

# **CAD Toolkit User Manual**

---

**LUSAS Version 21.0 : Issue 1**

LUSAS  
Forge House, 66 High Street, Kingston upon Thames,  
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999  
Fax +44 (0)20 8549 9399  
Email: [info@lusas.com](mailto:info@lusas.com)  
<http://www.lusas.com>

Distributors Worldwide

Copyright ©1982-2023 LUSAS  
All Rights Reserved.

## Table of Contents

General Import and Export Capabilities	1
Standalone programs .....	2
PATRAN Interface	3
Introduction .....	3
Installation .....	3
PATRAN to LUSAS (PATLUS) .....	4
LUSAS to PATRAN (LUSPAT) .....	10
Interface Integration with PATRAN .....	13
PATLUS Element Details .....	18
LUSPAT Results Columns .....	26
System Parameters.....	30
FEMView and FAM Interface	33
Introduction .....	33
LUSAS to FEMView/FAM.....	33
FEMView/FAM Stress Types .....	37



# General Import and Export Capabilities

Interface files are used to transfer external modelling or material data into and out of LUSAS Modeller. The full model or a selected portion of a model can, dependent upon the file format chosen, be exported to a chosen interface file. Currently supported interface file formats are:

- ABAQUS input files (.inp)** (used to import node and element data)
- ANSYS cdb files (.cdb)** (used to import node and element data)
- BIM/BrIM (\*.ifc)** (used to import and export BIM/BrIM file data)
- CMD (.cmd)** Format for import of LUSAS Modeller model files saved as command (CMD) files in previous versions of LUSAS.
- DXF (.dxf)** AutoCAD Drawing eXchange Format.
- ETABS input files (.e2k)** (used to import geometry and property/attribute data)
- IGES (.igs)** Initial Graphics Exchange Specification. Format for import and export of geometry data.
- LPI Script Files (.lbv or .lpy or .vbs)** contain LUSAS Modeller commands that, when they are replayed, a sequence of operations may be carried out automatically.
- MIDAS input files (.mct, .mgt)** (used to import geometry and property/attribute data)
- NASTRAN Bulk Data files (.bdf, .dat)** (used to import node and element data)
- PATRAN (.def)** Neutral file format for inputting phase I geometry information and outputting phase II mesh information
- SAP input files (.s2k)** (used to import geometry and property/attribute data)
- Search area topology (.inf)** contain details of search areas used by Autoloader vehicle load optimisation software. These when imported into LUSAS are converted into geometric line and surface data for graphical cross-checking.
- Solver Data Files (.dat)** LUSAS Solver data files (used to import or node and element data)

- ❑ **STEP** (.stp) S**T**andard for the Exchange of Product data.
- ❑ **STL** (.stl) S**T**ereolithography data files.

Full details on all these interfaces files can be found in the *Modeller Reference Manual*.

### Third-party interface tools

- ❑ **Grasshopper** is a visual programming language and environment which runs within Rhino. Using a third-party Grasshopper plug-in, LUSAS "components" can be accessed from Grasshopper allowing interaction with Grasshopper parametric models. A "bake to LUSAS" function is provided. See <https://technical.geometrygym.com/rhino-grasshopper/structuralanalysis/lusas-or-ggrhinolusas>
- ❑ **Revit** A Revit plug-in allows for opening a Revit model and exporting supported data to LUSAS. See the LUSAS help file provided within Revit for more information.

### Standalone programs

In addition to interface files that can be imported or exported from within LUSAS Modeller, other standalone programs are also available and are documented in this User manual. They comprise:

#### **PATRAN Interface**

The LUSAS-PATRAN Interface transfers data between LUSAS Solver and PATRAN. PATRAN is used to generate a neutral file which contains all of the information required to generate a valid LUSAS data deck. The conversion of the data is achieved using the program PATLUS.

The conversion of results from LUSAS format to PATRAN format is handled by the program LUSPAT. LUSPAT takes a LUSAS restart file and generates PATRAN results files for each load case and results type.

#### **FEMView and FAM Export**

FVLUSAS is the LUSAS Solver to FEMView/FAM export program supported by LUSAS.

The program reads results from the LUSAS restart file. The LUSAS restart file is named **jobname.rst**. A restart file is created by LUSAS using the **RESTART WRITE** or **RESTART WRITE BRIEF** commands in the data file or using the appropriate **OUTPUT** option in various analysis control sections in the data file.

# PATRAN Interface

## Introduction

This section describes the LUSAS-PATRAN Interface for the transfer of data between LUSAS Solver and PATRAN.

PATRAN is a registered trademark of the MSc Corporation.

## Installation

All software required for your LUSAS to PATRAN interface should be installed automatically with any software update from LUSAS. The files required for running the LUSAS to PATRAN interface (listed below) are all supplied on the standard LUSAS Release Kit.

The following files will be installed in the **\$FEADIR** directory:

<b>patlus</b>	Script for running the PATRAN to LUSAS translator.
<b>patlus.exe</b>	PATRAN to LUSAS translator.
<b>luspat</b>	Script for running the LUSAS to PATRAN translator.
<b>luspat.exe</b>	LUSAS to PATRAN translator.

The following files will be installed in the **\$FEADIR/config** directory:

<b>patlus.str</b>	Optional start-up file for the PATRAN to LUSAS translator.
<b>luspat.str</b>	Optional start-up file for the LUSAS to PATRAN translator.

The following files will be installed in a directory called **\$FEADIR/patran3** and its sub-directories.

<b>lusas.db</b>	Database template for the LUSAS preference.
<b>lusas.plb</b>	PCL library for the LUSAS preference.

<b>lusas_dis.res_tmpl</b>	Results template file.
<b>lusas_nod_els_2d_thermal.res_tmpl</b>	Results template file.
<b>lusas_nod_els_3d_thermal.res_tmpl</b>	Results template file.

<b>lusas_nod_els_membrane_stress.res_tmpl</b>	Results template file.
<b>lusas_nod_els_plane_strain.res_tmpl</b>	Results template file.
<b>lusas_nod_els_plane_stress.res_tmpl</b>	Results template file.
<b>lusas_nod_els_shells_stress.res_tmpl</b>	Results template file.
<b>lusas_nod_els_solid_axisymmetric.res_tmpl</b>	Results template file.
<b>lusas_nod_els_solid_stress.res_tmpl</b>	Results template file.
<b>pat3_lusas.ses</b>	Forward translator test session file.
<b>lusas_pat3.ses</b>	Reverse translator test session file.

If it is required to run PATLUS/LUSAS/LUSPAT from within PATRAN, the library of supplied files will need to be installed as root using the following procedure:

<b>Step One</b>	Go to the <b>patran3</b> subdirectory in your main LUSAS release area containing the required integration tools, for example:  <pre>cd \$FEADIR/patran3</pre>
<b>Step Two</b>	Copy the necessary integration tool files into the <b>\$P3_HOME</b> directory and its associated sub-directories <b>res_templates</b> and <b>test_files</b> , by executing the installation script by typing:  <pre>./p3install</pre>

## PATRAN to LUSAS (PATLUS)

The conversion of PATRAN data to LUSAS data is achieved using the PATRAN neutral file system. PATRAN is used to generate a neutral file which contains all of the information required to generate a valid LUSAS data deck. The conversion of the data is achieved using the program PATLUS. The input to PATLUS is the neutral file **filename.def** which is then translated to the LUSAS data deck **filename.dat**. PATLUS is invoked by typing **patlus** and specifying the root of the filename to be converted:

```
patlus filename
```

Alternatively, PATLUS and LUSAS may be run directly from within PATRAN using the scripts provided and detailed in this manual.

