MultiFat

Fatigue Analysis Tool

Quick Tutorial

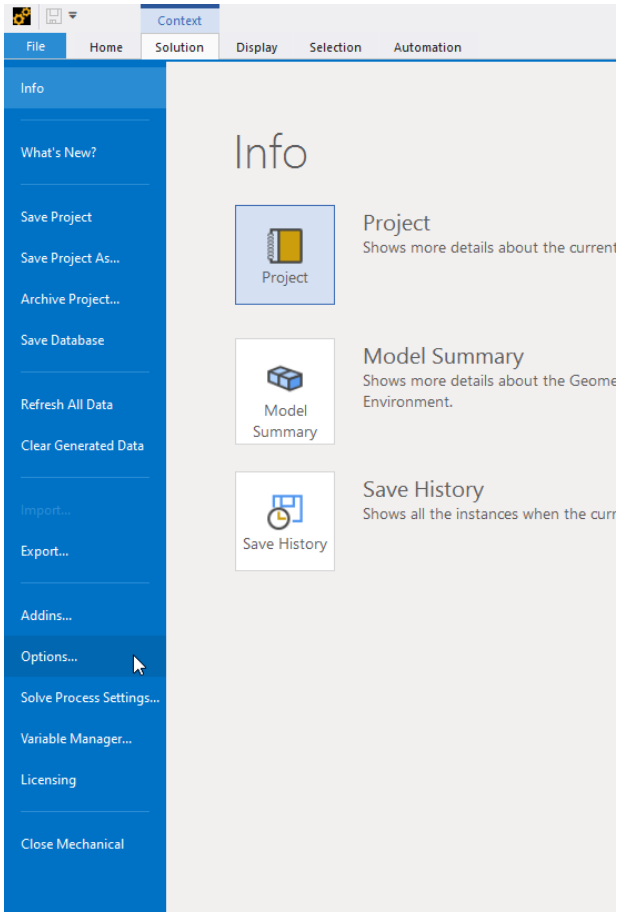
Ansys

1. Export stress results from Ansys
2. Convert the stresses in the MF format
3. Input files for MultiFat
4. Define the blocks

MultiFat Tutorial for Ansys

1. Export stress results from Ansys

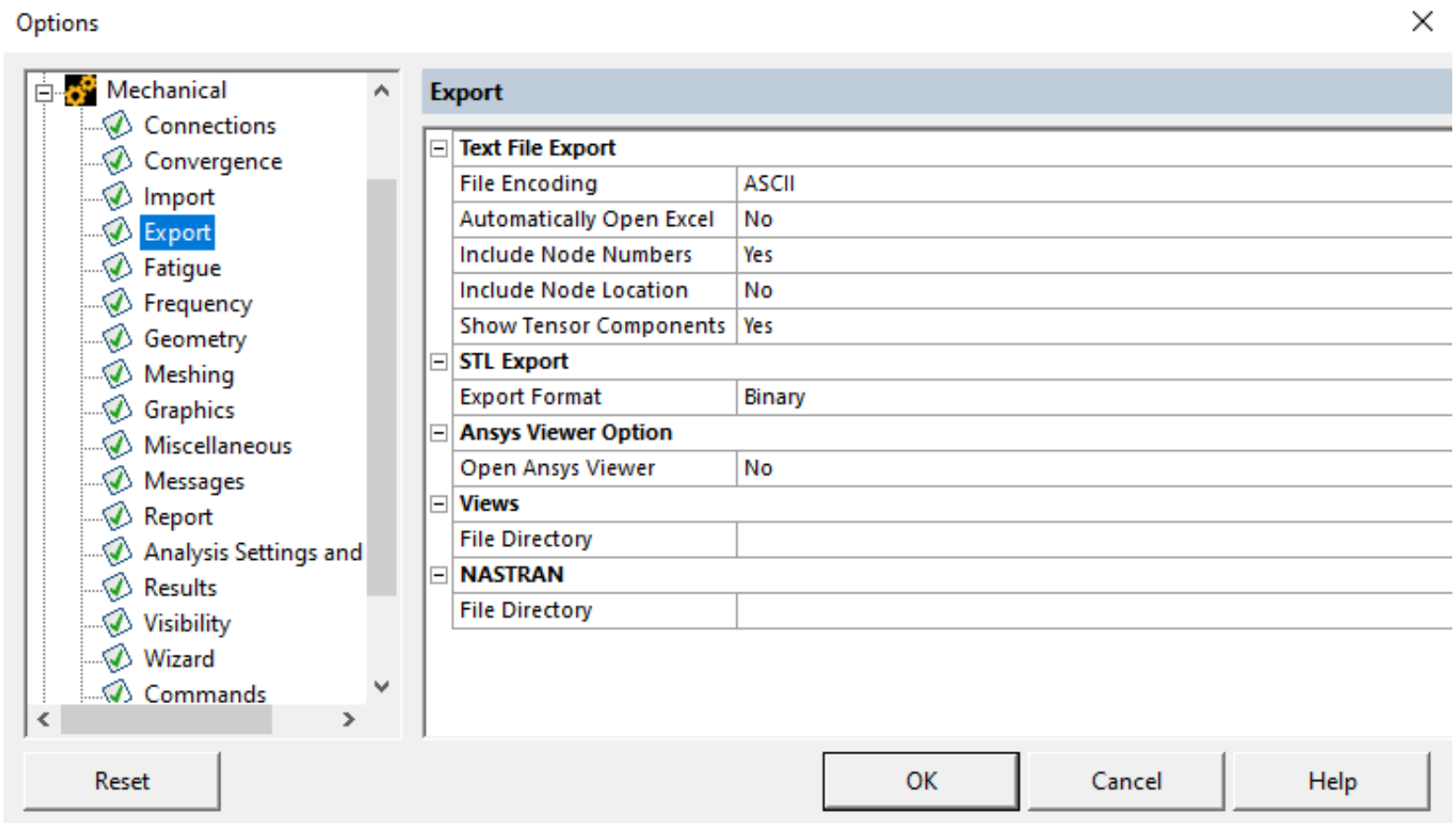
Select "Options…" in Ansys Mechanical



MultiFat Tutorial for Ansys

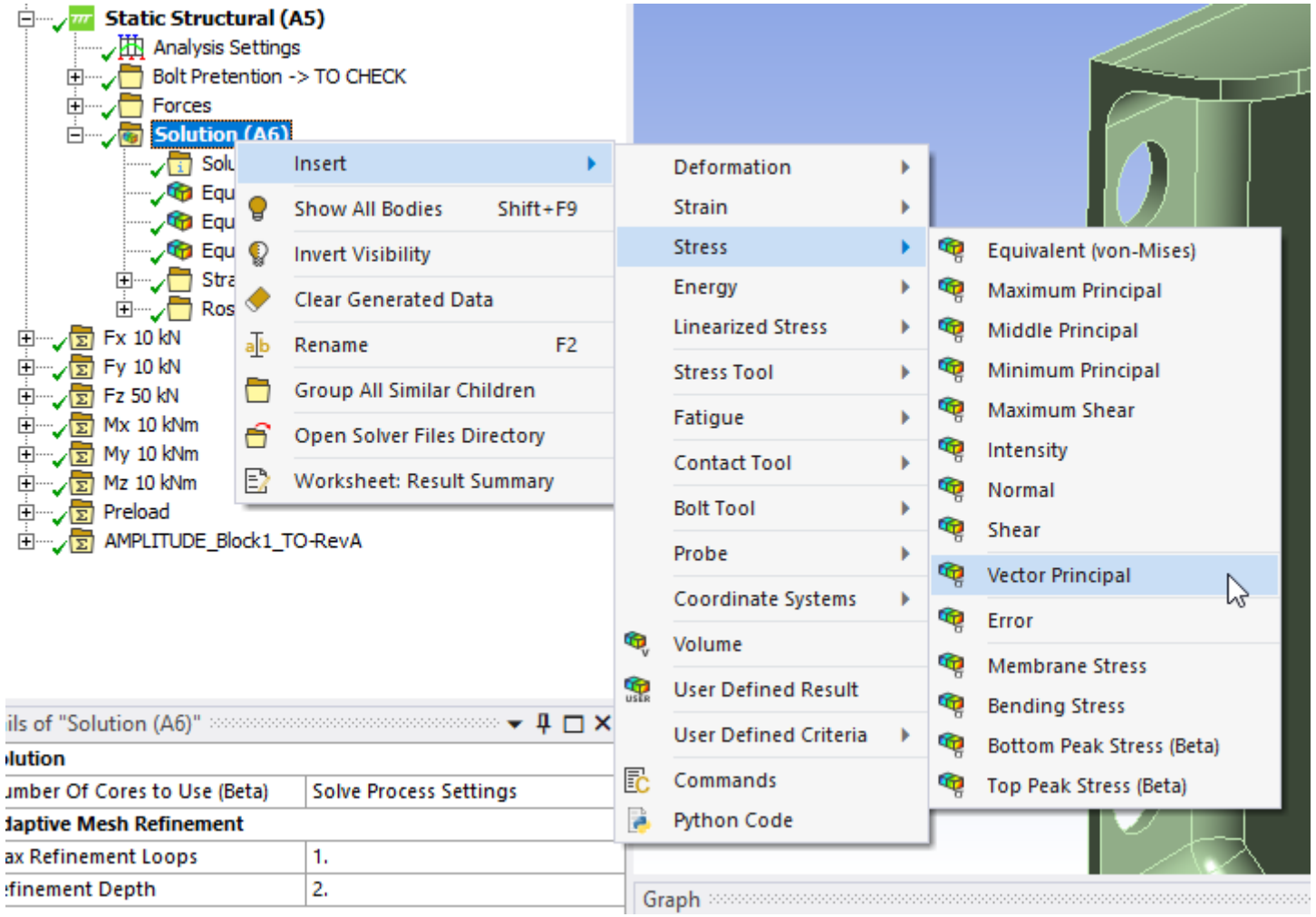
1. Export stress results from Ansys

Choose the following:



