

Interface for ANSYS

Note: If you use the V4W then you can perform easily the datatransfer between winLIFE and ANSYS to watch stress and fatigue results. Instead of the V4W you can use ANSYS macros and the use is described following. Using ANSYS macros is a more flexible solution and it may be in some special cases more convenient.

Interface winLIFE to ANSYS (V4.0)

This special interface has been developed by AWOTEC GmbH and Steinbeis.

The ANSYS interface is a collection of APDL (ANSYS Parametric Design Language) macros making the export of the FE results and the import of the winLIFE results possible. You can also then view these results.

The pre-processing and post-processing can be carried out in the classic ANSYS user surface "Mechanical APDL" and also in the work platform "ANSYS Workbench".

Below are the individual menus for interactive working followed by the enter sizes of the macros used. The macros can be integrated in an automatic program carried out by APDL Script.

Improvements compared to Version V3.2

- Now includes the export of element stresses for shell elements
- Error corrected in export of node stresses for shell elements

Runtime environment for the macros

The interface has been tested under the following conditions:

ANSYS Release V17.0

Supported element types:

Solid: 45, 92, 95, 185, 186, 187, 190

Plane: 42, 182, 183

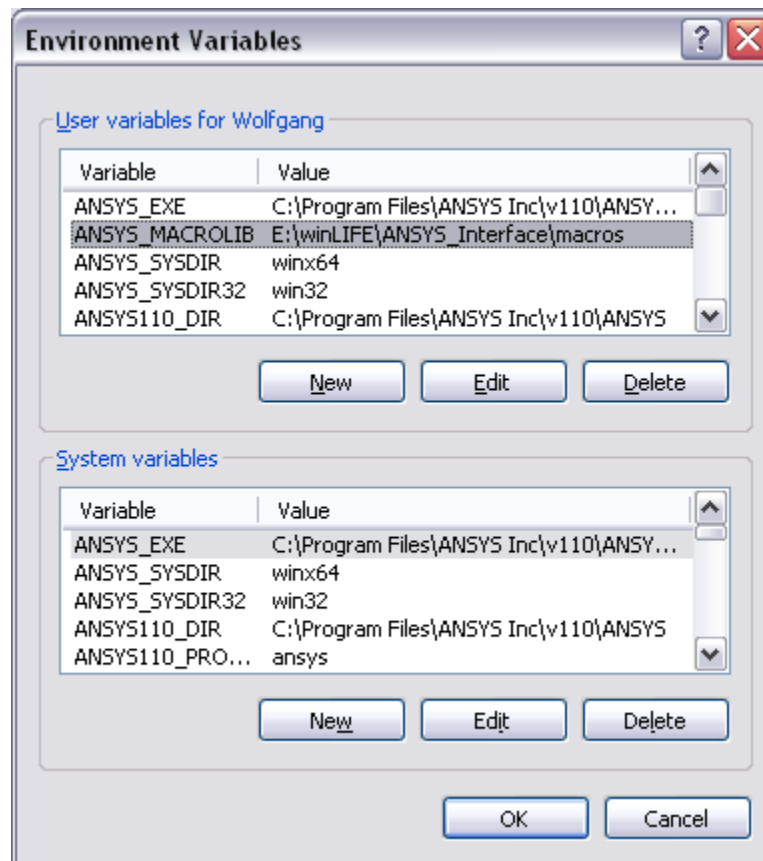
Shell: 41, 43, 63, 93, 181, 281

Supported calculation types:

Uniaxial and multiaxial; welded seams for solids and shells.

