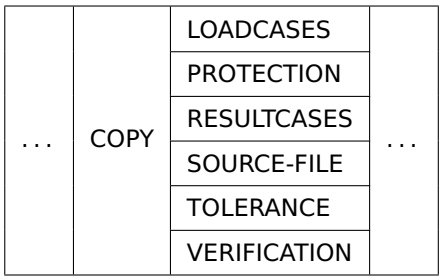
PURPOSE:
This command is used for defining name of a command-input-file containing Prepost commands. The commands of the COMMAND-INPUT-FILE are executed by subsequently entering the # command.

The command-input-file is a journal file created in a previous Prepost session, or edited by the user in a text editor.

PARAMETERS:

<prefix>	command-input-file prefix.
<filename>	command-input-file name.

SET COPY



PURPOSE:
To set up control information for the COPY command, such as copy source file name, source and destination resultcases and destination file resultcase protection. Note that the copy source file name must be specified before the copy protection and resultcases can be specified.

PARAMETERS:

LOADCASES	Define loadcases within the copy source file to be copied and the corresponding loadcases within the copy destination file.
PROTECTION	Define resultcase protection for resultcases within the destination file. The default is set to PROTECTED. The protection defined in this command is only temporary, and is not saved on the destination file. The effect is that in a subsequent Prepost session, all resultcases will be PROTECTED. Upon termination of the COPY command, the protection specification will also be cleared, leaving all resultcases PROTECTED.
RESULTCASES	Define resultcases within the copy source file to be copied, and the corresponding resultcases within the copy destination file.
SOURCE-FILE	Specification of copy source file, i.e. the file that contains the original data to be copied. The copy source file must be a SIN-DIRECT-ACCESS file. Prepost can create such files from other Sesam formats, by use of the SET PERMANENT WORKING-FILE and READ commands.
TOLERANCE	Specification of verification tolerance. During execution of the COPY command, the program will verify that source and destination geometry models are identical. The TOLERANCE option is used in cases where there are small numerical differences between source and destination models.
VERIFICATION	Specification of verification status for individual Sesam datatypes. In certain rare cases, the source and destination models may contain differences that causes the COPY command to refuse processing. The SET COPY VERIFICATION option allows the user to switch off the verification for the failing datatype. **It should be noted that this feature should be used with care, and ONLY when the causes for the verification failure are fully understood. Switching off datatype verification without investigating the reasons for verification failure, may lead to unpredictable program behaviour and irrecoverable errors in any program later accessing the corrupted SIN-file. If in doubt, consult your local Sesam representative for advice.**

SET COPY LOADCASES

. . .	<source loadcase>	<destination loadcase>
	CLEAR	
	END	

PURPOSE:
Define loadcases within the copy source file to be copied, and the corresponding loadcases within the copy destination file.

PARAMETERS:

<source loadcase>	Unique identification of an existing loadcase within load file. See Result identification for further details on loadcase identification.
<destination loadcase>	Unique identification of a loadcase within copy destination file. See next page for further details on loadcase identification. If the specified destination laodcase does not exist, it will be appended to the existing loadcase in the destination file, when the COPY command is used. If the specified destination loadcase already exist, the existing destination loadcase may be overwritten, provided that it is not protected, see the SET COPY PROTECTION command. If the destination resultcase is protected, the SET COPY LOADCASE request will be ignored for the specified source and destination loadcase.
CLEAR	Clear the loadcase copy list completely, and await new specification of load cases to copy.
END	End of loadtcase copy specification.

See also:

Result identification

COPY

SET COPY PROTECTION

SET COPY PROTECTION

...	ALL	...	PROTECTED
	<destination resultcase>		UNPROTECTED
	END		

PURPOSE:
To set up control information for the COPY command, such as copy source file name, source and destination resultcases and destination file resultcase protection. Note that the copy source file name must be specified before the copy protection and resultcases can be specified.

PARAMETERS:

<destination resultcase>	Unique identification of an existing resultcase within the destination file for which protection is to be set. See SET COPY RESULTCASES for details on unique resultcase identification.
ALL	Define resultcases within the copy source file to be copied, and the corresponding resultcases within the copy destination file.
END	End of protection specification.
PROTECTED	A resultcase is protected and may not be overwritten in COPY command.
UNPROTECTED	A resultcase may be overwritten in COPY command.