The elements and loading types supported are shown on page 18.



## LUSAS Data Deck

The LUSAS data deck generated by PATLUS will contain some or all of the following data sections:

<b>Problem Title</b>	This is read from the PATRAN neutral file.
<b>Options</b>	OPTIONS 2 117 118 are inserted automatically. Any further options should be input manually.
<b>Element Topology</b>	<p>The currently available elements are listed on page 18 together with their PATRAN identification and configuration numbers.</p> <p>For 2D models all element normals should be consistent throughout the structure.</p> <p>The PATRAN user should ensure that all element local axes are consistent, throughout the structure, with the (Hyper) Patch local axes.</p> <p>See the notes on beam and joint orientation at the end of this table.</p>
<b>Solution Order Automatic</b>	This data chapter is inserted automatically. The Sloan optimiser is selected by default.
<b>Node Coordinates</b>	This data chapter is generated automatically. PATLUS will automatically insert additional dummy nodes for the orientation of beams, joints and coordinate frames.
<b>Geometric Properties</b>	This data chapter is inserted automatically.
<b>Geometric Assignments</b>	This data chapter is inserted automatically.
<b>Material Properties</b>	<p>Isotropic Material Properties are available.</p> <p>Composite properties are not supported and should be input manually.</p>
<b>Material Assignments</b>	This data chapter is inserted automatically.

<p><b>Slideline Properties</b></p>	<p>This data chapter is inserted when slidelines are generated. All slidelines are initially assigned the default properties as defined in the <i>Modeller Reference Manual</i>.</p>
<p><b>Slideline_Surface Definition</b></p>	<p>This data chapter is inserted when slideline data is generated within PATRAN using pressure loading with reserved data values.</p> <p>Slidelines are assigned by applying pressures. A load case should be used exclusively for slidelines.</p> <p><b>Method 1 (INTSL3=0)</b></p> <p>Master slidelines are assigned a loadset-id in the range 1000-1999 and the corresponding slave slideline is assigned a loadset-id in the range 2000 - 2999 by adding 1000 to the master slideline loadset-id. The pressure value is ignored.</p> <p><b>Method 2 (INTSL3=1)</b></p> <p>Slidelines are assigned by inserting a code in the element pressure field. The magnitude of the first component of the applied pressure applied within this load case represents a code:</p> <p style="text-align: center;"><b>pressure = nnnst</b></p> <p>where:</p> <p><b>nnn</b> indicates slideline number. The number must be greater than 99).</p> <p><b>s</b> indicates whether it is a master (1) or a slave (2) slideline.</p> <p><b>t</b> indicates the slideline contact type. Options are: (1) sliding, (2) Sliding with friction, (3) Tied slidelines, (4) Sliding only with no lift-off.</p> <p>For example, slideline 111 allowing sliding with friction is defined by pressure load datasets of magnitude 11112 (master) and 11122 (slave).</p>
<p><b>Slideline Assignments</b></p>	<p>This data chapter is inserted when slidelines are generated. All slidelines are assigned property set 1.</p>

<p><b>Cartesian Sets</b></p>	<p>This data chapter is inserted automatically.</p> <p><b>Note.</b> Cartesian and cylindrical sets are supported but spherical sets are not currently supported in LUSAS.</p>
<p><b>Transformed Freedoms</b></p>	<p>This data chapter is inserted automatically.</p> <p><b>Note.</b> If a CID is assigned to a node in PATRAN the TRANSFORMED FREEDOMS will be used to reflect the transformation in LUSAS. This information is stored in the LUSAS database allowing the user to generate stress and displacement nodal results in either global or transformed directions.</p>
<p><b>Constraint Equations</b></p>	<p>This data chapter is inserted automatically when multi-point constraints (MPCs) are specified in PATRAN.</p>
<p><b>Support Nodes</b></p>	<p>This data chapter is inserted automatically.</p> <p><b>Notes</b></p> <ul style="list-style-type: none"> <li>• Each PATRAN data file must contain only one support case.</li> <li>• Non-standard nodal freedoms are translated in order, for example, a fully restrained semi-loof shell will restrain freedoms 1 to 5, whereas a fully restrained BMS3 will restrain freedoms 1 to 6.</li> </ul>

<p><b>Load Case</b></p>	<p>The currently available loading types for each element are listed on page 18.</p> <p><b>Notes</b></p> <ul style="list-style-type: none"> <li>• PATRAN does not allow constant body forces to be applied. This data chapter must, therefore, be input manually.</li> <li>• When non-standard freedoms are loaded, the loading should be entered in the order the freedoms appear in LUSAS, e.g. the semiloof elements require loof moments M1 and M2. These should be input in positions Mx and My.</li> <li>• For prescribed displacement loading the PATRAN constraint set identification number must be used to assign this loading type to the required load case.</li> <li>• For field elements the face load data input should be input to PATRAN in the C1 flag direction.</li> </ul>
<p><b>Analysis Control</b></p>	<p>Some analysis control options are available within PATRAN. See pages 8 and 16 for more information.</p>
<p><b>Element Output Control</b></p>	<p>All element output is suppressed by default. If specific output is required after the LUSAS run has completed this can be obtained using the LUSAS restart facility. For example:</p> <pre> RESTART READ 1 LOAD COMBINATION 1 1 OPTION 55 ELEMENT OUTPUT CONTROL 1 10 1 1 1 NODE OUTPUT CONTROL 1 20 1 3 END </pre>

<b>Node Output Control</b>	Only reactions are output by default. Displacements are suppressed.
<b>Restart Write Brief</b>	This LUSAS command is output automatically.
<b>End</b>	This LUSAS command is output automatically.

## Notes on Beam/Joint Orientation in PATRAN

There are two ways of orienting line elements in PATRAN:

- Node** Using a node to define the xy-plane.
- Local Coordinate System** Using a local coordinate system to define the local x axis and xy plane.

These are used with two different types of elements:

- Beam elements** These have a finite length and hence a node is sufficient to define the beam axes (x is taken as along the beam axis). All LUSAS beams in PATRAN are oriented using a method 1 above, but the node is an optional parameter. The usual rules apply regarding defaults.
- Joints** These are normally zero length, and hence require a local coordinate system to define both the local x axis and the xy plane. All LUSAS Joint elements in PATRAN are oriented using method 2 above, but the local coordinate is an optional parameter. The usual rules apply regarding defaults.

Element local axes are displayed from within the **Element Properties** dialog box. Use the following procedure:

1. Set the Action to **Show**. All of the element properties defined will be displayed.
2. Choose **X-Y Plane Orient. Node** for beams (see note 3 below), or **Orientation System** for joints (see note 4 below).
3. For beams, the following selections are available:
  - **Table** produces a report table.
  - **Marker Plot** marks the elements with their orientation node numbers.
  - **Scalar Plot** produces a fringe plot of orientation node numbers.
4. For joints, the following selections are available:
  - **Table** produces a report table.
  - **Vector Plot** marks the elements with a view of the local coordinate used to orient the joint, provided that the elements have been added to the appropriate Group and the **Functional Assignments Display** setting has been toggled to **Display on FEM Only**.