Installation

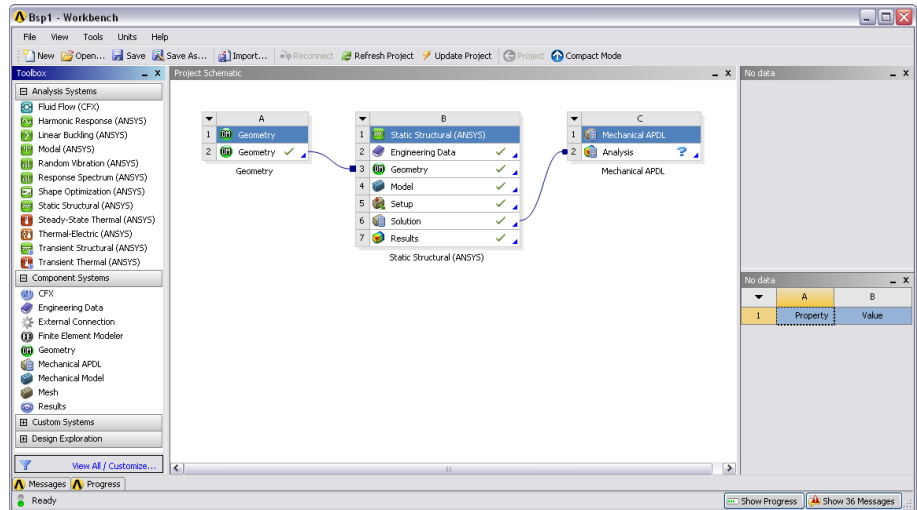
The file for the interface macros "ANSYS to winLIFE" has to be via the system variable ANSYS_macrolib defined in the environment variables of the Windows control system. The macros can be found on the winLIFE-CD.



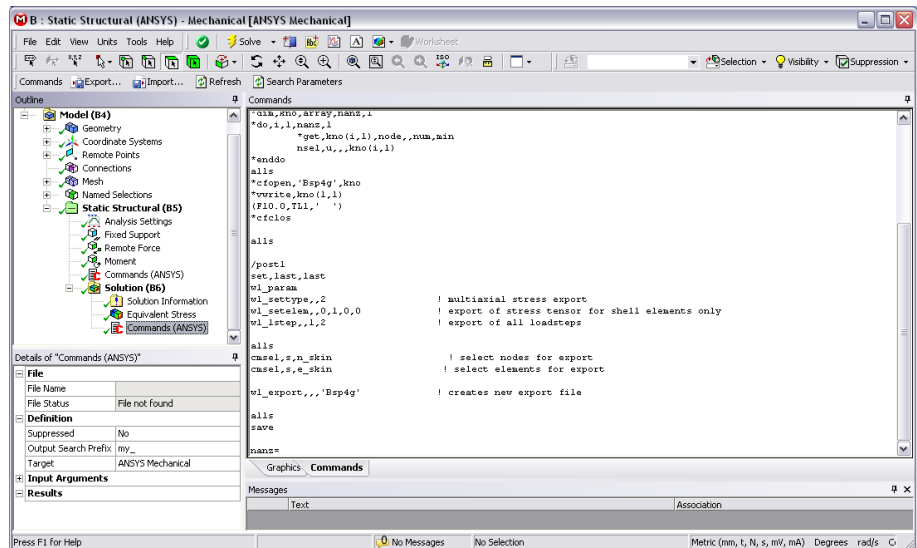
Preprocessing in ANSYS Workbench

Project Schematic Workflow

For the interactive use of the interface it is necessary to link an object from the *Mechanical APDL* with the solution from the *Mechanical* Object (Simulation). It is best if this link is added before the start of the solution!



The data export can also be carried out, however, via the interface macros directly into *Commands* Objects in the *Mechanical* Structure tree. *Mechanical APDL* is then only necessary to show the results.