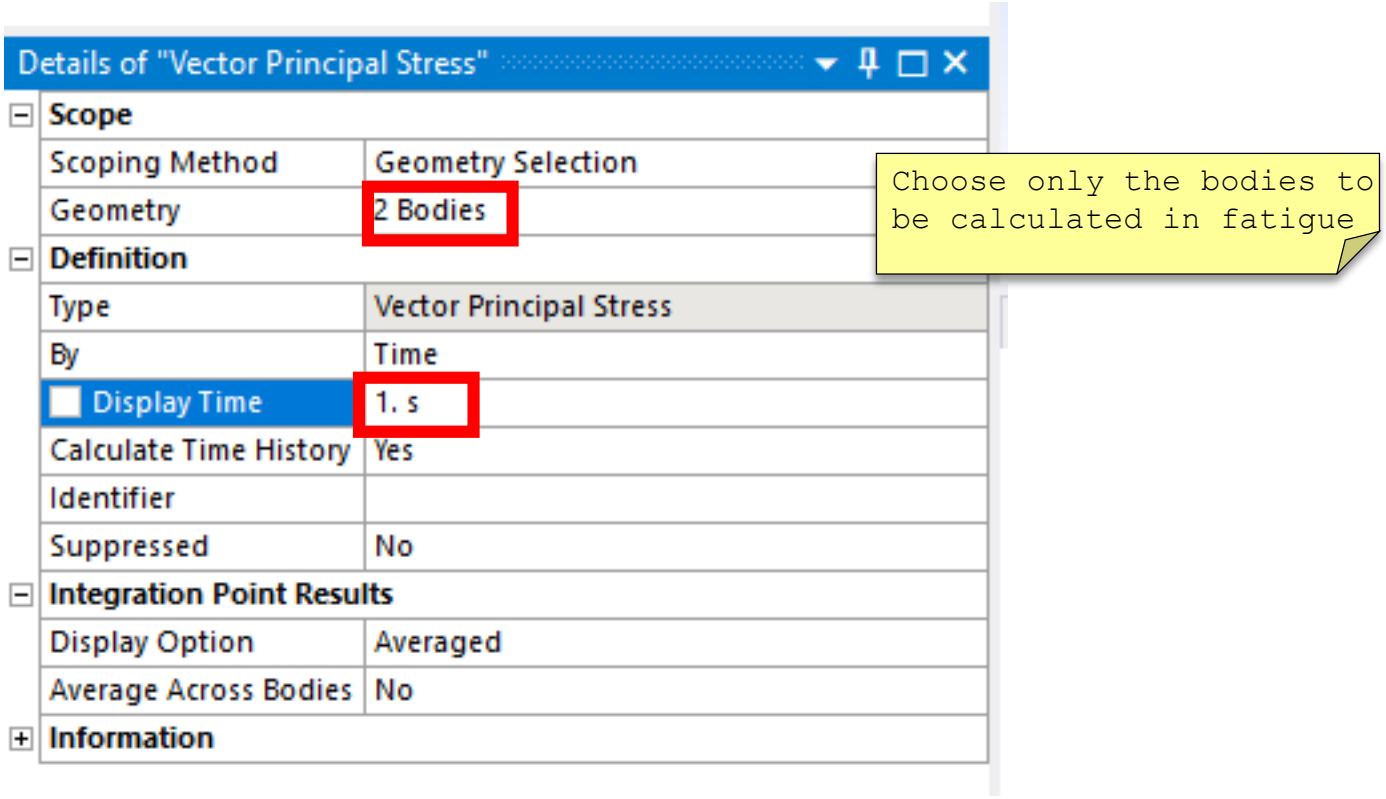
MultiFat Tutorial for Ansys

1. Export stress results from Ansys



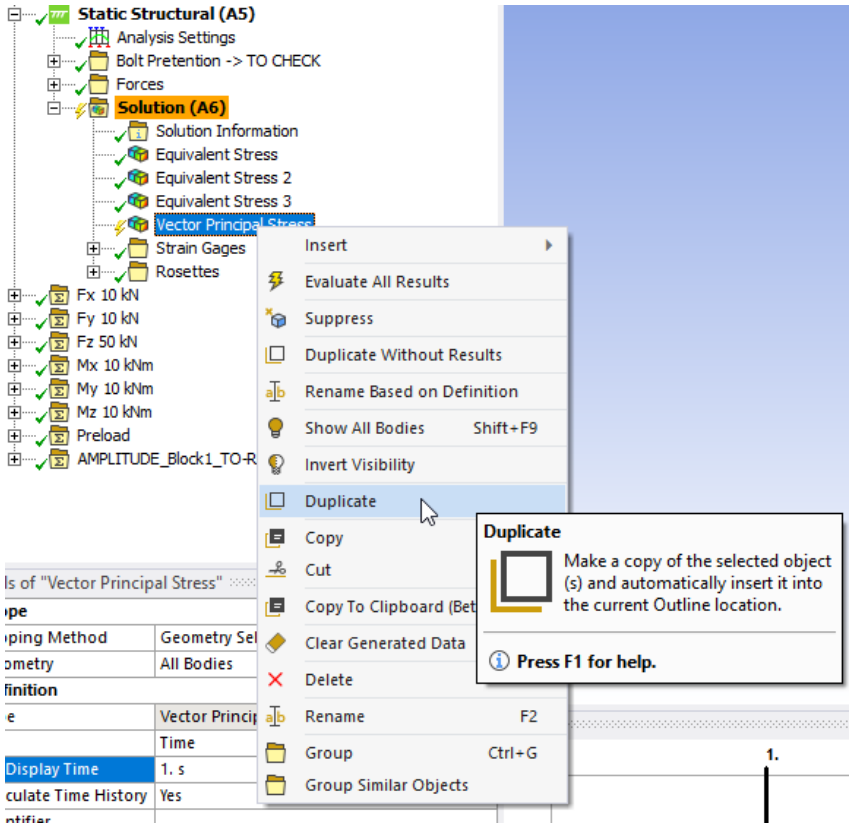
MultiFat Tutorial for Ansys

1. Export stress results from Ansys



MultiFat Tutorial for Ansys

1. Export stress results from Ansys



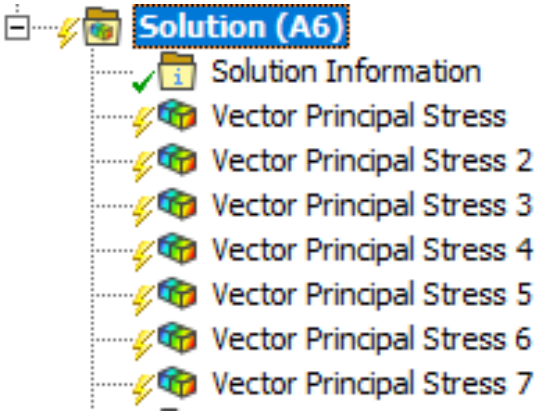
Create Vector Principal Stress for all the LC's

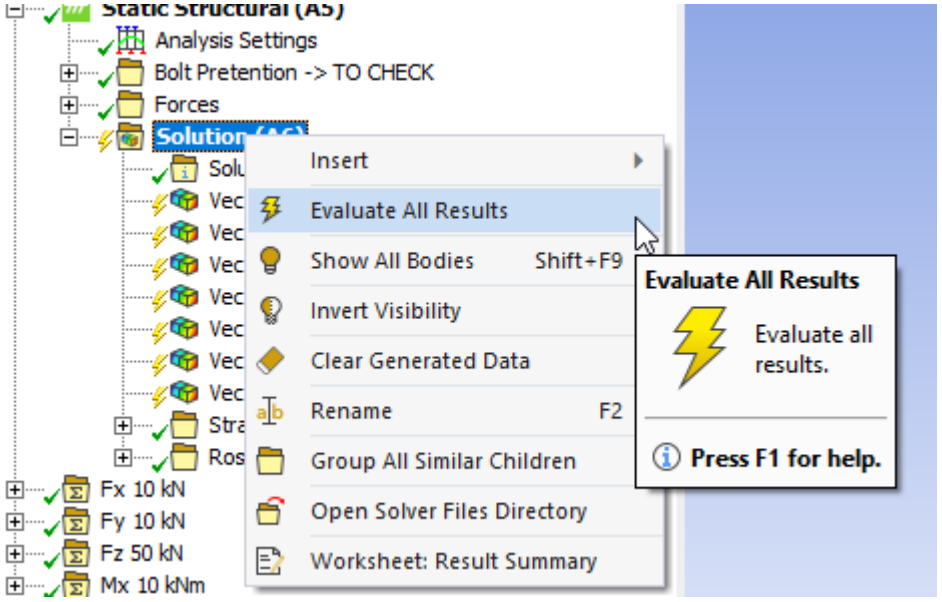
Details of "Vector Principal Stress 2"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Vector Principal Stress
By	Time
Display Time	2. s
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Information	