See also:

COPY

SET COPY RESULTCASES

SET COPY RESULTCASES

...	<source resultcase>	<destination resultcase>
	CLEAR	
	END	

PURPOSE:
Define resultcases within the copy source file to be copied, and the corresponding resultcases within the copy destination file.

PARAMETERS:

RESULTCASES	Define resultcases within the copy source file to be copied, and the corresponding resultcases within the copy destination file.
<source resultcase>	Unique identification of an existing resultcase within source file, see Result identification on the next page for further details.
<destination resultcase>	Unique identification of a resultcase within copy destination file. See next page for further details on resultcase identification. If the specified destination resultcase does not exist, it will be appended to the existing resultcases in the destination file, when the COPY command is used. If the specified destination resultcase already exist, the existing destination resultcase may be overwritten, provided that it is not protected, see the SET COPY PROTECTION command. If the destination resultcase is protected, the SET COPY RESULTCASE request will be ignored for the specified source and destination resultcase.
CLEAR	Clear the resultcase copy list completely, and await new specification of result cases to copy.
END	End of resultcase copy specification.

See Also:

Result identification

Resultcase identification:

Resultcases within source or destination files are referred to using the following syntax:

. . .	<resultcase>	<run>	<g-occurrence>
			<e-occurrence>

where

<resultcase> Resultcase number.
This usually corresponds to loadcase numbers (static analyses). For certain analyses, several resultcases may share the same resultcase number, e.g. for time domain analyses. In such cases, the occurrence numbers are used for unique resultcase reference.

<run> Run number.
This number is used to identify results from different analysis program runs. If the MERGE command has been used to combine results from two result files, the resultcases from the second analysis will be given new run numbers (see MERGE command for further information).

<g-occurrence> General occurrence number.
If entered as zero, Prepost will assume that the resultcase and run numbers are sufficient for unique identification. If entered as a positive number, Prepost will use the <g-occurrence> number and check whether this uniquely identifies a resultcase. In some cases, use of the <g-occurrence> number will not be sufficient for unique identification. Then, the explicit <e-occurrence> number should be used instead.

<e-occurrence> Explicit occurrence number.
For example, an excitation frequency number. This is the safest way of referring to occurrences, since the same resultcase may contain several occurrence numbers, e.g. WAVE-DIRECTION-NUMBER and EXITATION-FREQUENCY-NUMBER. The explicit occurrence number may be one of:

RESPONSE-FREQUENCY-NUMBER
TIME-INSTANT-NUMBER
ANGLE-NUMBER
LOAD-PARAMETER-NUMBER
WAVE-DIRECTION-NUMBER
WAVE-HEIGHT-NUMBER
EXCITATION-FREQUENCY -NUMBER
WATER-DEPTH-NUMBER
CRITICAL-LOAD-LEVEL-NUMBER

SET COPY SOURCE-FILE

. . .	<prefix>	<name>

PURPOSE:
Specification of COPY source file, i.e. the file that contains the original data to be copied in the COPY command. The copy source file must be a SIN-file. Prepost can create such files from other Sesam formats, by use of the OPEN command.

PARAMETERS:

<prefix>	Prefix of copy source file.
<name>	Name of copy source file.

See Also:

COPY

OPEN

SET COPY TOLERANCE

. . .	<abs-tolerance>	<rel-tolerance>

PURPOSE:
Specification of verification tolerance. During execution of the COPY command, the program will verify that the source and destination geometry models are identical. The TOLERANCE option is used in cases where there are small numerical differences between source and destination models.

PARAMETERS:

<abs-tolerance> Absolute tolerance:

$$|\text{<destination-value>} - \text{<source-value>}| \leq |\text{<abs-tolerance>}|$$

<rel-tolerance> Relative tolerance:

$$\frac{|\text{<destination-value>} - \text{<source-value>}|}{|\text{<source-value>}|} \leq |\text{<abs-tolerance>}|$$

SET COPY VERIFICATION

. . .	<datatype>	OFF
		ON

PURPOSE:
Specification of verification status for individual Sesam datatypes. In certain rare cases, the source and destination models may contain differences that causes the COPY command to refuse processing. The SET COPY VERIFICATION option allows the user to switch off the verification for the failing datatype.