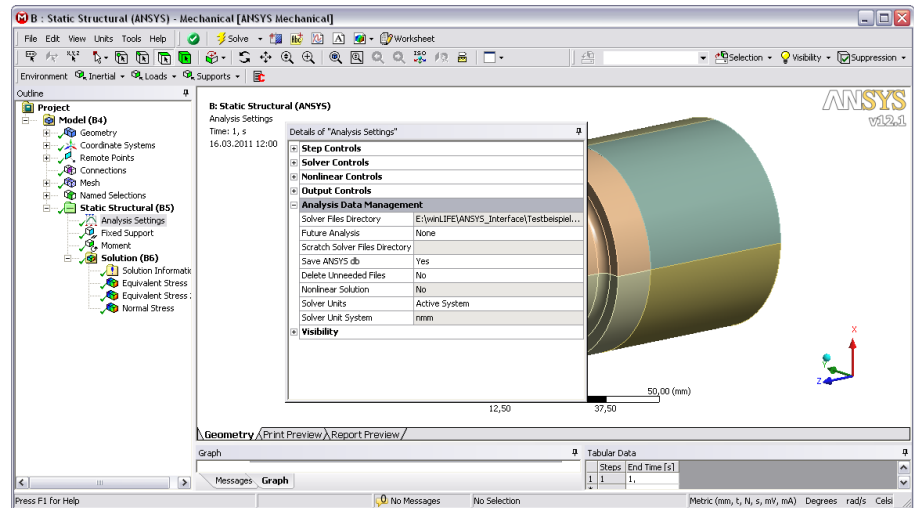


Before starting the equation solver with the command "SOLVE" in the *Mechanical* surface, it is necessary to make the following settings in the ANSYS settings in the detail window:

save ANSYS db: YES

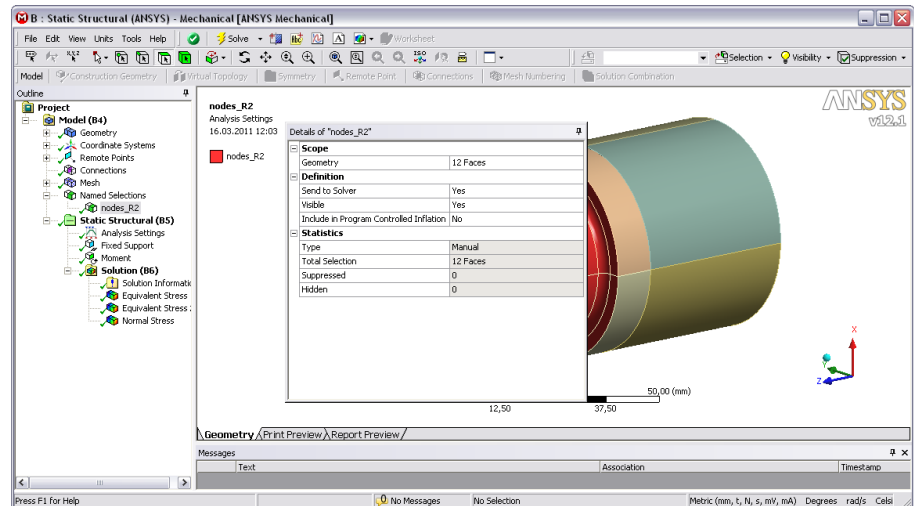
Delete Unneeded Files: NO

If the winLIFE Viewer is used subsequently, the data basis in the *.cdb format has to be saved in a Command Snippet using the command “cdwrite,db,”