MultiFat Tutorial for Ansys

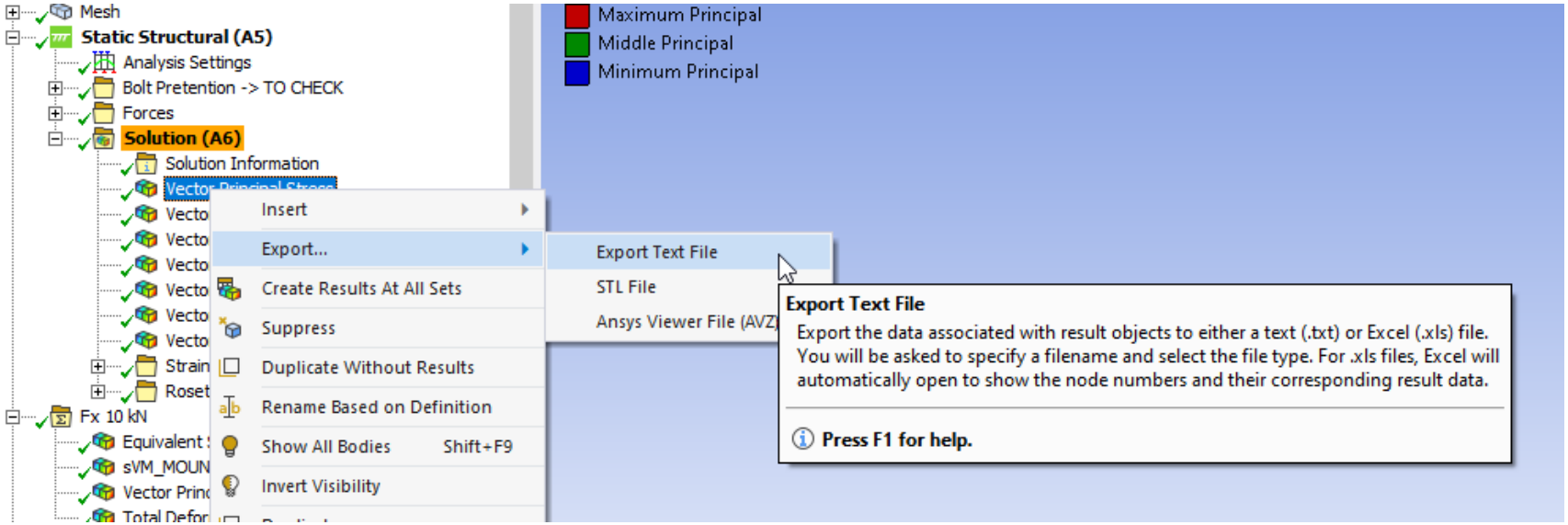
1. Export stress results from Ansys





MultiFat Tutorial for Ansys

1. Export stress results from Ansys

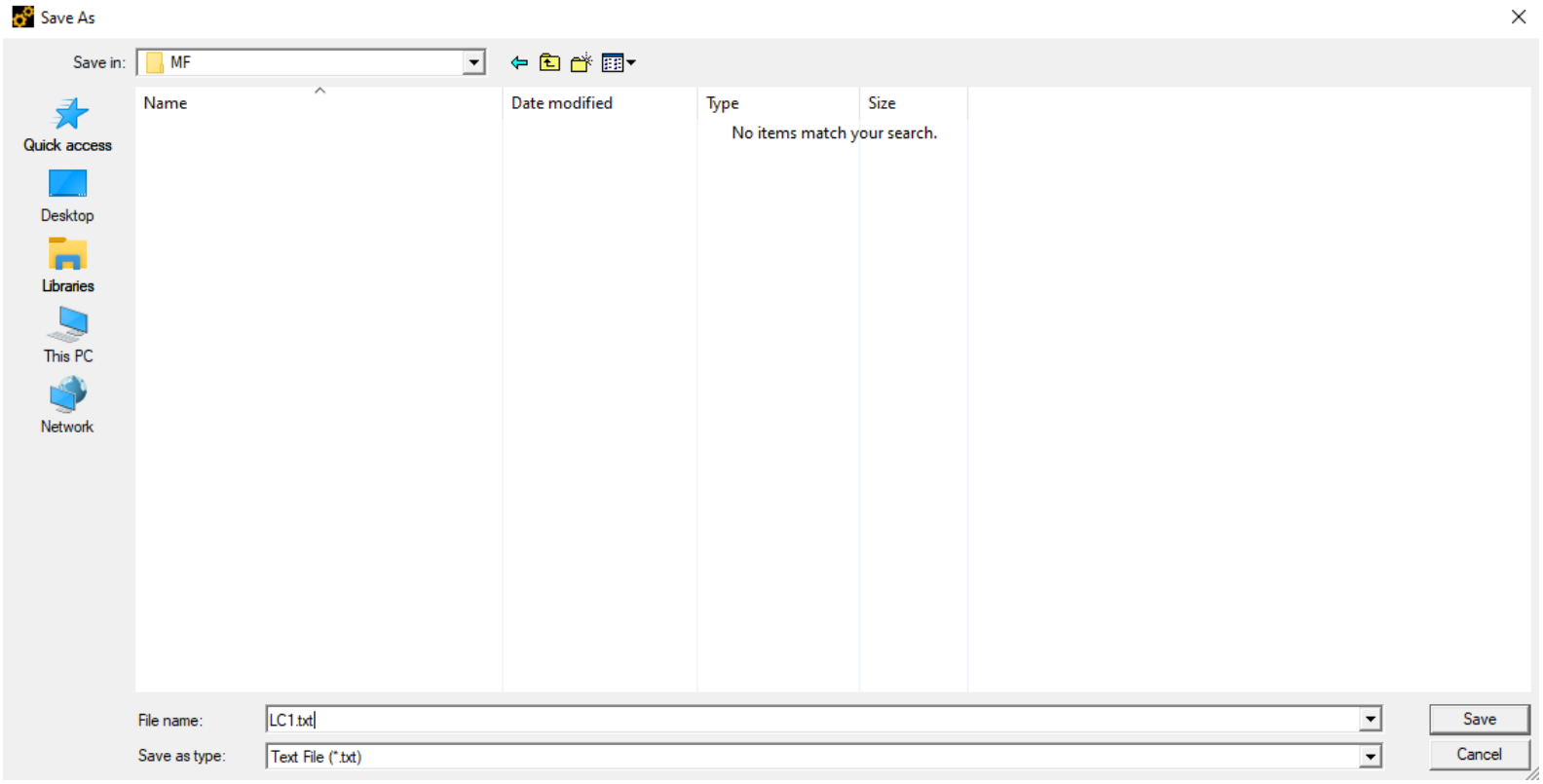


N.B. The bodies under analysis must be visible in Ansys.

Ansys will export the stresses only for the visible bodies.