It should be noted that this feature should be used with care, and ONLY when the causes for the verification failure are fully understood. Switching off datatype verification without investigating the reasons for verification failure, may lead to unpredictable program behaviour and irrecoverable errors in any program later accessing the corrupted SIN-file. If in doubt, consult your local Sesam representative for advice

PARAMETERS:

<datatype>	Sesam geometry or model description datatype name. See the summary list below.
OFF	The datatype shall not be verified.
ON	The datatype shall be verified.

Sesam geometry or model description datatypes. Below follows a summary of some datatype names and their description. For a full description of each datatype, see the Sesam Interface File documentation.

TDMATER	Material - name and description.
MISOSEL	Material - linear elastic, isotropic.
MISOPL	Material - Non-linear elastic, isotropic.
MISOHL	Linear heat conduction, isotropic.
MISOHNL	Non-linear heat conduction, isotropic.
MISOEML	Linear electromagnetic field problem, isotropic.
MISOAL	Linear acoustic field problem, isotropic.
MORSMEL	Linear elastic, anisotropic, thin shell.
MORSSEL	Linear elastic, anisotropic, thick shell.
MORSSOL	Linear elastic, anisotropic, solid.
MAXSPR	Spring constant - axial spring.
MAXDMP	Damping constant - axial damper.
MGSPRNG	Spring matrix - spring to ground.
MGDAMP	Damping matrix - damper to ground.
MSHGLSP	Spring matrix - between nodes.
MGLMASS	Mass matrix - between nodes.
MGLDAMP	Damping matrix - between nodes.
MGMASS	Mass matrix - on node.
MISTEL	Temperature dep. linear elastic, iso.
MTENOL	General material with temperature dep.
TDSECT	Section - name and description.
GELTH	Thickness of 2-d elements.
GBEAMG	Section - general beam.

GIORH	Section - type I OR H.
GUSYI	Section - type Unsymmetrical I beam.
GCHAN	Section - type Channel.
GBOX	Section - type Box.
GPIPE	Section - type Pipe.
GBARM	Section - type Massive bar.
GTONP	Section - type T on plate.
GDOBO	Section - type Double Bottom.
GLSEC	Section - type L.
GIORHR	Section - type I OR H with inside cum.
GCHANR	Section - type Channel with in curv.
GLSECR	Section - type L with inside curv.
BNTRCOS	Transformation matrix.
GUNIVEC	Local beam element coordinate system.
GECCEN	Eccentricity.
BELFIX	Flexible joint/hinge.
MTEMP	Scaling curve for temperature variation.
TDNODE	Node - name and description.
GNODE	Node - identification.
GCOORD	Node - coordinates.
BNBCD	Node - boundary condition.
BLDEP	Node - linear dependency.
BQDP	Node - quadratic dependency.
BNDOF	Node - transformation reference.
BNMASS	Node - mass.
TDELEM	Element - name and description.
GELMNT1	Element - identification.
GELREF1	Element - references.
MTRSOL	Local trans. of the axis of anisotropy.
MTRMEL	Local trans. of the axis of anisotropy.
MTRSEL	Local trans. of the axis of anisotropy.
BEMASS1	Element hydrodynamic added mass.
TDSETNAM	Set - name and description.
GSETMEMB	Set - list of members.
RDNODRES	Nodal results record description.
RDSTRESS	Element stress component definition.
RDSTRAIN	Element strain component description.
RDFORCES	Element force component record descript.
RDPOTNTS	Element result point definition
RDIELCOR	Internal element coordinates
RSUPTRAN	1st level superelement transformation

TRANSFORM

TRANSFORM	CARTESIAN	. . .
	CYLINDRICAL	
	REFERENCE	
	SELECT	
	SPHERICAL	
	END	

PURPOSE:
Results are stored as components in a local results coordinate system which may be different for each result point. Thus, the results components stored on the results may not be in a coordinate system convenient for the user. The TRANSFORM command transforms the stress components for each result point to a local axis system specified by the user.