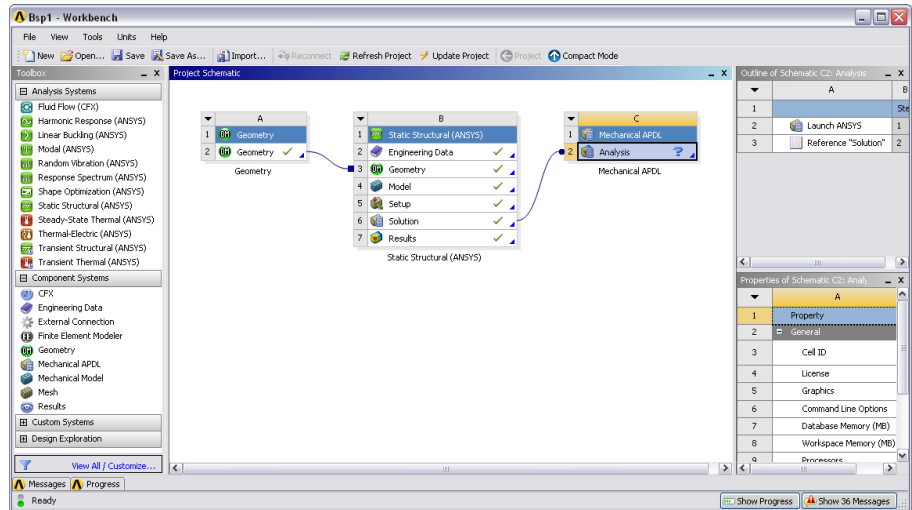
Named Selections

All nodes and elements, for which an evaluation is to be carried out, must be defined by Named Selections (Components). When doing this it is important to set the definition "Send to Solver" in the detail window to YES.



Starting the Interface

To start the interface, the *Mechanical APDL* window must be opened using the RMB-menu with the command "Edit in Mechanical APDL".



Then enter the following commands in the command line.

RESUME

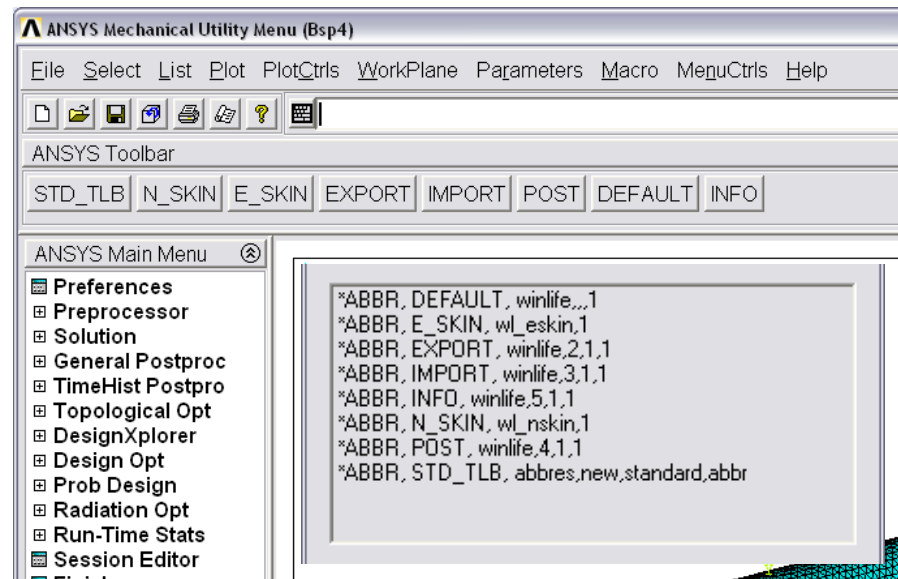
WINLIFE,1

The winLIFE Toolbar

For interactive use there are several command lines in the *Mechanical APDL*. These can be activated with the command **WINLIFE,1**.

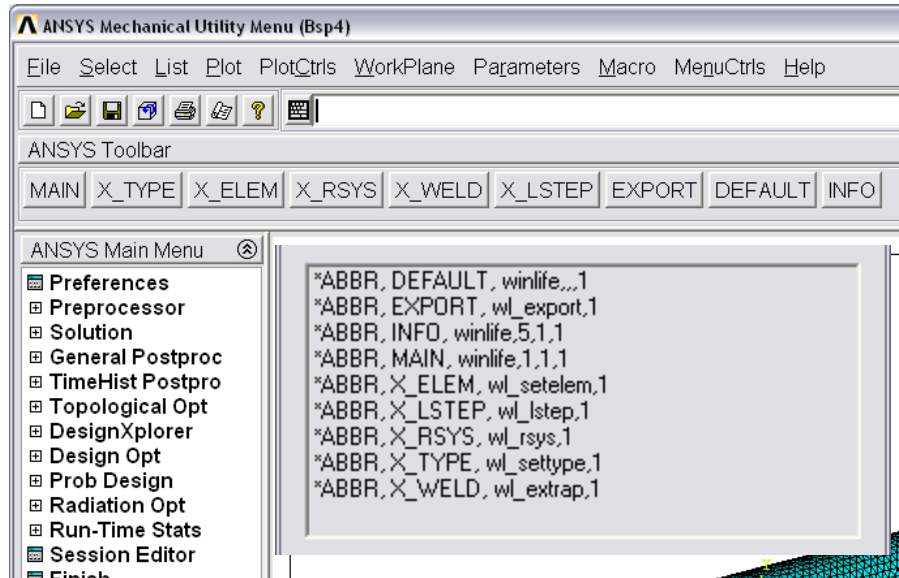
winLIFE MAIN Toolbar - winLIFE_main.abbr