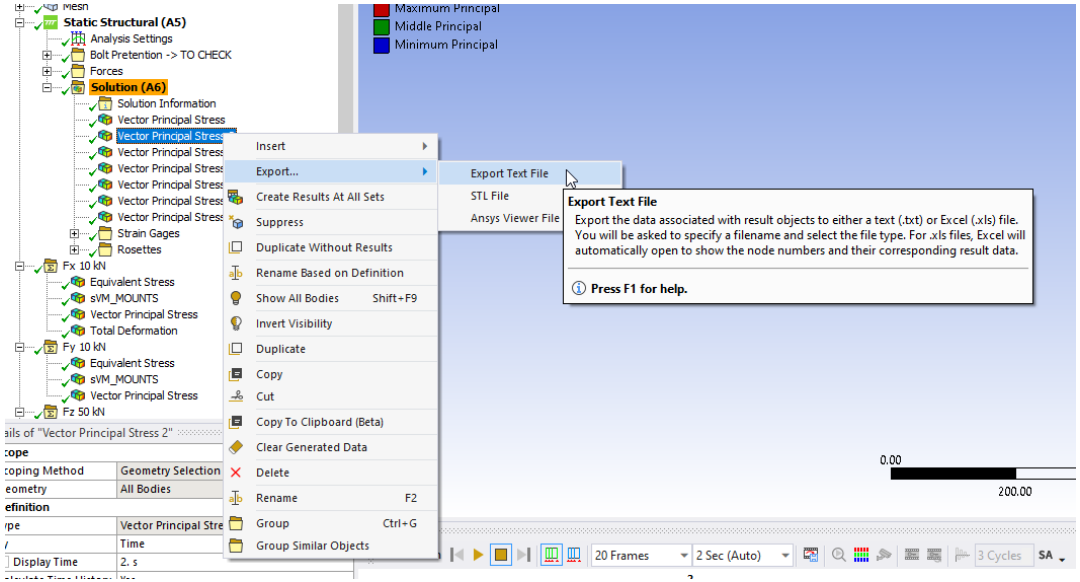
MultiFat Tutorial for Ansys

1. Export stress results from Ansys

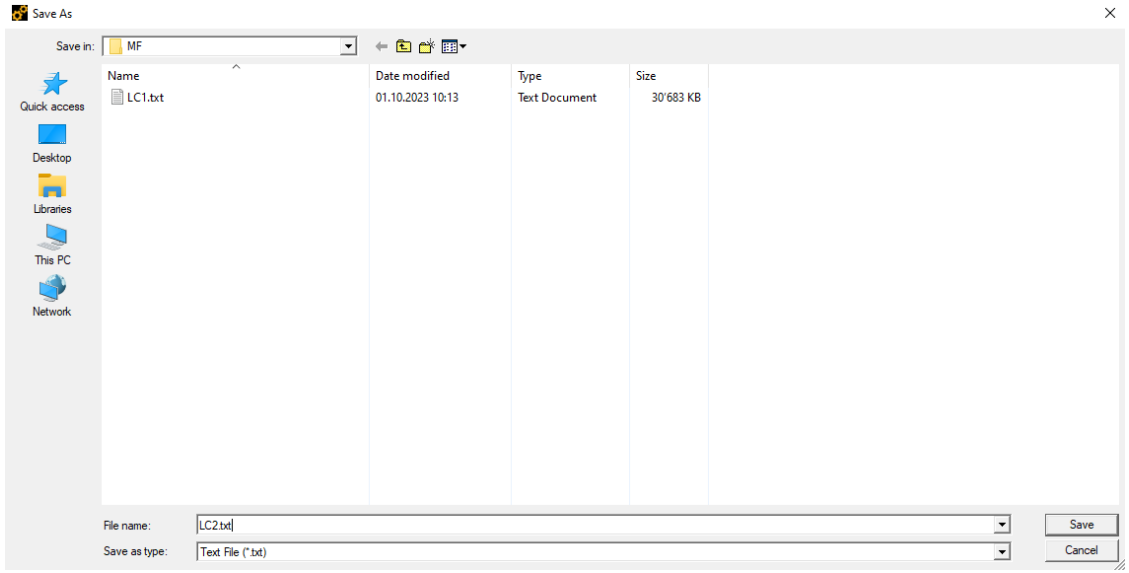


MultiFat Tutorial for Ansys

1. Export stress results from Ansys



Repeat the export for all the LC's



MultiFat Tutorial for Ansys

2. Convert the stresses in the MF format

readstress.m

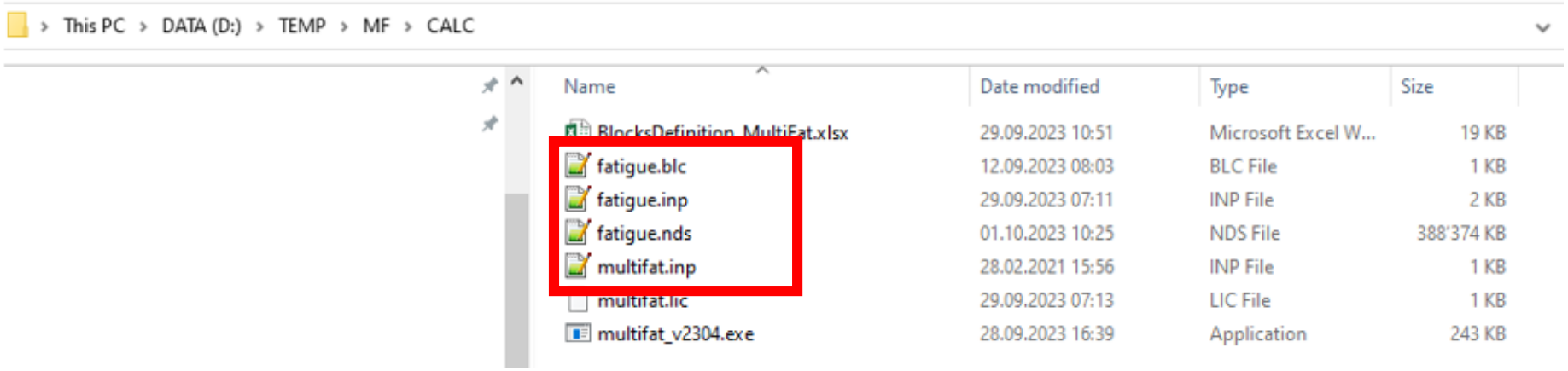
```
% Matlab routine to transform stresses exported from Ansys into MultiFat format "*.nds"

sa1 = dlmread('LC1.txt','\t', 1, 0);
sa2 = dlmread('LC2.txt','\t', 1, 0);
sa3 = dlmread('LC3.txt','\t', 1, 0);
sa4 = dlmread('LC4.txt','\t', 1, 0);
sa5 = dlmread('LC5.txt','\t', 1, 0);
sa6 = dlmread('LC6.txt','\t', 1, 0);
sa7 = dlmread('LC7.txt','\t', 1, 0);

nnd2 = size(sa1);
nnd = nnd2(1)

idmat = 1001*ones(nnd,1);

% if the stresses have been exported directly from LC's
sa1 = [sa1(:,1),idmat,sa1(:,2:7)];
sa2 = [sa2(:,1),idmat,sa2(:,2:7)-sa1(:,2:7)];
sa3 = [sa3(:,1),idmat,sa3(:,2:7)-sa1(:,2:7)];
sa4 = [sa4(:,1),idmat,sa4(:,2:7)-sa1(:,2:7)];
sa5 = [sa5(:,1),idmat,sa5(:,2:7)-sa1(:,2:7)];
sa6 = [sa6(:,1),idmat,sa6(:,2:7)-sa1(:,2:7)];
sa7 = [sa7(:,1),idmat,sa7(:,2:7)-sa1(:,2:7)];

% if the stresses have been exported as Linear Combinations differences with the LC1 preload
%sa1 = [sa1(:,1),idmat,sa1(:,2:7)];
%sa2 = [sa2(:,1),idmat,sa2(:,2:7)];
%sa3 = [sa3(:,1),idmat,sa3(:,2:7)];
%sa4 = [sa4(:,1),idmat,sa4(:,2:7)];
%sa5 = [sa5(:,1),idmat,sa5(:,2:7)];
%sa6 = [sa6(:,1),idmat,sa6(:,2:7)];
%sa7 = [sa7(:,1),idmat,sa7(:,2:7)];

samf = [sa1; sa2; sa3; sa4; sa5; sa6; sa7];

save fatigue.nds samf -ascii
```

Run 'readstress.m' in Matlab

The file "fatigue.nds" will be created

Name	Date modified	Type	Size
LC1.txt	01.10.2023 10:13	Text Document	30'683 KB
LC2.txt	01.10.2023 10:15	Text Document	26'316 KB
LC3.txt	01.10.2023 10:15	Text Document	25'750 KB
LC4.txt	01.10.2023 10:15	Text Document	25'224 KB
LC5.txt	01.10.2023 10:15	Text Document	25'341 KB
LC6.txt	01.10.2023 10:16	Text Document	25'934 KB
LC7.txt	01.10.2023 10:16	Text Document	25'560 KB
readstress.m	01.10.2023 10:23	MATLAB Code	2 KB

3. Input files for MultiFat



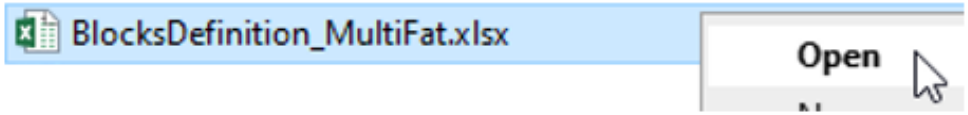
N.B. "fatigue.nds" is the file created with the Matlab routine

N.B. This applies to «Blocks of Cycles» analysis (see MultiFat User Manual) only

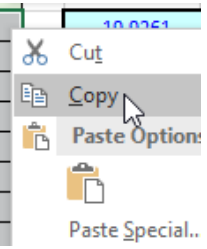
Blocks of cycles

4. Define the blocks

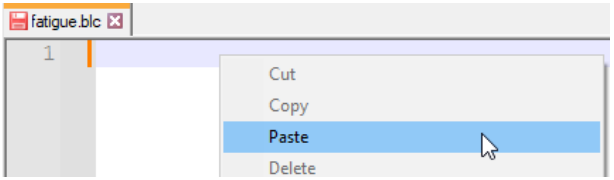
Use the Excel file

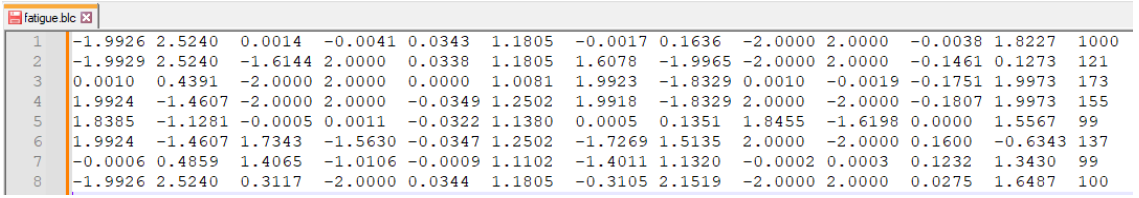


-1.9926	2.5240	0.0014	-0.0041	0.0343	1.1805	-0.0017	0.1636	-2.0000	2.0000	-0.0038	1.8227	1000
-1.9929	2.5240	-1.6144	2.0000	0.0338	1.1805	1.6078	-1.9965	-2.0000	2.0000	-0.1461	0.1273	121
0.0010	0.4391	-2.0000	2.0000	0.0000	1.0081	1.9923	-1.8329	0.0010	-0.0019	-0.1751	1.9973	173
1.9924	-1.4607	-2.0000	2.0000	-0.0349	1.2502	1.9918	-1.8329	2.0000	-2.0000	-0.1807	1.9973	155
1.8385	-1.1281	-0.0005	0.0011	-0.0322	1.1380	0.0005	0.1351	1.8455	-1.6198	0.0000	1.5567	99
1.9924	-1.4607	1.7343	-1.5630	-0.0347	1.2502	-1.7269	1.5135	2.0000	-2.0000	0.1600	-0.6343	137
-0.0006	0.4859	1.4065	-1.0106	-0.0009	1.1102	-1.4011	1.1320	-0.0002	0.0003	0.1232	1.3430	99
-1.9926	2.5240	0.3117	-2.0000	0.0344	1.1805	-0.3105	2.1519	-2.0000	2.0000	0.0275	1.6487	100



Copy & Paste into the file "fatigue.blc"





MultiFat Tutorial for Ansys

5. Define the input

Use **MultiFat_Input.xlsx**

Stress-strain relationship: fty, E, ftu, Atot%

420	71000	490	9

Elastoplastic correction is performed (1) or not (0)