## LUSAS to PATRAN (LUSPAT)

The conversion of results from LUSAS format to PATRAN format is handled by the program LUSPAT. LUSPAT takes a LUSAS restart file and generates PATRAN results files for each load case and results type.

Since the results are created from the LUSAS restart file this should be first created by inserting RESTART WRITE BRIEF in the LUSAS data set. This line is written automatically by PATLUS.

### Interactive Use

If LUSPAT is to be used in an interactive mode the user will be prompted for the following information.

**File:**

The program will initially prompt the user for the input job file. Input the LUSAS filename without the **rst** extension.

**Enter model name (<CR>for jobname)\_:**

An optional model name is prompted for which is used to generate the results file names. If omitted the **jobname** is used. When run in batch mode the model name will default to the **jobname**.

**Enter RESTART DUMP number required (zero for all dumps)\_:**

The program will prompt for the RESTART DUMP number. For linear and eigenvalue extraction problems, dump number 1 should be specified. For nonlinear or transient problems all the dumps may be translated by specifying zero.

**Do you require eigen analysis + nonlinear results [N]\_:**

If an eigenvalue analysis has been preceded by a nonlinear analysis the program will inquire whether the eigen analysis plus the nonlinear results are to be output.

**Enter LOAD CASE/MODE required (zero for all loads)\_:**

For linear or eigen problems the load case/mode number must be input. If a zero is specified all load cases will be translated. If a load case is specified the prompt will be repeated until a carriage return is specified. When more than one load case/mode is specified the results will be combined with the corresponding load factors. Up to 50 load combinations can be processed at any time.

**Enter FACTOR required**

For each load case/mode specified a corresponding factor must be input. If all load cases are to be translated a factor of one is used.

**Enter STRUCTURE TYPE**

---

Beam	-1
Plane Membrane	-2
Space Membrane	-3
Plate Flexure	-4
Shell	-5
Axisymmetric/Plane strain	-6
Solid	-7
Field 2D	-8
Field 3D	-9
Field 1D	-10
Enter number __:	

To prevent invalid data being assembled at common nodes, i.e. between beams and shells, the structure type under consideration must be input. If a model contains two or more structure types the translation process should be repeated for each structure type required. If zero is specified auto structure type selection will be used. When several structure types are present the structure type selected will be the first identified in the results file.

**Enter material number required (zero for all materials)\_:**

Enter the selected material number as defined in the LUSAS material assignments to enable averaging of stresses for that material only. This feature allows selective nodal averaging of different parts of the structure and may be used, for example, to carry out selective nodal averaging of a branched shell. In this case each branch of the shell should be modelled in PATRAN with a different material assignment.

**Do you require nodal results in transformed directions [N]\_:**

The displacements and stresses may be output in the transformed directions as defined in the LUSAS transformed freedoms data chapter if required. The transformed freedom assignments in LUSAS are controlled by the CID of the nodes in PATRAN.

**Do you wish to continue [N]\_:**

Further load cases or modes can be processed by continuing. The default answer stops the program.

## Batch Mode

LUSPAT has been designed to be interactive but may be run in batch by setting the system parameter IBATCH=1 in the LUSPAT start-up file **lusp.at.str** as described on

page 30. The prompts will then take the following default values which may be overridden as indicated:

- All restart dumps will be translated, unless the system parameter **NDUMP** is set to the required dump number.
- All load cases will be translated.
- Auto structure type selection will be used, unless the system parameter **NSTRUT** is set to the required structure type number.
- All material types will be translated together, unless the system parameter **NMAT** is set to the required material type number.
- Eigenvector results will not be superimposed on the nonlinear deformation unless the system parameter **IEIGNL** is specified.
- Results will be translated in the global coordinate system, unless the system parameter **IRSTRF** is specified.

Full details of supported system parameters are given on page 30.

### File Naming Convention

The conversion used for the result file names produced is:

**model\_number.ext**

where **model** is the specified model name, **number** is the LUSAS restart dump number or the LUSAS load case or eigen mode number, and **ext** is the extension which indicates the file content. The extensions adopted are:

<b>name.def</b>	structure definition
<b>model_number.adp</b>	nodal displacements (.dsp for binary file)
<b>model_number.and</b>	nodal results (.ndl for binary file)
<b>model_number.ael</b>	element results (.elm for binary file)

where:

**name** is the **jobname** corresponding to the restart file name.

For example, **jobname** for the file **test.rst** is **test**.



**Note.** On certain machine types the documented file naming convention is not adhered to due to system limitations.

### Results File Titles

All results file titles are automatically assembled to include the problem title, the selected material number and the structure type. For nonlinear analysis the load increment and arc length load factor are output. For transient analysis the time step and



response time are output. For frequency analysis the mode number, eigenvalue, frequency and error norm are output. For buckling analysis the mode number and load factor are output.

The titles given in the displacement, nodal results and element results files take the form:

```
TITLE:  problem title
TITLE1: model_number structure_type results_type
        material_no
TITLE2: analysis data
```

where:

<b>problem title</b>	is as defined in the LUSAS data file
<b>model_number</b>	is as defined in the file name, i.e. test_001
<b>structure_type</b>	is as selected, i.e. shell
<b>results_type</b>	is the description of the results, e.g. TRANSFORMED AVERAGED RESULTS
<b>material_no</b>	is the material group selected
<b>analysis data</b>	is defined differently for each problem type. Options are: Linear (blank), Frequency (frequency, error norm), Buckling (load factor), Nonlinear (load increment, load factor), Transient (time step number, response time).

## Interface Integration with PATRAN

### Introduction

LUSAS has been integrated within PATRAN 3 and above using PCL This enables LUSAS to be run directly from within PATRAN using a single command or menu pick. From within PATRAN the user can use the LUSAS defined menus to set LUSAS up to run different analysis types, control solution optimisation, set output file options and set up the results types and files to be read.

The integrated program uses the following files (which should be installed in the directory **\$P3\_HOME**):

**lusas.plb** LUSAS PCL Library of menu functions.  
**lusas.db** LUSAS database template.

The LUSAS preference modifies the following main areas of PATRAN:

- Materials** Material properties are entered which generate the equivalent LUSAS MATERIAL PROPERTIES data chapters.
- Element Library** Elements are selected from those available in the LUSAS element library. See page 18.

- Loads and Boundary Conditions** LUSAS-style Loads, Supports and Constraint Equations can be entered.
- Analysis Forms** The analysis dialogs are configured to suit a LUSAS analysis, with options available for controlling the LUSAS solution procedure.

### Using Interfaces with PATRAN

The analysis and post-processing procedure may be controlled fully using the **Analysis** dialog box within PATRAN.