For the transformation to be performed, one of the alternatives CARTESIAN or CYLINDRICAL must be selected to to define the user defined local coordinate system. When the transformation has been performed the stresses will be stored as 3-dimensional stress components, also for element types with a reduced number of stress components, e.g. the 4 node shell element.

If the TRANSFORM command is repeated, specifications and selections from the previous command will remain as defaults.

For the TRANSFORM command to be used, the direct access results file must be opened in a mode that allows writing data on the file, see the OPEN command.

PARAMETERS:

CARTESIAN	Cartesian transformation. Two orthogonal vectors are specified, the third is orthogonal to the specified vectors.
CYLINDRICAL	Cylindrical transformation. The coordinates of the centre and the axial direction vector of the cylinder is specified.
REFERENCE	Select the reference coordinate system for specification of centre axis and direction of vectors. The initial default is the local coordinate system for each individual superelement.
SELECT	Select the part of the model where the transformation is applied. The default is the entire model.
SPHERICAL	This option is obsolete.

See Also:

OPEN

TRANSFORM CARTESIAN

...	VECTOR
	X-AXIS
	Y-AXIS
	Z-AXIS

PURPOSE:
Specify a local Cartesian coordinate system.

Two orthogonal vectors are specified, the fist gives the local x-axis the second the local y-axis, the third is orthogonal to the specified vectors and gives the local z-direction. To specify the vectors either select X-AXIS, Y-AXIS or Z-AXIS, or select VECTOR and give the three values of the vector.

The coordinate system used to specify the local coordinate system is either the global top level or the local superelement coordinate system, see also the TRANSFORM REFERENCE command.

PARAMETERS:

VECTOR	Prompts the user for three numbers, x, y and z, the coordinates of the local axis in the given coordinate system.
X-AXIS	Selects the x-axis of the given coordinate system as the local axis.
Y-AXIS	Selects the y-axis of the given coordinate system as the local axis.
Z-AXIS	Selects the z-axis of the given coordinate system as the local axis.

See Also:

TRANSFORM REFERENCE

TRANSFORM CYLINDRICAL

. . .	<x, y, z>	VECTOR
		X-AXIS
		Y-AXIS
		Z-AXIS

PURPOSE:
Specify a local cylindrical coordinate system by giving the coordinates of the origin and the axial direction vector of the cylinder.

The coordinate system used to specify the local coordinate system is either the global top level or the local superelement coordinate system, see also the TRANSFORM REFERENCE command.

PARAMETERS:

<x, y, z>	The user is prompted for the x-, y- and z-coordinates of the origin
VECTOR	The x-, y- and z-coordinates of the axial direction vector.
X-AXIS	Selects the x-axis of the reference coordinate system as the direction vector.
Y-AXIS	Selects the y-axis of the reference coordinate system as the direction vector.
Z-AXIS	Selects the z-axis of the reference coordinate system as the direction vector.

See Also:

TRANSFORM REFERENCE

TRANSFORM REFERENCE

. . .	GLOBAL-TOPLEVEL
	LOCAL-SUPERELEMENT

PURPOSE:
Select the reference coordinate system used to specify the local coordinate system given in the TRANSFORM command. The reference system can be either the coordinate system of the global top level or the actual superelement. The initial default is the local coordinate system for each individual superelement.

PARAMETERS:

GLOBAL-TOPLEVEL	The reference system is the coordinate system of the global top level
LOCAL-SUPERELEMENT	The reference system is the coordinate system of the local superelement.

See Also:

TRANSFORM

TRANSFORM SELECT

. . .	ELEMENT-SET	SELECTION-SET-NAME
		ALL
	SUPERELEMENT	<select superelements>

PURPOSE:
Select the elements where the stresses are to be transformed. Select an element set, all elements, or one or more superelements. The default is the entire model.

PARAMETERS:

ELEMENT-SET	Select the set of elements to be transformed, either named sets or ALL.
SUPERELEMENT	Select elements in superelements, see select superelements.

See Also:

select superelements

WRITE

WRITE	SIF-FORMATTED	. . .
	SIU-UNFORMATTED	
	FEM-FORMATTED	
	ANSYS	
	PATRAN	
	INTERGRAPH	

PURPOSE:
Use the contents of the current Prepost working file and write an output file (or a set of output files). When necessary, the proper data-translation will be performed before storing the data on the output file(s).