0

Elastoplastic correction with Linearized (1) or classical Neuber (2)

1

Stress-path based correction performed (1) or not (0)

1

S-N curve in the format of Palmgren (1), Weibull (2) or Multilinear (3)

1

Palmgren parameters

L1,p,C,EndLim

9.17754	-2.07258182	10026	156.3

Weibull parameters

sainf,sasup,alfaw,betaw

13.97	319	3	4.3

Multilinear parameters

sE, NE, kE, N1, k1, N2, k2, k3

32.8	10000000	3.7	560	1000000	1000000000	6	0

Goodman correction exponent, type of behavior for Sm<0, SmMAX

1	2	490

Stress Sequence (1), Linear Combination (2) or Blocks (3)

3

Coeffs A, B when Linear Combination or Blocks are used

1	1

Write complete output to the following node ID

1644

Analysis made with one material (1) or multiple materials (2)

1

Number of Nodes and Number of Load Cases (Preload excluded)

913877	6

LVF for Sa in Blocks Fatigue

1

For linear combination only: GAG analysis (1) or All Cycles analysis (0)

0

For Stress Sequence only: extract cycles (1) or not (0)

1

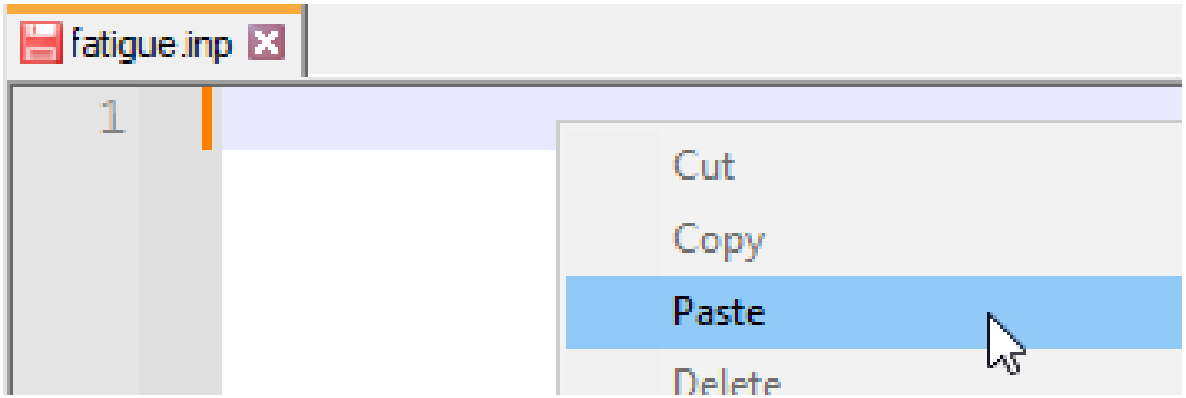
For Blocks analysis only: sum of damages (1), highest (0), highest flight (2)

1

Number of Flight Hours

1884

5. Define the input



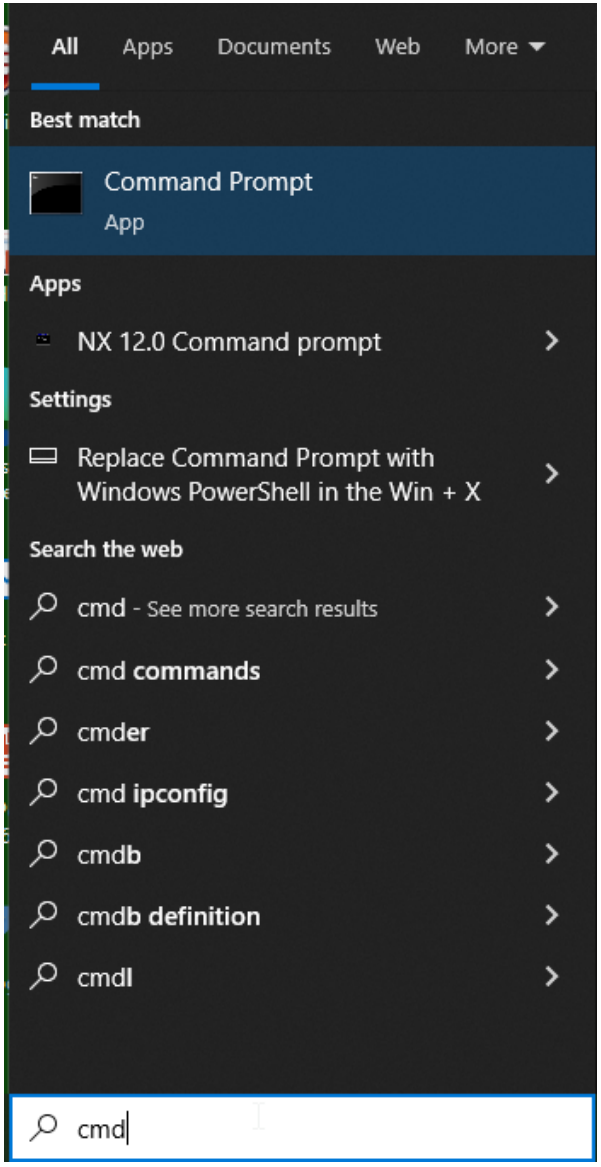
5. Define the input

fatigue.inp

```
Stress-strain relationship: fty, E, ftu, Atot%
420 71000   490 9
Elastoplastic correction is performed (1) or not (0)
0
Elastoplastic correction with Linearized (1) or classical Neuber (2)
1
Stress-path based correction performed (1) or not (0)
1
S-N curve in the format of Palmgren (1), Weibull (2) or Multilinear (3)
1
Palmgren parameters
L1,p,C,EndLim
9.177539713 -2.072581815    10026   156.3
Weibull parameters
sainf,sasup,alfaw,betaw
13.97   319 3   4.3
Multilinear parameters
sE, NE, kE, N1, k1, N2, k2, k3
32.8    10000000    3.7 560 1000000 1000000000  6   0
Goodman correction exponent, type of behavior for Sm<0, SmMAX
1   2   490
Stress Sequence (1), Linear Combination (2) or Blocks (3)
3
Coeffs A, B when Linear Combination or Blocks are used
1   1
Write complete output to the following node ID
1644
Analysis made with one material (1) or multiple materials (2)
1
Number of Nodes and Number of Load Cases (Preload excluded)
913877  6
LVF for Sa in Blocks Fatigue
1
For linear combination only: GAG analysis (1) or All Cycles analysis (0)
0
For Stress Sequence only: extract cycles (1) or not (0)
1
For Blocks analysis only: sum of damages (1), highest (0), highest flight (2)
1
Number of Flight Hours
1884.0
```

MultiFat Tutorial for Ansys

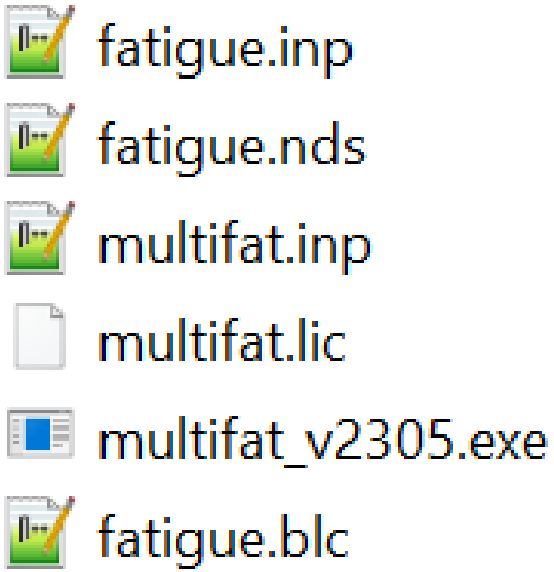
6. Run MultiFat



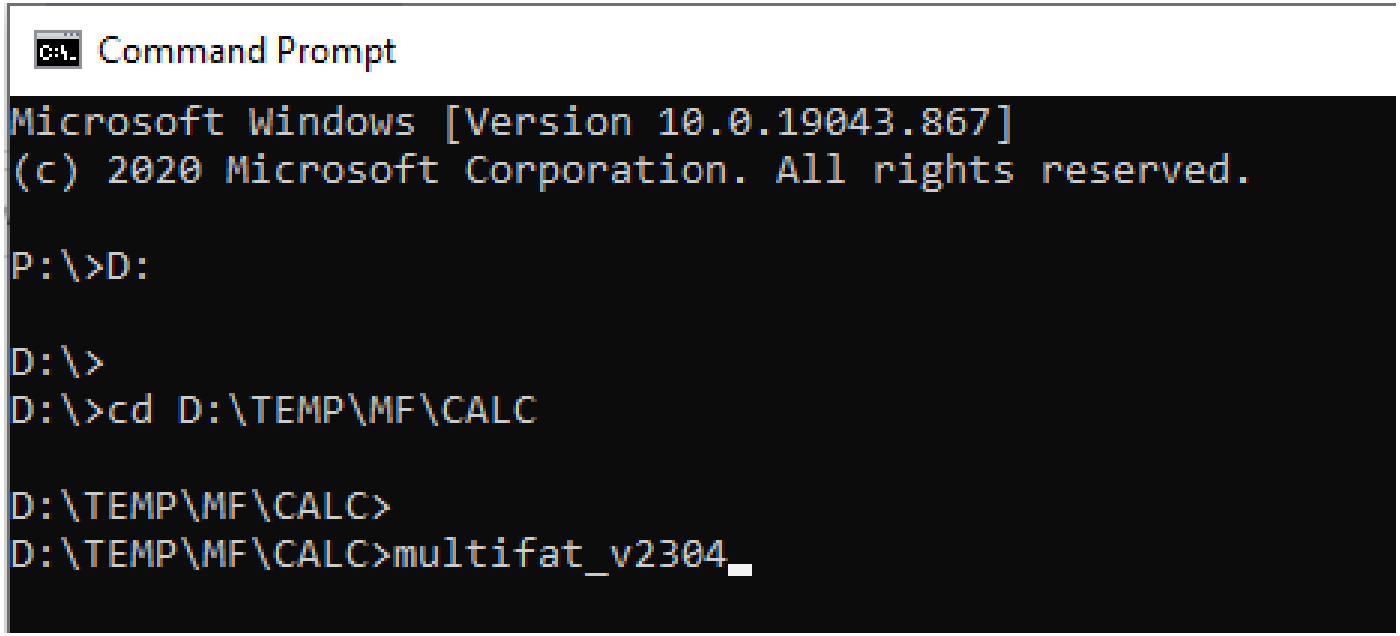
6. Run MultiFat

N.B. This applies to «Blocks of Cycles» analysis (see MultiFat User Manual) **Blocks of cycles**