Main menu for using the interface:



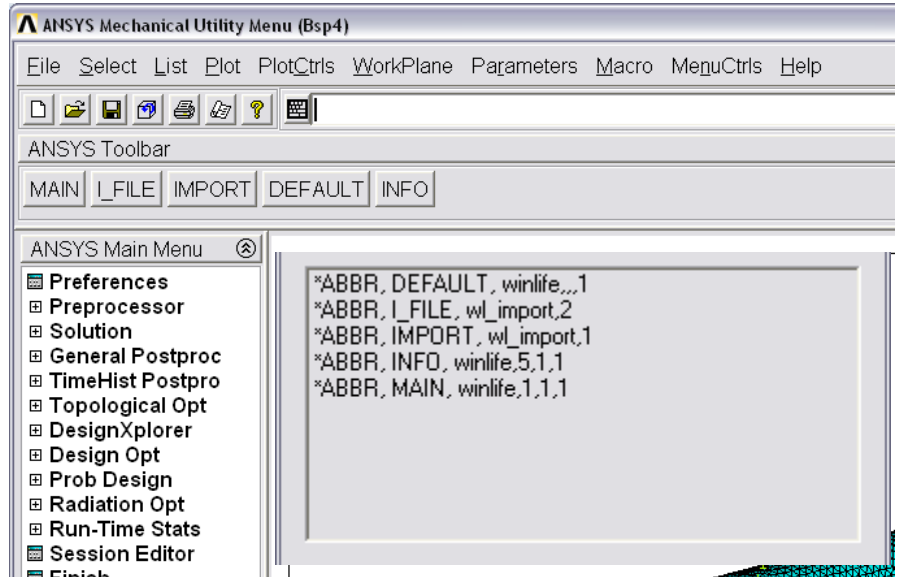
winLIFE EXPORT Toolbar - winLIFE_exp.abbr

Definition of the parameters, file names and settings for the export of stresses from the FE-Calculation.



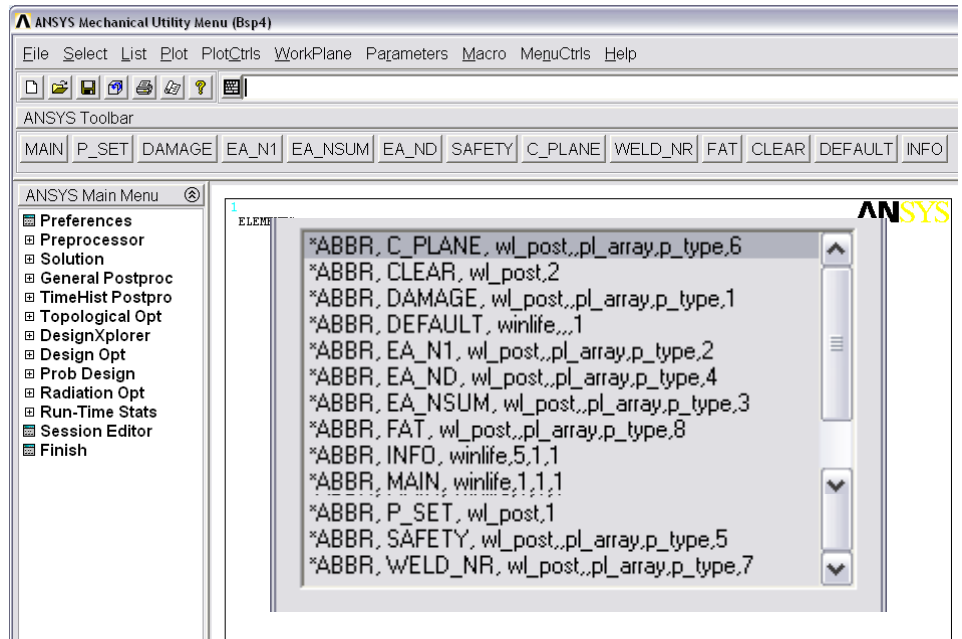
winLIFE IMPORT Toolbar - winLIFE_imp.abbr