See Chapter 2 for an introduction to the Sesam Interface File concepts, and the terms used in connection with the different file types.

PARAMETERS:

SIF-FORMATTED	The output is one R*.SIF file for direct analysis, or a set of R*.SIF files for superelement analysis. R*.SIF files are ASCII formatted files.
SIU-UNFORMATTED	The output is one R*.SIU file for direct analysis, or a set of R*.SIU files for superelement analysis. SIU-files are sequential unformatted files.
FEM-FORMATTED	The output is one 1st level T*.FEM file.
ANSYS	The output is an Ansys file.
PATRAN	This option is obsolete.
INTERGRAPH	The input is one Intergraph neutral file, ASCII formatted.

WRITE SIF-FORMATTED

...	<prefix>	<name>	<select superelements>

PURPOSE:
Write a R*.SIF file or a set of R*.SIF files using the contents of the current working file. The SIF-format is an ASCII file format, which is commonly used when transporting data between different computers and operating systems.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	Interface File name, usually on the form "R<superelement number>". For direct analyses enter the name of the first level SIF-file, superelement analyses enter the name of the top level SIF-file. See Chapter 2 for more details on file naming conventions.
<select superelements>	Select superelements, see select superelements for a description. Prepost will automatically generate file names and write the selected first level superelements. Superelements that are not found on the working file will be reported, and then ignored.

WRITE SIU-UNFORMATTED

. . .	<prefix>	<name>	<select superelements>

PURPOSE:
Write a R*.SIU file or a set of R*.SIU files using the contents of the current working file. The SIU-format is an unformatted sequential file format. The SIU-format is up to 5 times as compact as the SIF-format, and therefore well suited for long-term data-storage.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	Interface File name, usually on the form "R<superelement number>". For direct analyses enter the name of the first level SIU-file, for superelement analyses enter the name of the top level SIU-file. See Chapter 2 for more details on file naming conventions.
<select superelements>	Select superelements, see select superelements for a desciption. Prepost will automatically generate file names and write the selected first level superelements. Superelements that are not found on the working file will be reported, and then ignored.

WRITE FEM-FORMATTED

...	<prefix>	<name>	<select superelements>

PURPOSE:
Write a first level T*.FEM file, a Sesam Input Interface File containing model geometry data, and optionally load data, using a superelement from the current working file. See Chapter 2 for more general information on FEM-files.

PARAMETERS:

<prefix>	Interface File prefix.
<name>	Interface File name, usually on the form "T<superelement number>"
<select superelements>	Select superelements, see select superelements. The program will present a list of 1st level superelements, from which the user may select one 1st level superelement. The selected superelement will subsequently be written to the specified file.

NOTES:

1. Top or intermediate level T*.FEM files cannot be written.
   Only first level T*.FEM files can be written in this command.
2. Separate Loads Interface Files cannot be written.
   The first level T*.FEM file written in this command will contain model geometry and model load data, i.e. creation of separate Loads Interface Files is not relevant.
3. Analysis results data is not written.
   Any analysis results stored with the model in the working file, will not be written. See commands WRITE SIF-FORMATTED or WRITE SIU-UNFORMATTED for writing model and results data.

See Also:

WRITE SIF-FORMATTED

WRITE SIU-UNFORMATTED

select superelements

WRITE ANSYS

. . .	<prefix>	<name>	<superelement number>	<default conversion parameters>	. . .

. . .	<Ansys release>	<blocked format>	<real constant naming>	<title>	. . .

. . .	<loadcase file format>	<write loadstep files>

PURPOSE:
Translates the data of one superelement in the working file and writes to Ansys input format.