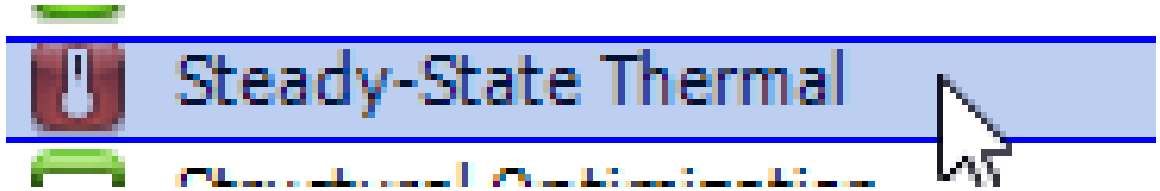
Files needed to run MultiFat are:



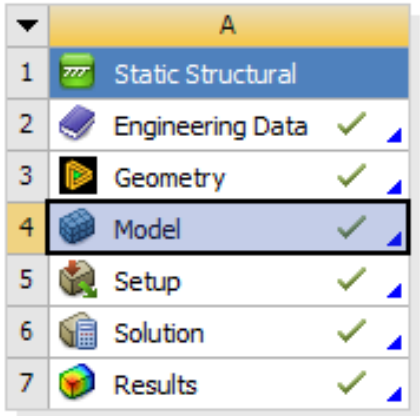
6. Run MultiFat



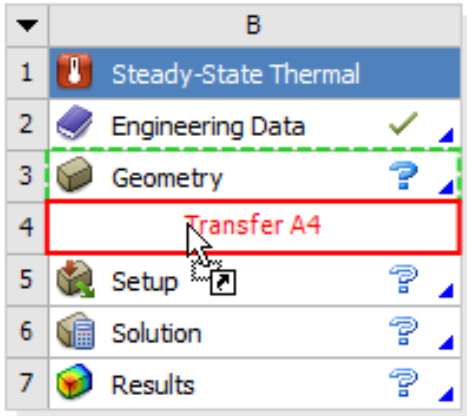
7. Import results in Ansys



7. Import results in Ansys

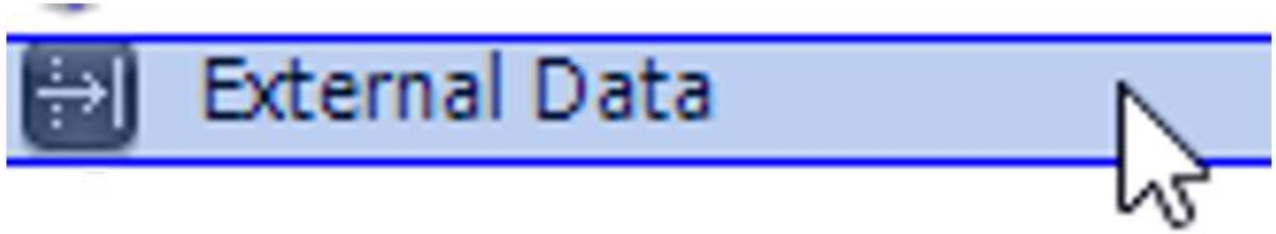


MGB_Mounts_Bench_Test_PS5

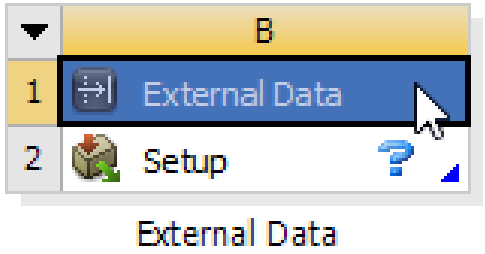


Steady-State Thermal

7. Import results in Ansys

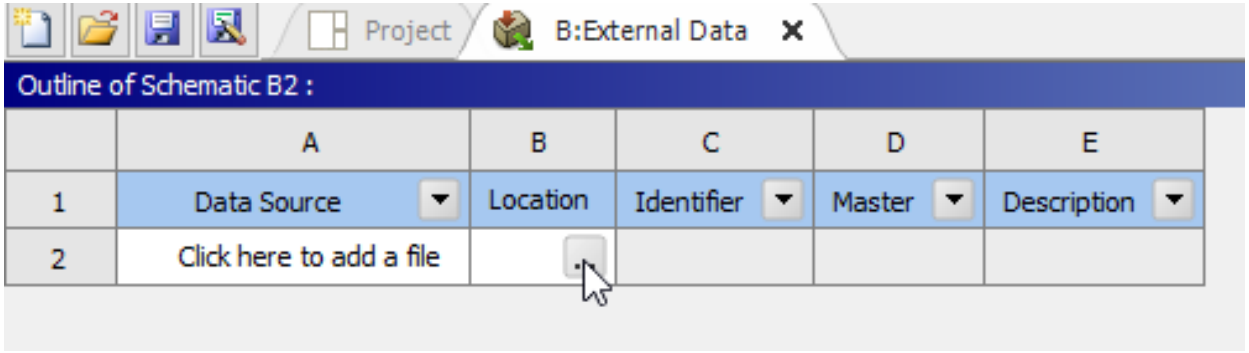


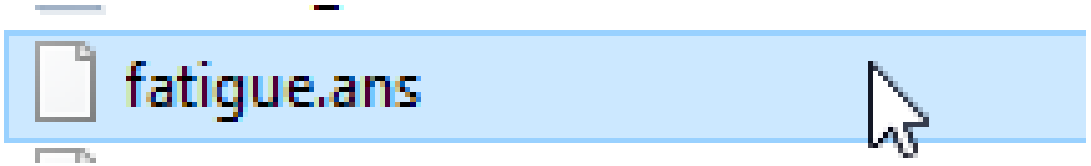
7. Import results in Ansys



Double-Click

7. Import results in Ansys





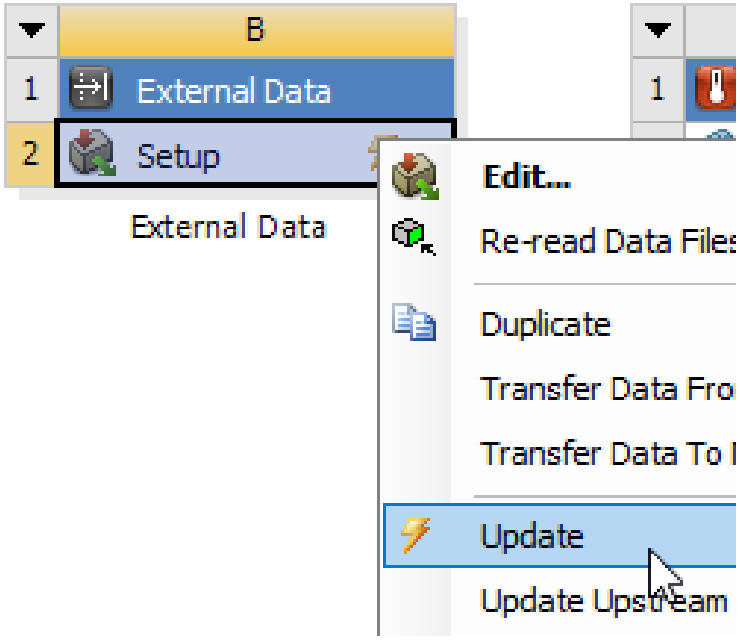
7. Import results in Ansys

Properties of File - D:\TEMP\MF\CALC\fatigue.ans

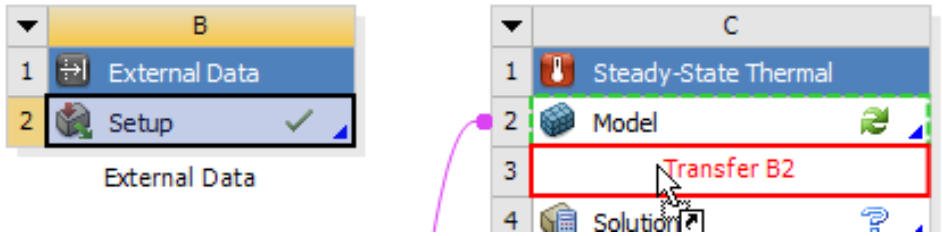
	A	B	C
1	Property	Value	Unit
2	Definition		
3	Dimension	3D	
4	Start Import At Line	1	
5	Format Type	Delimited	
6	Delimiter Type	Space	
7	Delimiter Character	Space	
8	Length Unit	mm	
9	Coordinate System Type	Cartesian	
10	Material Field Data		
11	Analytical Transformation		
12	X Coordinate	x	
13	Y Coordinate	y	
14	Z Coordinate	z	
15	Rigid Transformation		
16	Origin X	0	mm
17	Origin Y	0	mm
18	Origin Z	0	mm
19	Theta XY	0	radian
20	Theta YZ	0	radian
21	Theta ZX	0	radian