Definition of the file names and parameters for the import of winLIFE result file.



winLIFE POST Toolbar - winLIFE_post.abbr

With the commands of the POST-Toolbar the results from winLIFE such as damage or safety factor can be shown in ANSYS as contour plots.



winLIFE to ANSYS Interface Macros

winlife.mac

WINLIFE,TLB_KEY,PAR_KEY,STD_KEY

Activates winLIFE to ANSYS Toolbar

TLB_KEY ... activates winLIFE menu bars

- 1 = Standard Toolbar
- 2 = Export Toolbar
- 3 = Import Toolbar
- 4 = Post Toolbar
- 5 = INFO Text

PAR_KEY ... 0 = Default Parameter values are set ((wl_param.mac)

≠0 no action

STD_KEY ... 0 = Default Parameter values are set ((wl_param.mac)

≠0 no action

Note:

For the use of the interface macros using the command line or in a program script, it is not necessary to carry out this command. If, however, the default values of parameters are to be used, it is recommended that these are set at the beginning of the export using the command "wl_param".

wl_nskin.mac

WL_NSKIN,XMODE,CMN_NAME

WL_NSKIN creates a node component 'cmn_name' with the "external" nodes of the selected elements. ("External" nodes are nodes on free element surfaces.)

XMODE ... Selection of the type of interactive mode
 0 = Execution by command line or script
 1 = activates the display of input menus

CMN_NAME ... Name of the node component [Character Parameter]

Note:

When defining the component name with the command, this must be placed between two single inverted commas. (e.g. wl_nskin,,'notch')

For the interactive definition via the dialogue field, however, you must not use these!

wl_eskin.mac

WL_ESKIN,XMODE,CMN_NAME,CME_NAME

WL_ESKIN creates shell elements for evaluating the stresses on the surface of solid elements. This technique is also called "skinning". The nodes of the components 'cmn_name' describes the "skin" area. The elements created are grouped in the components 'cme_name'.

XMODE ... Selection of the type of interactive mode
 0 = Execution by an command line or script
 1 = activates the display of input menus

CMN_NAME ... Name of the node component [Character Parameter]

CME_NAME ... Name of the element component [Character Parameter]

Note:

When defining the component name with the command, this must be placed between two single inverted commas. (e.g. wl_eskin,,'notch','skin')

For the interactive definition via the dialogue field, however, you must not use these!

wl_settype.mac

WL_SETTYPE,XMODE,WET

WL_SETTYPE defines the export type

XMODE ... Selection of the type of interactive modulus

0 = Execution by a command line or script
 1 = activates the display of input menus

WET ... Export type

1 = UNIAXIAL
 2 = MULTIAXIAL
 3 = Weld seam extrapolation
 4 = Weld seam Multiplier
 5 = non-linear / transient export

Note:
 The export type WET=3 is currently only supported for solid elements

wl_setelem.mac

WL_SETelem,XMODE,ESOLID,ESHELL,EBEAM,ELINK,SKINKEY

WL_SETelem defines the element type for export

XMODE ... Selection of the type of interactive modus

0 = Execution by an command line or script
 1 = activates the display of input menus