PARAMETERS:

<prefix>	Ansys command files prefix.
<name>	Ansys command files name.
<superelement number>	Superelement number, see Chapter 2 .
<default conversion parameters>	1 = yes. Default parameters. No more input required. 0 = no. User selected parameters.
<Ansys release>	1 – release 5.5 and newer. 2 – release 5.4 and older
<blocked format>	Select data to be in blocked or unblocked format. 1 = blocked 2 = unblocked 3 = user defined The following data can be blocked: <nodes>, <elements>, <components>, <real constants>, <material definitions>. Blocked format is used to speed up reading of the Ansys file.
<real constant naming>	The real constants have unique integer names. 1 = simple, the naming is sequential starting with 1. 2 = grouped, select names to group the following real constants:

<thickness low>	Start value for lower thickness name. Default: 0.
<thickness high>	Start value higher thickness name. Default: 600.
<point masses>	Start value for point masses. Default: 2000.
<beam4>	start value for beam4 real constants. Default: 700.
<beam44>	start value for beam144 real constants. Default: 3000
<beam188/beam189>	start value for beam188 and beam189 section data. Default: 2.

<title>	Title line of the Ansys conversion.
<loadcase file format>	The loadcase file will either use node and element id’s for adding loads or use special commands for identifying nodes from their coordinates and identifying elements from nodes.

<write loadstep files>	0 = no. Write loadcases to the Ansys cdb file. 1 = yes. Write one loadstep file for each loadcase. <prefix> Prefix for loadstep files. <name> Name of loadstep files. The name of the first loadstep file will be <prefix><name>.s01.

WRITE INTERGRAPH

...	<prefix>	<name>	<superelement key>	<additional node no.>

PURPOSE:
Translate the data of one superelement in the current working file, and write the translated data to an Intergraph neutral file.
This Sesam to Intergraph translation facility is a preprocessor to analysis program interface tool, meaning that the user can use Sesam to create the finite element model, and perform the structural analysis in a finite element analysis system that is compatible with the Intergraph neutral file. The model can also be imported for display and/or further modification within Intergraph.

PARAMETERS:

<prefix>	Intergraph neutral file prefix
<name>	Intergraph neutral file name.
<superelement key>	Superelement key as described for command PRINT SUPERELEMENT. The program will present a list of 1st level superelements, from which the user may select one 1st level superelement. The selected superelement will subsequently be written to the specified Intergraph file.
<init. mass element no>	Initial external node number for additional nodes that may be created at beam element ends. If the zero is specified, no additional nodes will be created. See Appendix I of this manual for a full description of the Sesam to Intergraph communication.

NOTES:

1. Load translation.
   Loads are in general not translated, with the exception of node loads/displacements, line loads on beams, normal pressure on 3 and 4 noded shell elements and uniform element temperature loads.
2. Result translation.
   Finite element analysis results are not translated.

See Also:

PRINT SUPERELEMENT

#

#	<number of commands>
	ALL

PURPOSE:
This command is used for execution of commands stored in a command-input-file. The name of the file must first be defined by use of SET COMMAND-INPUT-FILE.

After execution of the commands taken from the command input file, the program will return control to the user.

PARAMETERS:

<number of commands>	Number of main commands on the command-input-file to be executed. Main commands are READ, WRITE, CREATE, ... etc.
ALL	All commands on the command-input-file shall be executed.

See Also:

SET COMMAND-INPUT-FILE

<select superelements>

	ALL	
...	<superelement number>	*
	END	

PURPOSE:
Subcommand used to specify superelements.

PARAMETERS:

ALL	Select all superelements.
<superelement number>	An integer referring to the superelement. See PRINT SUPERELEMENTS for description.
END	Ends a list of superelement numbers.

See Also:

PRINT SUPERELEMENTS

H Patran Interface

Below is a scanned version of one page from Appendix H in the previous version of the Prepost User manual. It is include since it is referenced from Appendix I.

H 3 Reading PATRAN neutral files

A PATRAN neutral file is translated for use in SESAM by use of the READ command in PREPOST (see Chapter 5, READ command):

```
#READ PATRAN <prefix> ...
```

After specifying the file prefix, the user/program dialogue continues as follows:

PATRAN neutral file name ? /PATRAN/
The PATRAN neutral file must be of type OUT, hence the default name on VAX is PATRAN.OUT (on CDC: PATRAN_OUT).

H 3.1 Superelement & boundary condition interpretation

Superelement type number ? /1/
The superelement type number is the SESAM identification, and the name of the interface file generated will be T1.FEM if the type number is 1. (on CDC: T1_FEM)

Direct or superelement analysis ? /SUPERELEMENT/

DIRECT	When specifying DIRECT, fixed degrees of freedom are kept as fixed.
SUPERELEMENT	When specifying SUPERELEMENT, fixed degrees of freedom are converted to super degrees of freedom.