7. Import results in Ansys

7. Import results in Ansys

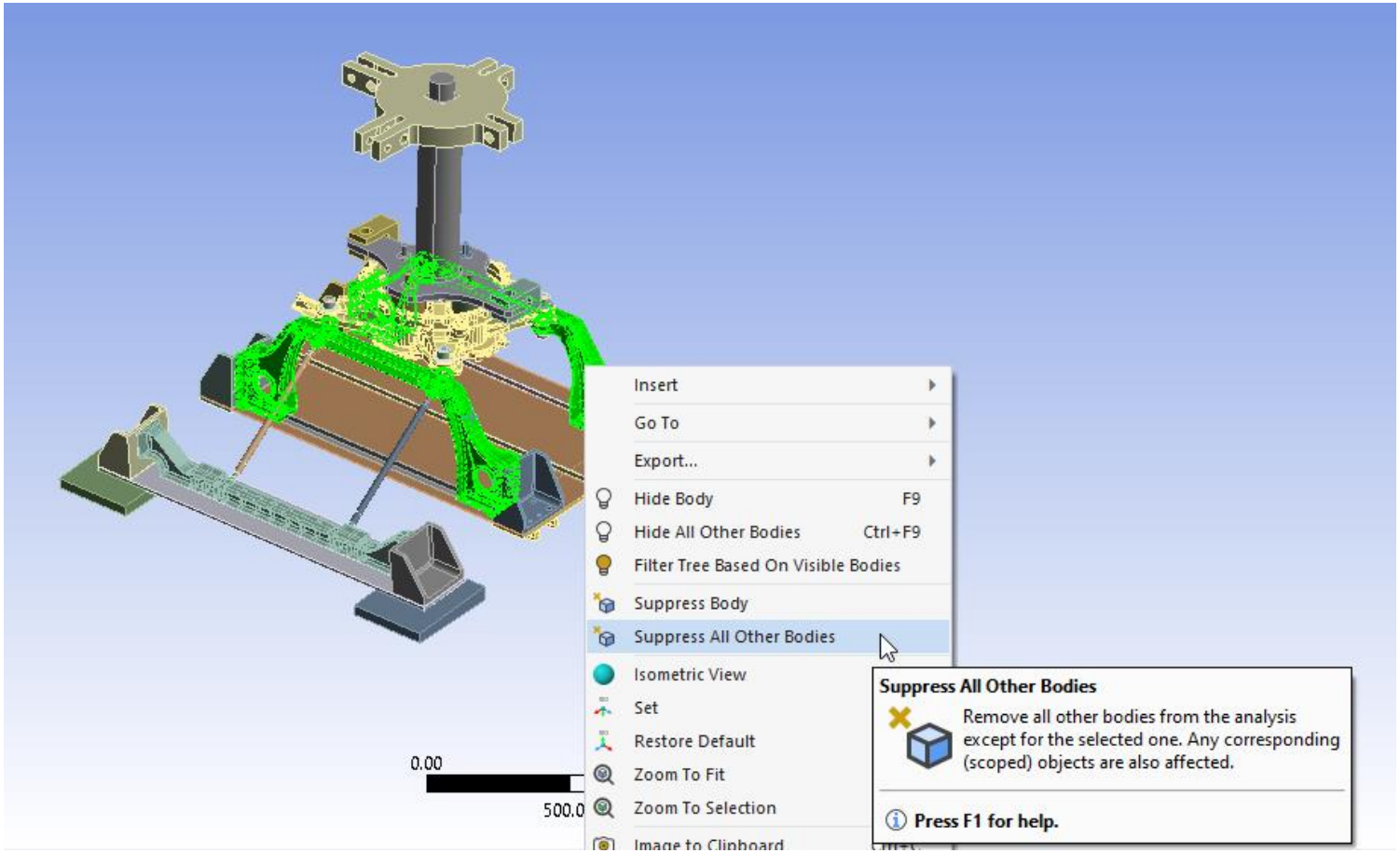


7. Import results in Ansys

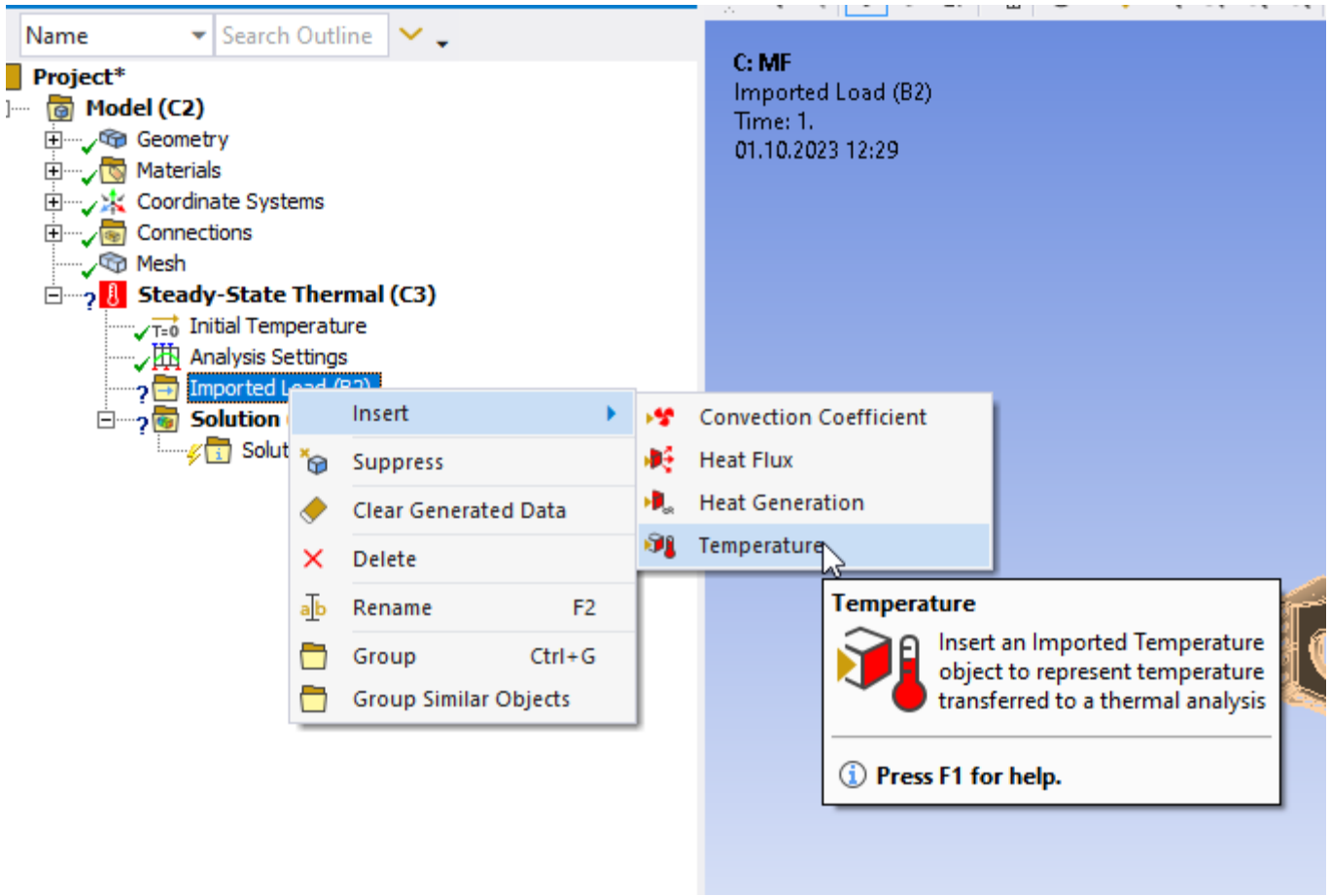


… then start Ansys Steady-State Thermal

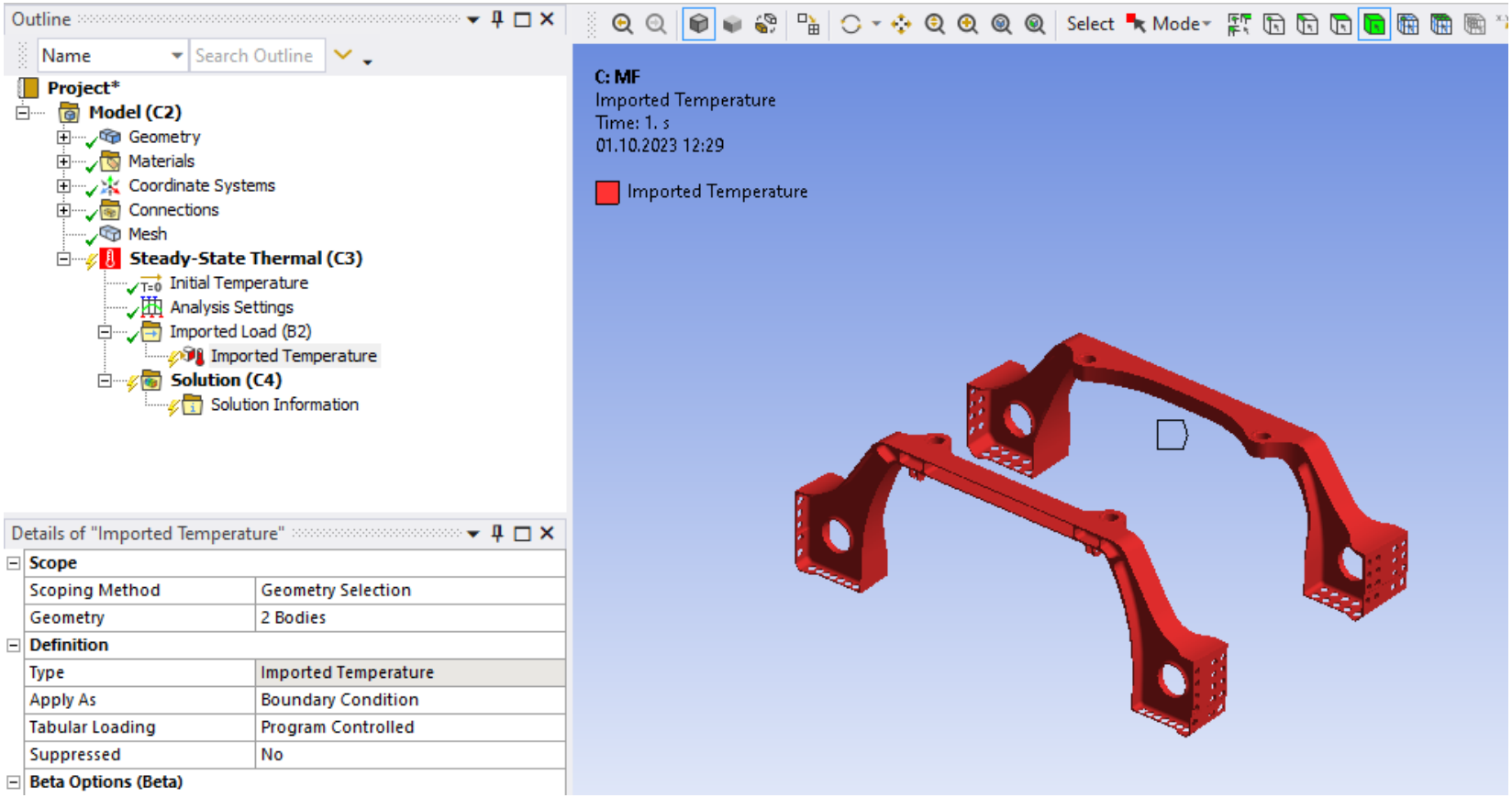