### PATRAN to LUSAS

Data is transferred from PATRAN to LUSAS by setting the **Action** on the **Analysis** dialog to **Analyze**. The following **Action** settings are available:

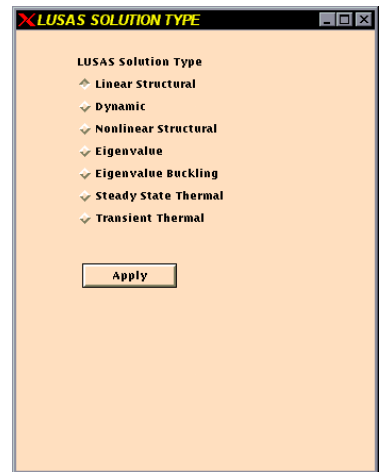
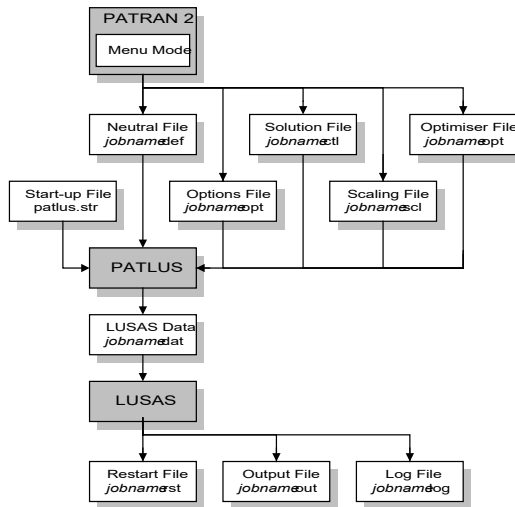
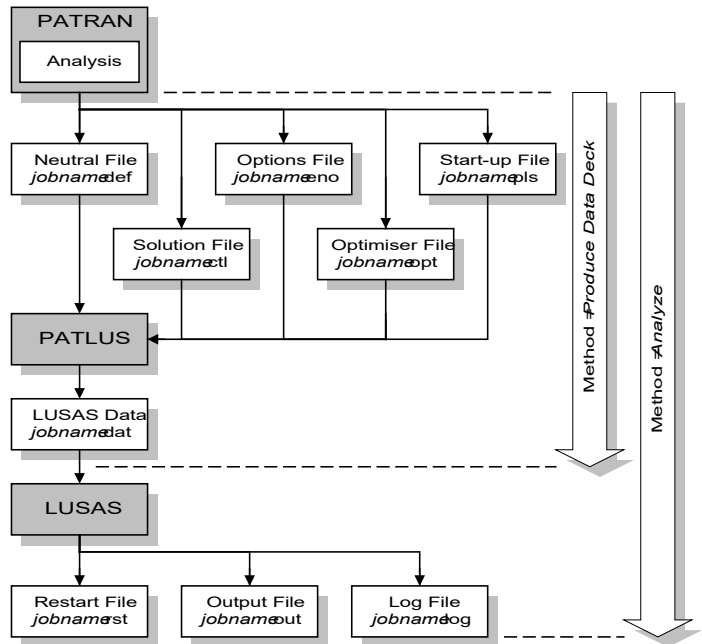
- Run Analysis** translates the data and carries out an analysis.
- Produce Data Deck** translates the data to a LUSAS data deck, but no analysis is carried out.

The type of analysis to be carried out is selected using the **Solution Type** dialog. This will generate a file **jobname.ctl** which is picked up by PATLUS. The default solution type is linear static.

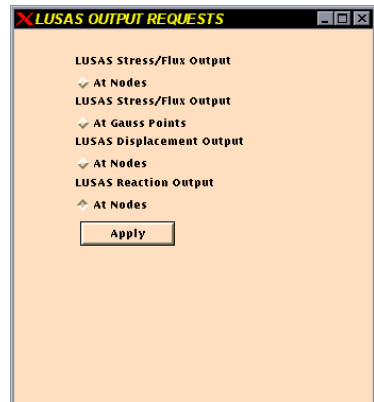
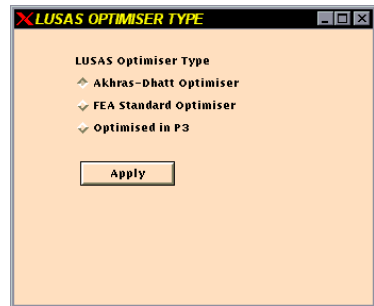
The type of frontwidth optimiser to be used during solution is selected using the **Optimiser Type** dialog. This will generate a file **jobname.opt** which is picked up by PATLUS. The default optimiser type is Sloan.

The amount of output written to the LUSAS text output file is controlled using the **Output Requests** dialog. This will generate a file **jobname.eno** which is picked up by PATLUS. The default is for output of reactions at nodes only. This does not affect the results available for viewing in PATRAN which are extracted from the restart file.

**PATRAN to LUSAS Translation Procedure**



**Analysis Dialog for  
PATRAN to LUSAS  
Subordinate dialogs  
shown right.**

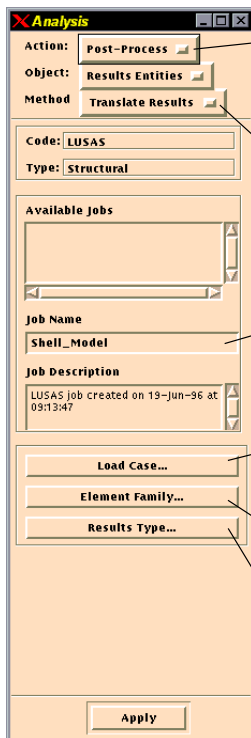
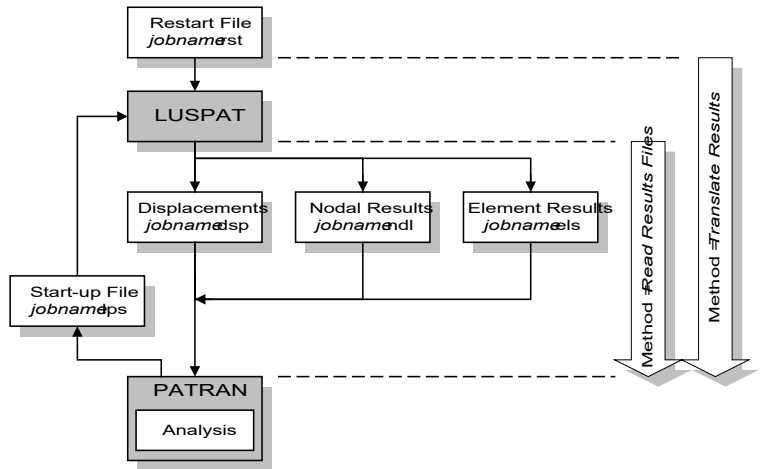


## **LUSAS to PATRAN**

Data is transferred from LUSAS to PATRAN by setting the **Action** options list on the **Analysis** dialog to **Post Process**. The Method setting is used as follows:

- Translate Results** LUSPAT will be invoked to generate results files from the LUSAS restart file **jobname.rst**.
- Read Results Files** it will be assumed that the relevant binary PATRAN results files already exist.

