



INTRODUCTION

EnSight supports files in the results (.rst, .rmg, .rfl, .rth, etc.) format as written from ANSYS (a commercial FEM solver). Versions 5.0–5.6 are supported. Note that not all element types are supported (although most are). Certain variables may read more slowly than others. While variables such as displacements and nodal solution variables read quickly, stress and strain require additional calculations in order to be useful to EnSight.

EnSight also supports results from the FLOTRAN CFD solver from ANSYS. FLOTRAN results are written to .rfl files.

Reading data into EnSight is a two-step process. First, the appropriate files are selected. This step is largely the same regardless of the format of the data being read. Second, parts are constructed using an interface that is specific to the applicable data format. This article covers the second step for ANSYS data. See [How To Read Data](#) for more information on selecting the appropriate files.

ANSYS datasets consist of the following files. Note that the entry in the File Name column is only a suggestion – it typically does not matter to EnSight what the actual file name is.

File	File Name	Notes	Required?
Geometry	file.rst or file.rth or file.rmg or file.rfl	Contains coordinates, element connectivity, and variables	required

BASIC OPERATION

After you have specified the appropriate data files with the File Selector (opened with File > Data (Reader)... as discussed in [How To Read Data](#)) and clicked Okay, the Data Part Loader (ANSYS) dialog will open. You use this dialog to build the desired parts. To build parts for ANSYS format data:

1. If the Data Part Loader dialog is not open, select File > Data (Part Loader)...

All parts defined in the .rst file will be loaded to the EnSight server. However, you have a choice for the initial visual representation of some parts as displayed on the client. The choice is made with the Load pull-down:

All Parts: all parts are loaded to the client in the default visual representation (typically 3D Border, 2D Full).

Part 1 Only: Only the first part is loaded to the client in the default visual representation. The other parts will have the NonVisual representation.

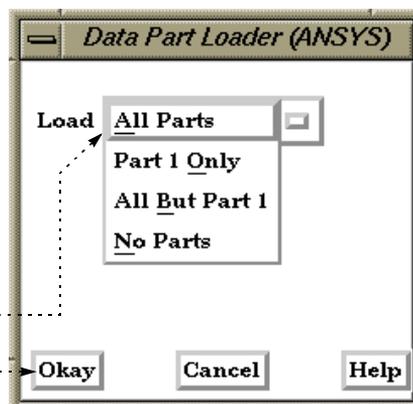
All But Part 1: All parts *other* than part 1 are loaded to the client in the default visual representation. Part 1 will be NonVisual.

No Parts: No parts are loaded to the client (*i.e.* the representation of all parts is set to NonVisual).

Note that you can easily change the visual representation of a part at any time. See [How To Change Visual Representation](#) for more information.

2. Select the desired Load option.

3. Click Okay.





OTHER NOTES

You can reopen the Data Part Loader dialog at any time to build additional parts. Simply select File > Data Part Loader)... and build the desired parts as described above.

By default, EnSight includes all possible variables, so most users will not need to deal with the .var file described below. However, since many of the stress variables may actually be zero, and because the list of variables is quite long, there is a way to filter which variables will be accepted into EnSight. Simply provide a .var file as described below.

```
#-----
# EnSight .var file - for use with ANSYS results files
#-----
#
# Place in the same directory as the .rst file.
#
# example:  If your result file were:      my_file.rst
#           make a file like this and name it:  my_file.var
#-----
# Below:
# 0 = variable is turned off,
# 1 = variable is turned on
#
# The first column is significant:
# #, space, or a newline will ignore the whole line. (Comment)
#
# The variable keys like "_eqv" or "displacement" must start in 1st column.
#
# The stress and strain variable flags on a line (0's and 1's), must come
# in the order shown, but are free format.
#
# For Contact:  The line indicated as:      Will be recognized as:
# -----
#               _x                          _stat
#               _y                          _pene
#               _z                          _pres
#               _xy                         _stot
#               _yz                         _slid
#
# Any extended DOF scalars that are not recognized will be created.
#
# (Note that components or vector sum for vector variables like
# "displacement" are controlled within EnSight)
#-----
# stress and strain variables:  (Created if flagged on here AND any element
# ===== nodal results in the ANSYS results file)
# stress total elast plast creep thermal
# strain strain strain strain strain strain contact
# -----
_x 0 0 0 0 0 0 1
_y 0 0 0 0 0 0 1
_z 0 0 0 0 0 0 1
_xy 0 0 0 0 0 0 1
_yz 0 0 0 0 0 0 1
_xz 0 0 0 0 0 0 0
_1 1 1 0 0 0 0 0
_2 1 1 0 0 0 0 0
_3 1 1 0 0 0 0 0
_int 1 1 0 0 0 0 0 0
_eqv 1 1 0 0 0 0 0 0
#
# DOFs:  (Created if flagged on here AND in the ANSYS result file)
# =====
displacement 1 # UX, UY, UZ
rotation 1 # ROTX, ROTY, ROTZ
acceleration 1 # AX, AY, AZ
velocity 1 # VX, VY, VZ
#
PRES 1
TEMP 1
VOLT 1
MAG 1
ENKE 1
ENDS 1
EMF 0
CURR 0
```

SEE ALSO

[How To Read Data](#)

User Manual: [ANSYS RESULTS Reader](#)