Number of degrees of freedom per node? /SPECIFIED/

3-DOF	All nodes have 3-dof (use for solid element models).
6-DOF	All nodes have 6-dof (use for beam and shell models).
SPECIFIED	The nodes have the number of degrees of freedom specified on each node command from PATRAN

I Intergraph Interface

Below is a scanned version of Appendix I in the previous version of the Prepost User manual.

I 1 INTEGRAPH to SESAM

Translation of a INTERGRAPH FEM neutral file to SESAM interface file may be done by the command:

```
# READ INTEGRAPH <prefix> <filename> <superelement type> ...
                 ... <direct_or_super> <number_of_dof> ...
                 ... <load_options>
```

The options <direct_or_super> and <number_of_dof> are interpreted as described for PATRAN in Appendix H. The <load-options> loop is repeated until END is encountered. The following load_options may be used for INTERGRAPH:

```
( <llc>                                  ) ( NORMAL    ( INSIDE      ) )
( GROUP <llc1> <llc2> <inc>              ) ( HYDRO     ( MID-SURFACE ) )
( ALL                                    ) ( HYDRO     ( OUTSIDE     ) )
(                                        ) (                           )
(                                        ) ( TEMPERATURE               )
(                                        )
( END                                    )
```

Give END immediately if no loads are to be translated. In order to increase efficiency, use GROUP instead of the ALL option.

At the end of load options NORMAL and HYDRO, the user must give the surface of thick shell elements on which surface pressure is acting. This information is disregarded for thin shell, solid and beam elements.

INSIDE	negative local Z
MID-SURFACE	middle of element (default)
OUTSIDE	positive local Z

The load subcommand TEMPERATURE should be used for temperature loadcases, for all others NORMAL should be used. If a hydrodynamic model for WADAM is created, the HYDRO option defines direction of hydrostatic and dynamic pressure on the wetted surfaces.

Limitations:

1 Only cross section types dimensioned properties ROUND and RECTANGULAR (solid or box) may be translated.

2 Translation of loads is ONLY available for 2 node beams and normal pressure on 3 and 4 node shell elements.

3 Loadcases should have the same numbers in SESAM and INTERGRAPH.

Example:

```
# READ INTERGRAPH MODEL_ T12 12 DIRECT SPECIFIED
        GROUP 1 20 1 NORMAL MID
        GROUP 21 22 1 TEMPERATURE
        END
```

Reads the INTERGRAPH FEM neutral file MODEL_T12.IRM, and translates loadcases 1 to 20 as node and element loads, 21 and 22 as temperature loads.

Remember that the superelement created must be written out using the commmand:

WRITE FEM <prefix> <filename> <key reference>

I 2 SESAM to INTERGRAPH

Translation from SESAM to INTERGRAPH neutral file is also available, using the command:

READ FEM <prefix> <filename> <key reference>

and then:

WRITE INTERGRAPH <prefix> <filename> <key reference> <addit_node_no>

where <addit_node_no> is the initial external node number for additional nodes that MAY be created at the element ends. If <addit_node_no> is set to zero, then no additional nodes are created. The option of giving <addit_node_no> > 0 should only be used if a model is transferred for the preparation of drawings.

Limitations:

1 Loads are in general not translated, with the exception of node loads/displacements, line loads on beams, normal pressure on 3 and 4 node shell elements, uniform element temperature loads.

ABOUT DNV GL

Driven by our purpose of safeguarding life, property and the environment, DNV GL enables organizations to advance the safety and sustainability of their business. We provide classification and technical assurance along with software and independent expert advisory services to the maritime, oil and gas, and energy industries. We also provide certification services to customers across a wide range of industries. Operating in more than 100 countries, our 16,000 professionals are dedicated to helping our customers make the world safer, smarter and greener.

SOFTWARE

DNV GL is the world-leading provider of software for a safer, smarter and greener future in the energy, process and maritime industries. Our solutions support a variety of business critical activities including design and engineering, risk assessment, asset integrity and optimization, QHSE, and ship management. Our worldwide presence facilitates a strong customer focus and efficient sharing of industry best practice and standards