**LUSAS to PATRAN Translation Procedure**



Action is set **Post-Process** for LUSAS to PATRAN

**Translate Results** run LUSAS then re results

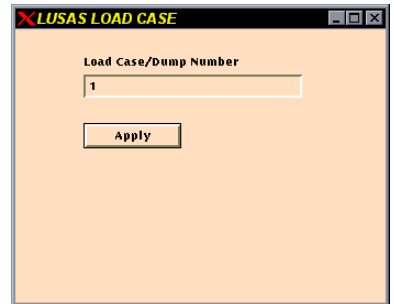
**Read Results Files** reads PATRAN format results files

**Job Name** used as root of all file name

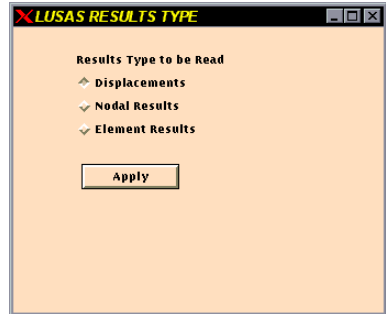
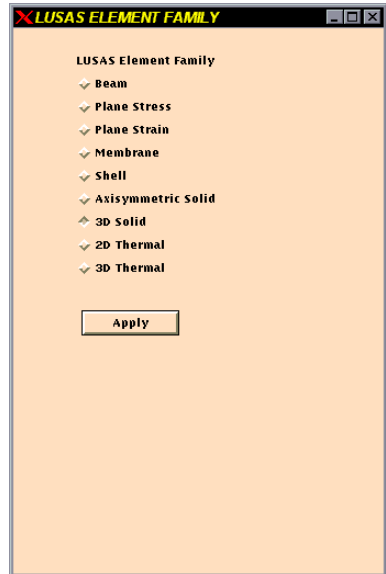
Click here to set the **Load Case** the results to be read

Click here to specify **Element Family** the results to be read

Click here to specify **Results Type** to be read



**Analysis Dialog for LUSAS to PATRAN Subordinate dialogs shown right**



**PATLUS Element Details**

The following tables detail which elements are supported by the PATRAN-LUSAS interface and also which loading types are applicable to each element configuration set.



**Note.** The non-shaded cells in the tables below indicate the elements supported in pre-PATRAN 3 versions of the interface.

BAR 2		Load Types				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
BAR2	1	Yes			Yes	Yes

<b>BAR 2</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>BRS2</b>	2	Yes			Yes	Yes
<b>BEAM</b>	3	Yes	Yes		Yes	Yes
<b>GRIL</b>	4	Yes	Yes		Yes	Yes
<b>BRP2</b>	5	Yes	Yes		Yes	Yes
<b>BMS3</b>	6	Yes	Yes		Yes	Yes
<b>BXM2</b>	7	Yes			Yes	Yes
<b>BXS2</b>	8	Yes				Yes
<b>BFD2</b>	9	Yes				Yes
<b>BFX2</b>	10	Yes				Yes
<b>BFS2</b>	11	Yes				Yes
<b>LFD2</b>	12	Yes				Yes
<b>LFX2</b>	13	Yes				Yes
<b>LFS2</b>	14	Yes				Yes
<b>BTS3</b>	17	Yes				Yes
<b>JNT3</b>	101	Yes				Yes
<b>JPH3</b>	102	Yes				Yes
<b>JF3</b>	103	Yes				Yes
<b>JRP3</b>	104	Yes				Yes
<b>JNT4</b>	105	Yes				Yes
<b>JL43</b>	106	Yes				Yes
<b>JSH4</b>	107	Yes				Yes
<b>JL46</b>	108	Yes				Yes
<b>JSL4</b>	109	Yes				Yes

<b>BAR 2</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>JAX3</b>	110	Yes				Yes
<b>JXS3</b>	111	Yes				Yes

<b>BAR 3</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>BAR3</b>	1	Yes			Yes	Yes
<b>BRS3</b>	2	Yes			Yes	Yes
<b>BM3</b>	3	Yes	Yes		Yes	Yes
<b>BSL3</b>	4	Yes	Yes		Yes	Yes
<b>BSL4</b>	5	Yes	Yes		Yes	Yes
<b>BS3</b>	6	Yes	Yes		Yes	Yes
<b>BS4</b>	7	Yes	Yes		Yes	Yes
<b>BMX3</b>	8	Yes	Yes		Yes	Yes
<b>BXS3</b>	9	Yes	Yes		Yes	Yes
<b>BXS4</b>	10	Yes	Yes		Yes	Yes
<b>BXL4</b>	11	Yes	Yes		Yes	Yes
<b>BXM3</b>	12	Yes			Yes	Yes
<b>BXE3</b>	13	Yes				Yes
<b>BFD3</b>	14	Yes				Yes
<b>BFX3</b>	15	Yes				Yes
<b>BFS3</b>	16	Yes				Yes



<b>TRI 3</b>		<b>Load Types</b>				
<b>Elem. Name</b>	<b>Config. No.</b>	<b>Nodal Point Load</b>	<b>Elem. Distrib. Load</b>	<b>Nodal Distrib. Load</b>	<b>Nodal/Elem. Load</b>	<b>Nodal Prescr. Value</b>
<b>TPM3</b>	1	Yes		Yes	Yes	Yes
<b>TAX3</b>	2	Yes		Yes	Yes	Yes
<b>TF3</b>	3	Yes	Yes	Yes	Yes	Yes
<b>TRP3</b>	5	Yes	Yes	Yes	Yes	Yes
<b>TSM3</b>	7	Yes	Yes	Yes	Yes	Yes
<b>TS3</b>	9	Yes	Yes	Yes	Yes	Yes
<b>TFD3</b>	13	Yes		Yes	Yes	Yes
<b>TXF3</b>	14	Yes		Yes	Yes	Yes
<b>TPN3</b>	16	Yes		Yes	Yes	Yes
<b>TPM3E</b>	17	Yes		Yes	Yes	Yes
<b>TAX3E</b>	18	Yes		Yes	Yes	Yes
<b>TPN3E</b>	19	Yes		Yes	Yes	Yes
<b>TAX3F</b>	20	Yes		Yes	Yes	Yes
<b>TTS3</b>	21	Yes		Yes	Yes	Yes

<b>TRI 6</b>		<b>Load Types</b>				
<b>Elem. Name</b>	<b>Config. No.</b>	<b>Nodal Point Load</b>	<b>Elem. Distrib. Load</b>	<b>Nodal Distrib. Load</b>	<b>Nodal/Elem. Load</b>	<b>Nodal Prescr. Value</b>
<b>TPM6</b>	1	Yes		Yes	Yes	Yes
<b>TAX6</b>	2	Yes		Yes	Yes	Yes
<b>TF6</b>	3	Yes		Yes	Yes	Yes
<b>TTF6</b>	4	Yes		Yes	Yes	Yes
<b>TSL6</b>	5	Yes		Yes	Yes	Yes
<b>TFD6</b>	6	Yes		Yes	Yes	Yes

<b>TRI 6</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>TXF6</b>	7	Yes		Yes	Yes	Yes
<b>TPK6</b>	8	Yes		Yes	Yes	Yes
<b>TXK6</b>	9	Yes		Yes	Yes	Yes
<b>TPN6</b>	16	Yes		Yes	Yes	Yes
<b>TNK6</b>	17	Yes		Yes	Yes	Yes
<b>TAX6F</b>	18	Yes		Yes	Yes	Yes
<b>TTS6</b>	19	Yes		Yes	Yes	Yes