ESOLID ... 0/1 = No/Yes - solid elements

ESHELL ... 0/1 = No/Yes - shell elements

EBEAM ... 0/1 = No/Yes - beam elements (at present not yet supported)

ELINK ... 0/1 = No/Yes - bar elements (at present not yet supported)

SKINKEY ... 0/1 = No / Yes – use skinning technique for solids

GRDKEY ... 0/1=No/Yes – node information for stress gradient

Note:
 To use the skinning technique it is necessary to pre-define the required nodes and elements using the macros *wl_nskin.mac* and *wl_eskin.mac*

wl_rsys.mac

WL_RSYS,XMODE,CS_EXP

WL_RSYS defines the output coordinate system for the export

XMODE ... Selection of the type of interactive modus

0 = Execution by an command line or script
 1 = activates the display of input menus

CS_EXP ... Output coordinate system

-1 = RSYS SOLU (element coordinate system)

≠-1 = Number of a defined coordinate system

Note:

When exporting unstructured meshed shell elements it is particularly important to check the result coordinate system, because the averaged node results do not automatically take into account the different orientations of the coordinate systems. After automatic meshing but before starting the solver it is strongly recommended to orientate the element coordinate systems correctly!

wl_extrap.mac

WL_EXTRAP,XMODE,XTYPE,XF_0,HSTYP,HSTHK,WLFAT,HSTOE,HSPATH,HSPLO

WL_EXTRAP defines the parameter for extrapolation for the export of weldings

Selection of the type of interactive modus

0 = Execution by an command line or script

1 = activates the display of input menus

XTYPE ... Method of extrapolation

0 = automatic classification according to IIW

1 = linear extrapolation (2 reference points)

2 = quadratic extrapolation (3 reference points)

3 = rough extrapolation for coarse meshes

XF_0 ... Multiplier for multiplier method

HSTYP ... 1 = „a“ / 2 = „b“ HotSpot

HSTHK ... thickness of sheet at hot spot

WLFAT... FAT class (only for documentation)

HSTOE ... line of the foot point

HSPATH ... path of extrapolation perpendicular to the weld

HSPLO ... 0/1 = No / Yes – create a path-plot of the extrapolation

Note:

The multipliers according to the IIW recommendations are implemented within the macro. Changes of these multipliers are only possible in the programming code.

wl_lstep.mac

WL_LSTEP,XMODE,LSSTART,LSEND

WL_LSTEP defines the area of the load cases to be exported.

XMODE ... Selection of the type of interactive modus

0 = execution by a command line or script

1 = activates the display of input menus

LSSTART ... first load case
LSEND ... last load case

Note:

With the two parameters LSSTART and LSEND all available load cases between these will be exported. If this is not required, the load cases must be exported individually one after the other.

wl_export.mac

WL_EXPORT,XMODE,F_APPEND,E_FILNAM,CME_NAME,WLSTART,WLEND

WL_EXPORT writes the stress tensors of the selected nodes taking into account the settings previously selected into the winLIFE input files *.lst

XMODE ... Selection of the type of interactive modus
0 = Execution by a command line or script
1 = activates the display of input menus

F_APPEND ... Activates writing of data into existing files
0 = any existing data is overwritten
1 = any existing data is amended

E_FILNAM ... File name of the export file [Character Parameter]

CME_NAME ... Name of the skin – element component [Character Parameter]

WLSTART ... Number of the first weld or hot-spot for export

WLEND ... Number of the last weld or hot spot for the export

Note:

When defining the file name with the command, this must be placed between two single inverted commas. (e.g. wl_import,, 'example').

For the interactive definition via the dialogue window, however, you must not use these!

WARNING! The length of the file name should be limited to 8 characters!

All existing welds in between WLSTART und WLEND are exported. If this is not wished, the welds must be exported one after the other with separate commands.