<b>QUAD 4</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>QPM4</b>	1	Yes		Yes	Yes	Yes
<b>QAX4</b>	2	Yes		Yes	Yes	Yes
<b>QF4</b>	3	Yes	Yes	Yes	Yes	Yes
<b>QSC4</b>	4	Yes	Yes	Yes	Yes	Yes
<b>QRP4</b>	5	Yes	Yes	Yes	Yes	Yes
<b>RPI4</b>	6	Yes	Yes	Yes	Yes	Yes
<b>QSM4</b>	7	Yes	Yes	Yes	Yes	Yes
<b>SMI4</b>	8	Yes	Yes	Yes	Yes	Yes
<b>QS4</b>	9	Yes	Yes	Yes	Yes	Yes
<b>QSH4</b>	10	Yes	Yes	Yes	Yes	Yes
<b>SHI4</b>	11	Yes	Yes	Yes	Yes	Yes
<b>QSI4</b>	12	Yes	Yes	Yes	Yes	Yes
<b>QFD4</b>	13	Yes		Yes	Yes	Yes

<b>QUAD 4</b>		<b>Load Types</b>				
<b>Elem. Name</b>	<b>Config. No.</b>	<b>Nodal Point Load</b>	<b>Elem. Distrib. Load</b>	<b>Nodal Distrib. Load</b>	<b>Nodal/Elem. Load</b>	<b>Nodal Prescr. Value</b>
<b>QXF4</b>	14	Yes		Yes	Yes	Yes
<b>PMI4</b>	15	Yes		Yes	Yes	Yes
<b>QPN4</b>	16	Yes		Yes	Yes	Yes
<b>QPM4E</b>	17	Yes		Yes	Yes	Yes
<b>QAX4E</b>	18	Yes		Yes	Yes	Yes
<b>QPN4E</b>	19	Yes		Yes	Yes	Yes
<b>QAX4F</b>	20	Yes		Yes	Yes	Yes
<b>QTS4</b>	21	Yes		Yes	Yes	Yes
<b>QPM4M</b>	22	Yes		Yes	Yes	Yes
<b>QPN4M</b>	23	Yes		Yes	Yes	Yes
<b>QAX4M</b>	24	Yes		Yes	Yes	Yes

<b>QUAD 8</b>		<b>Load Types</b>				
<b>Elem. Name</b>	<b>Config. No.</b>	<b>Nodal Point Load</b>	<b>Elem. Distrib. Load</b>	<b>Nodal Distrib. Load</b>	<b>Nodal/Elem. Load</b>	<b>Nodal Prescr. Value</b>
<b>QPM8</b>	1	Yes		Yes	Yes	Yes
<b>QAX8</b>	2	Yes		Yes	Yes	Yes
<b>QF8</b>	3	Yes	Yes	Yes	Yes	Yes
<b>QTF8</b>	4	Yes	Yes	Yes	Yes	Yes
<b>QSL8</b>	5	Yes	Yes	Yes	Yes	Yes
<b>QFD8</b>	6	Yes		Yes	Yes	Yes
<b>QXF8</b>	7	Yes		Yes	Yes	Yes
<b>QPK8</b>	8	Yes		Yes	Yes	Yes
<b>QXK8</b>	9	Yes		Yes	Yes	Yes

<b>QUAD 8</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>QPN8</b>	16	Yes		Yes	Yes	Yes
<b>QNK8</b>	17	Yes		Yes	Yes	Yes
<b>QAX8F</b>	18	Yes		Yes	Yes	Yes
<b>QTS8</b>	19	Yes		Yes	Yes	Yes

<b>QUAD 12</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>QP12</b>	1	Yes		Yes	Yes	Yes
<b>QA12</b>	2	Yes		Yes	Yes	Yes

<b>TET 4</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>TH4</b>	1	Yes				
<b>TF4</b>	2	Yes		Yes	Yes	Yes
<b>TH4E</b>	3	Yes		Yes	Yes	Yes

<b>TET 10</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>TH10</b>	1	Yes		Yes	Yes	Yes
<b>TF10</b>	2	Yes		Yes	Yes	Yes

<b>WEDG 6</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>PN6</b>	1	Yes		Yes	Yes	Yes
<b>PF6</b>	2	Yes		Yes	Yes	Yes
<b>PN6E</b>	3	Yes		Yes	Yes	Yes

<b>WEDG 12</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>PN12</b>	1	Yes		Yes	Yes	Yes
<b>PF12</b>	2	Yes		Yes	Yes	Yes

<b>WEDG 15</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>PN15</b>	1	Yes		Yes	Yes	Yes
<b>PF15</b>	2	Yes		Yes	Yes	Yes

<b>HEX 8</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>HX8</b>	1	Yes		Yes	Yes	Yes
<b>HF8</b>	2	Yes		Yes	Yes	Yes
<b>HX8E</b>	3	Yes		Yes	Yes	Yes
<b>HX8M</b>	4	Yes		Yes	Yes	Yes

<b>HEX 16</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>HX16</b>	1	Yes		Yes	Yes	Yes
<b>HF16</b>	2	Yes		Yes	Yes	Yes

<b>HEX 20</b>		<b>Load Types</b>				
Elem. Name	Config. No.	Nodal Point Load	Elem. Distrib. Load	Nodal Distrib. Load	Nodal/Elem. Load	Nodal Prescr. Value
<b>HX20</b>	1	Yes		Yes	Yes	Yes
<b>HF20</b>	2	Yes		Yes	Yes	Yes

## LUSPAT Results Columns

### Displacement Results

The displacement results files have the following column definitions. Displacements are in global directions unless transformed directions have been selected.

**X, Y, Z** Translation      **THX, THY, THZ** Rotations      **T** Field values

Col.	Value	Col.	Value
<b>1</b>	X translation or T nodal temperature	<b>4</b>	THX rotation
<b>2</b>	Y translation	<b>5</b>	THY rotation
<b>3</b>	Z translation	<b>6</b>	THZ rotation

### Average Nodal and Element Results

The averaged nodal results and element results have the following column definitions. Averaged nodal results are in global directions unless transformed directions have been selected.

**SX, SY, SZ** Direct stresses      **SE** Von Mises equivalent stress

**SXY, SYZ, SXZ** Shear stresses      **S1, S2, S3** Principal stresses

**Nodal/Element Results Beam**

Not supported.