The macros eu_solid.mac, eu_shell.mac, em_solid.mac, em_shell.mac, ewm_solid.mac, ewm_shell.mac, ewx_solid.mac, enl_solid.mac, enl_shell.mac, ewmu_shell.mac und eme_shell.mac are sub-routines for the export and can only be used in this context.

wl_import.mac

WL_IMPORT,XMODE,I_FILNAM,I_ARRAY

WL_IMPORT reads the winLIFE result values out of the file *.exp into a user defined ARRAY-parameter.

XMODE ...	Selection of the type of interactive modus 0 = Execution by a command line or script 1 = interactive import start 2 = activates the display of the input menu
I_FILNAM ...	File name of the import file [Character Parameter]
I_ARRAY ...	Name of the result - ARRAYS [Character Parameter]

Note:

When defining the file name with the command, this must be placed between two single inverted commas. (e.g. wl_import,,,'example')

For the interactive definition via the dialogue window, however, you must not use these!

WARNING! The length of the file name should be limited to 8 characters!

wl_post.mac

WL_POST,XMODE,PL_ARRAY,P_TYPE,R_TYPE

WL_POST reads the winLIFE result values out of a defined ARRAY-Parameter and creates a plot or a list of the user requested results.

XMODE ...	Selection of the type of interactive modus 0 = Execution by an command line or script 1 = activates the display of input menus 2 = Reset the display to ANSYS results
PL_ARRAY ...	Name of the result - ARRAYS [Character Parameter]
P_TYPE ...	Type of result presentation 0 = Data only read in 1 = Contour plots are created 2 = Results are listed in output window
R_TYPE ...	Result value 1 = Damage [D] 2 = Equivalent Amplitude for one cycle [EA_1] 3 = Equiv. Amplitude for the sum cycle number [EA_nsum] 4 = Equiv. Amplitude for the endurance limit [EA_ND]

5 = Utilisation rate

6 = critical plane

7 = Weld number

8 = FAT class

Note:

The presentation of the contour plot occurs in the graphic mode /Graphics,Full The mode /Graphics,Power cannot used for the presentation of the results!

wl_param.mac

This file contains the default values for the parameters "Standard" file-names of winLIFE to ANSYS interface macros. These values can be changed and adapted by the user. The parameters and the standard values delivered are shown below:

Parameter	Default
e_filnam	active jobname
i_filnam	active jobname
wet	1
esolid	1
eshell	1
ebeam	0
elink	0
Skinkey	0
cs_exp	0
hstype	1
xsthk	1
xtype	1
xf_0	1.00
wlfat	100
hsplo	0
hstoe	1
hspath	1
wlstart	1
wlend	1
lsstart	1
lsend	1
f_append	0
cmn_name	n_skin
cme_name	e_skin
i_array	wl_res
pl_array	%i_array%
p_type	1
r_type	1

Note:

File Summary (V4.0)

Term	File	Version
Menu bars	winLIFE_main.abbr	1.0
	winLIFE_exp.abbr	1.0
	winLIFE_imp.abbr	1.0
	winLIFE_post.abbr	2.0
0) General	winLIFE.mac	4.0
1) Pre-processing	wl_nskin.mac	2.0
	wl_eskin.mac	2.0
2) Export of results	wl_settype.mac	4.0
	wl_setelem.mac	4.0
	wl_rsys.mac	2.0
	wl_extrap.mac	2.2
	wl_lstep.mac	2.0
	wl_export.mac	4.0
	eu_solid.mac	3.1
	eu_shell.mac	3.1
	em_solid.mac	3.1
	em_shell.mac	4.0
	ewm_solid.mac	3.1
	ewm_shell.mac	4.0
	ewx_solid.mac	3.1
	enl_solid.mac	3.1
	enl_shell.mac	3.1
	ewmu_shell.mac	3.2
	eme_shell.mac	4.0
3) Import of results	wl_import.mac	2.1
4) Presentation of results in the Post-processor	wl_post.mac	2.0
5) Parameter	wl_param.mac	2.3