### Nodal/Element Results Plane Membrane

Col.	Value	Col.	Value
1	Stress in X direction (SX)	4	Von Mises stress (SE)
2	Stress in Y direction (SY)	5	Maximum principal stress (S1)
3	Stress in XY direction (SXY)	6	Minimum principal stress (S2)

### Nodal/Element Results Space Membrane

Col.	Value	Col.	Value
1	Stress in X direction (SX)	6	Stress in ZX direction (SZX)
2	Stress in Y direction (SY)	7	Von Mises stress (SE)
3	Stress in Z direction (SZ)	8	Maximum principal stress (S1)
4	Stress in XY direction (SXY)	9	Principal stress (S2)
5	Stress in YZ direction (SYZ)	10	Minimum principal stress (S3)

### Nodal/Element Results Plate Flexure

Col.	Value	Surface	Col.	Value	Surface
1	Stress in X direction (SX)	Top	7	Stress in X direction (SX)	Bottom
2	Stress in Y direction (SY)	Top	8	Stress in Y direction (SY)	Bottom
3	Stress in XY direction (SXY)	Top	9	Stress in XY direction (SXY)	Bottom
4	Von Mises stress (SE)	Top	10	Von Mises stress (SE)	Bottom
5	Max principal stress (S1)	Top	11	Max principal stress (S1)	Bottom
6	Min principal stress (S2)	Top	12	Min principal stress (S2)	Bottom

### Nodal/Element Results Shells

Col.	Value	Surface	Col.	Value	Surface
1	Stress in X direction (SX)	Top	11	Stress in X-direction (SX)	Bottom



2	Stress in Y direction (SY)	Top	12	Stress in Y-direction (SY)	Bottom
3	Stress in Z direction (SZ)	Top	13	Stress in Z-direction (SZ)	Bottom
4	Stress in XY direction (SXY)	Top	14	Stress in XY direction (SXY)	Bottom
5	Stress in YZ direction (SYZ)	Top	15	Stress in YZ direction (SYZ)	Bottom
6	Stress in ZX direction (SZX)	Top	16	Stress in ZX direction (SZX)	Bottom
7	Von Mises stress (SE)	Top	17	Von Mises stress (SE)	Bottom
8	Max principal stress (S1)	Top	18	Max principal stress (S1)	Bottom
9	Principal stress (S2)	Top	19	Principal stress (S2)	Bottom
10	Min principal stress (S3)	Top	20	Min principal stress (S3)	Bottom

**Nodal/Element Results Axisymmetric Solid/Plane Strain**

Col	Value	Col.	Value
1	Stress in X direction (SX)	5	Von Mises stress (SE)
2	Stress in Y direction (SY)	6	Maximum principal stress (S1)
3	Stress in XY direction (SXY)	7	Minimum principal stress (S2)
4	Hoop stress		

**Nodal/Element Results Solids**

Col.	Value	Col.	Value
1	Stress in X-direction (SX)	6	SZX Stress in ZX-direction
2	Stress in Y-direction (SY)	7	SE Von Mises stress
3	Stress in Z-direction (SZ)	8	S1 Maximum principal stress
4	SXY Stress in XY-direction	9	S2 Principal stress
5	SYZ Stress in YZ-direction	10	S3 Minimum principal stress

**Nodal/Element Results Field (1D)**

Not supported.

### Nodal/Element Results Field (2D)

Col.	Value
1	dT(X) Gradient in X direction
2	dT(Y) Gradient in Y direction

### Nodal/Element Results Field (3D)

Col.	Value
1	dT(X) Gradient in X direction
2	dT(Y) Gradient in Y direction
3	dT(Z) Gradient in Z direction

## System Parameters

When using the LUSAS to PATRAN interface default responses may be overridden with system parameters by specifying values in a start-up file (**patlus.str/luspat.str**). The start-up file should be created in the working directory and should be formatted as in the example shown below:

```

SYSTEM
ISTDEF=1
IELSTR=1
EXIT
    
```

The following system parameters are supported:

Parameter	Default	Description
<b>INTFP3</b>	0	PATRAN Version identifier (0 for P2.5, 1 for P3).
<b>IASCII</b>	1	Format for output of results files. (0 for binary). The results files default to an ASCII format but may be selected as binary by specifying the system parameter IASCII=0 (this will be carried out automatically when running LUSPAT from within PATRAN). ASCII PATRAN results files are easily transferable between different machine types and can be modified to change the title information if required. Binary files require less disc storage and can be read faster by PATRAN but cannot be modified directly. The READER utility supplied with PATRAN permits conversion between ASCII and binary formats if required.
<b>IBATCH</b>	0	Switch to allow LUSPAT to be run in batch with all prompting suppressed (1 for batch operation).

Parameter	Default	Description
<b>ISLIDE</b>	1	Switch to assume that when run in batch, face loads with the prescribed value should be converted to slides (0 for no conversion).
<b>NDUMP</b>	0	Restart dump number selected.
<b>NSTRUT</b>	0	Structure type number for stress averaging.
<b>NMAT</b>	0	Material group number for stress averaging.
<b>IEIGNL</b>	0	Switch to add eigenvectors to displacements for nonlinear problems. (1 adds eigenvectors to nonlinear displacements)
<b>ISTDEF</b>	0	Switch to assemble PATRAN definition file for elements topology and node coordinates (1 creates <b>.def</b> file). The structure topology and coordinates are not translated into PATRAN format by default but may be required when a model not generated by PATRAN is to be processed subsequently. To generate a PATRAN definition file set the system parameter ISTDEF=1. The node and element records of structure definition will be output in a file with the <b>.def</b> extension. Transformed freedoms are not supported.
<b>INDDSP</b>	1	Assemble displacement file (0 suppresses displacement results). The displacements or temperatures (field values) are translated by default. To suppress their creation set the system parameter INDDSP=0.
<b>INDSTR</b>	1	Switch to assemble averaged nodal results (0 suppresses averaged nodal results). The averaged nodal results for the selected structure type are translated by default. To suppress their creation set the system parameter INDSTR=0. The column definitions for the average nodal results are described in tables starting on page 26.
<b>IELSTR</b>	0	Switch to assemble element centroid results (1 outputs element centroid results). The values at element centroid positions for the selected structure are not translated by default. To translate element values set the system parameter IELSTR=1. The column definitions for the element results are described on page 26.
<b>IRSTRF</b>	0	Switch to output results in transformed directions (1 outputs results in transformed directions)
<b>ICNGDF</b>	0	Default element configuration code to use when zero configuration code encountered. If not specified PATLUS will prompt for an element name when a zero configuration

Parameter	Default	Description
		code is encountered.

# FEMView and FAM Interface

## Introduction

This section describes the use of FVLUSAS, the LUSAS Solver to FEMView/FAM export program supported by LUSAS.

The program reads results from the LUSAS restart file. The LUSAS restart file is named **jobname.rst**. A restart file is created by LUSAS using the RESTART WRITE or RESTART WRITE BRIEF commands in the data file or using the appropriate OUTPUT option in various analysis control sections in the data file.

FVLUSAS creates a sequential ASCII file named **model.prv**.

## LUSAS to FEMView/FAM

### Running The Program

The interface is designed to be interactive and hence should be initiated from a command prompt. Before data processing begins the interface prompts for certain information allowing you to select your output requirements. The prompts are outlined in the next few paragraphs:

**Enter FEMVIEW model name (up to 6 characters)\_:**

Enter the dataset name. The default model name is the current jobname.

**Is this a new FEMVIEW model [Y]\_:**

Identify the data set as a new PREVIEW data set or, if you wishes to append additional information to an existing PREVIEW model, an existing data set.

**Input dump number required (zero for all dumps) [0]\_:**

---

By default all the restart dumps will be processed. In linear runs all the results are stored in restart dump 1. For nonlinear or dynamic analysis the restart dump numbers are as specified by the user in the analysis control data.

The remaining options refer to the type of information to be transferred to the Preview data file. All results are not available for all element types. Consult the section titled *FEMView/FAM Results Availability* on page 36 for full details.

## Generation of Load Case Names

FEMView identifies each load case by a six character name. The interface will generate these load case names automatically and you have no control over the chosen names.

- Linear** For linear problems, load cases will be named LC0001, LC0002 etc.
- Eigenvalue** For eigenvalue problems, eigenvectors will be named LC0001, LC0002 etc.
- Nonlinear** For nonlinear transient field and step-by-step dynamic problems only one load case per restart dump is written. For such problems the result dumps are referred to as D01L01, D02L01 etc. where D refers to the dump number, e.g. D01 refers to dump 1, and L refers to the load case number within that dump.

## Average Stress Results Grouping

Since it is invalid to average stress results across material discontinuities, average stress results are available by material groups only. These groups are defined by the material assignment numbers in the LUSAS data deck.

## Supported Elements

For convenience in specifying the available results supported LUSAS elements have been divided into groups. These groups are shown in the following table.

No	Group Name	LUSAS Elements
1	Solids	HX8, HX16, HX20, PN6, PN12, PN15, TH4, TH10, HX8M, HX8E, PN6E, TH4E
2	Space Membrane	TSM3, QSM4, SMI4
3	Plate Membrane	TPM3, TPM6, QPM4, QPM8, TPM3E, QPM4E
4	Axisymmetric Solid/Plane Strain	TAX3, TAX6, QAX4, QAX8, TAX3E, QAX4E, TPN3, TPN6, QPN4, QPN8, TPN3E, QPN4E

No	Group Name	LUSAS Elements
5	Thick Plate Flexure	TTF6, QTF8, QSC4, QSC8
6	Thin Plate Flexure	TF3, TF6, QF4, QF8
7	Ribbed Plate	TRP3, QRP4, RPI4
8	Semi-loof Shell	TSL6, QSL8
9	Flat Shell	QSH4, QSH6, SHI4, SHI6
10	Bar	BAR2, BAR3, BRS2, BRS3
11	2D Beam Explicit Integration	BEAM
12	Grillage Explicit Integration	GRIL
13	Ribbed Beam Explicit Integration	BRP2
14	3D Beam Explicit Integration	BMS3
15	2D Thin Beam	BM3
16	3D Thin Beam	BS3, BS4, BXS4, BSL3, BSL4, BXL4
17	3D Thick Beam	BTS4, BTS6, BTW6, BOX6
18	Joint	JNT4, JNT3, JPH3, JAX3, JF3, JRP3, JSL4, JSH4, JXS3, JL43, JL46, LFD2, LFX2, LFS2
19	2D Field	TFD3, TFD6, QFD4, QFD8, TXF6, QXF4, QXF8, QXF9
20	Axisymmetric Sheet	BXM2, BXM3
21	Axisymmetric Shell	BXS3, BXE3
22	3D Field	HF8, HF16, HF20, PF6, PF12, PF15, TF4, TF10
23	1D Field	BFD2, BFD3, BFX2, BFX3, BFS2, BFS3
24	Fourier Elements	TAX3F, TAX6F, QAX4F, QAX8F
25	Thick Shell Elements	TTS3, QTS4, TTS6, QTS8

The group numbers are broadly dependent on the stress/stress resultant output produced from the analysis and determine which results data is available.

## FEMView/FAM Results Availability

No	Nodal I Disp.	Nodal Temp.	Avge. Nodal Stress	Nodal Prin. Stress	Nodal von Mises Stress	Elem. General Stress	Elem. Stress
<b>1</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>2</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>3</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>4</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	n/a	<b>Yes</b>
<b>5</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>6</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>7</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>8</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>9</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>
<b>10</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>11</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>12</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>13</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>14</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>15</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>16</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>17</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	<b>Yes</b>	n/a
<b>18</b>	<b>Yes</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	n/a
<b>19</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	n/a	<b>Yes</b>
<b>20</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	n/a	<b>Yes</b>
<b>21</b>	<b>Yes</b>	n/a	n/a	n/a	n/a	n/a	<b>Yes</b>
<b>22</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	n/a	<b>Yes</b>
<b>23</b>	n/a	<b>Yes</b>	n/a	n/a	n/a	n/a	n/a
<b>24</b>	n/a	n/a	n/a	n/a	n/a	n/a	n/a



No	Nodal Disp.	Nodal Temp.	Avg. Nodal Stress	Nodal Prin. Stress	Nodal von Mises Stress	Elem. General Stress	Elem. Stress
<b>25</b>	<b>Yes</b>	n/a	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>	<b>Yes</b>



**Notes**

- Since joint elements are not supported by FEMView any results for joints will be ignored.
- Displacements are always output in the system global axis set.
- Element general stresses are available for all beam, plate and shell elements and some other elements. For these elements these stresses refer to the local element axis set, which is determined by the node order as specified in the LUSAS element topology. For shells and plate flexure elements the stresses at the mid-surface and bending moments are available.
- Element stress resultants are available for all but the 3D and axisymmetric solid elements.

**FEMView/FAM Stress Types**

No	Stress Components						No	Stress Components					
<b>1</b>	$\sigma_X$	$\sigma_Y$	$\sigma_Z$	$\sigma_{XY}$	$\sigma_{YZ}$	$\sigma_{XZ}$	<b>14</b>	Fx	Fy	Fz	Mx	My	Mz
<b>2</b>	Nx	Ny	Nxy				<b>15</b>	Fx	Mz				
<b>3</b>	sX	sY	sXY				<b>16</b>	Fx	Mx	My	Mz		
<b>4</b>	sX	sY	sZ	sXY			<b>17</b>	Fx	Fy	Fz	Mx	My	Mz
<b>5</b>	MX	MY	MXy	SX	SY		<b>18</b>	-					
<b>6</b>	MX	MY	MXy				<b>19</b>	gX	gY	gXY			
<b>7</b>	Nx	Ny	Nxy	Mx	My	Mxy	<b>20</b>	Fx					
<b>8</b>	Nx	Ny	Nxy	Mx	My	Mxy	<b>21</b>	Fx	Mz				
<b>9</b>	Nx	Ny	Nxy	Mx	My	Mxy	<b>22</b>	gX	gY	gZ	gXY	gYZ	gZX
<b>10</b>	Fx						<b>23</b>	gx					

No	Stress Components					No	Stress Components					
<b>11</b>	Fx	Fy	Mz			<b>24</b>	-					
<b>12</b>	Fx	Mx	My			<b>25</b>	Nx	Ny	Nxy	Mx	My	Mxy
<b>13</b>	Fx	Fy	Fz	Mx	My							

### Table Key

<b>F</b>	Force	$\sigma$	Stress
<b>N</b>	Force Resultant	$g$	Field Gradient
<b>M</b>	Moment	<b>X, Y, Z</b>	Global directions are indicated by capital letters
<b>S</b>	Shear Force	<b>x, y, z</b>	Local direction results are indicated by lower case letters



### Notes

- Strains may be output instead of stresses by setting ISTRN=1 in the fvlusas.str file
- To change from I5 to I10 digit node and element numbers set IFMT=1 in the fvlusas.str file.
- To write high precision node coordinates with 10 digit node and element numbers set IFMT=2 in the fvlusas.str file.