7. Import results in Ansys



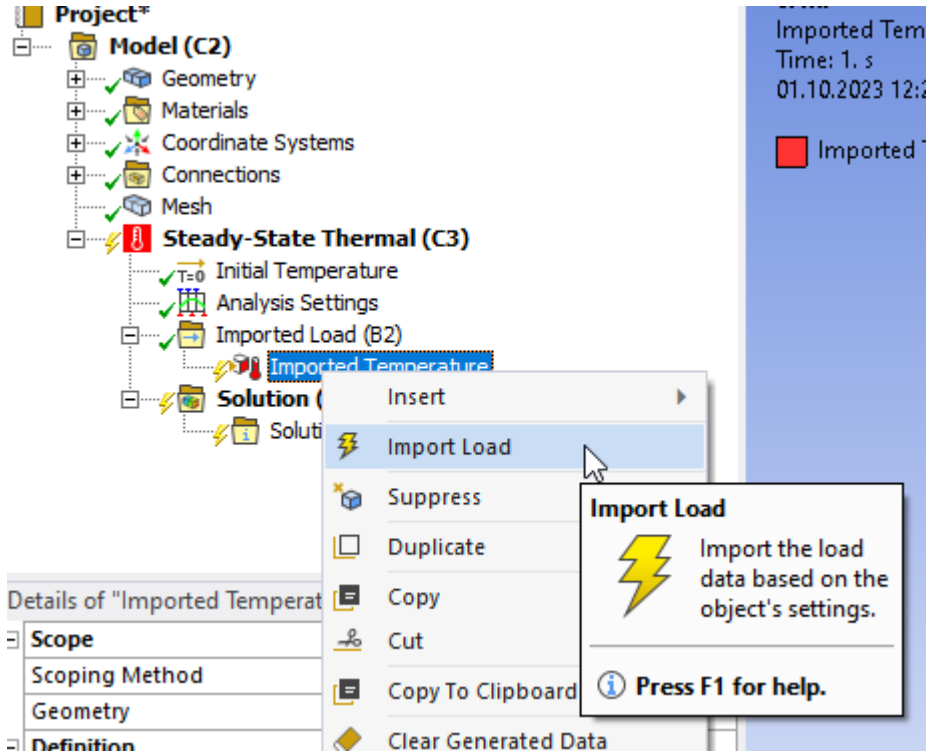
7. Import results in Ansys



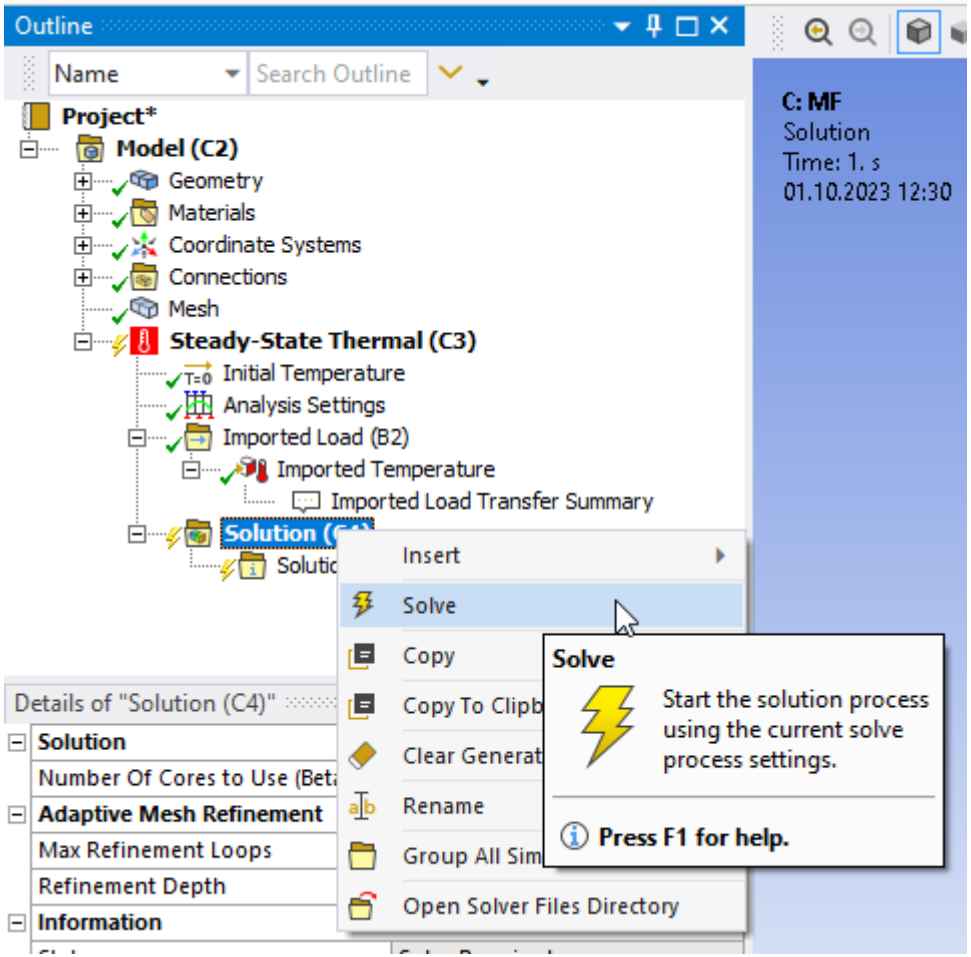
7. Import results in Ansys



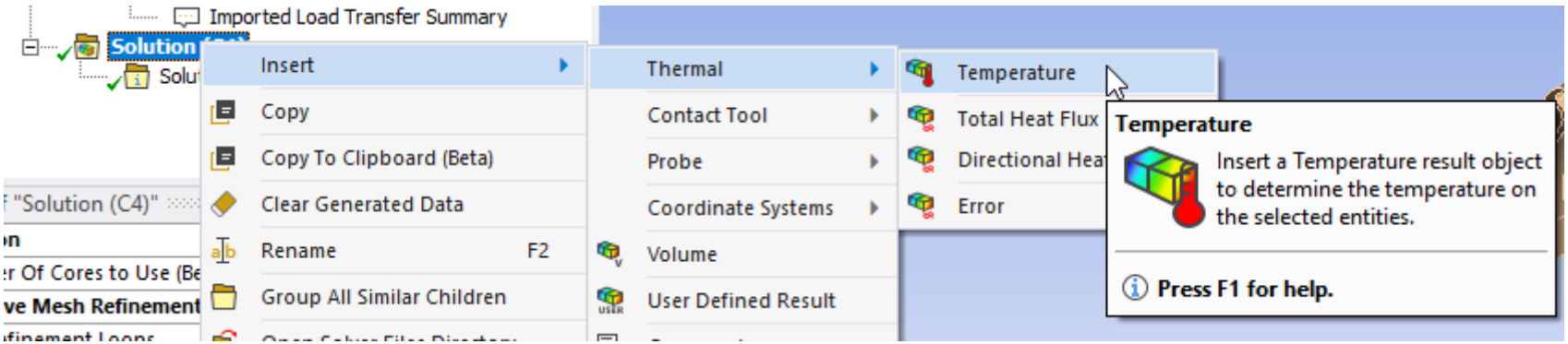
7. Import results in Ansys



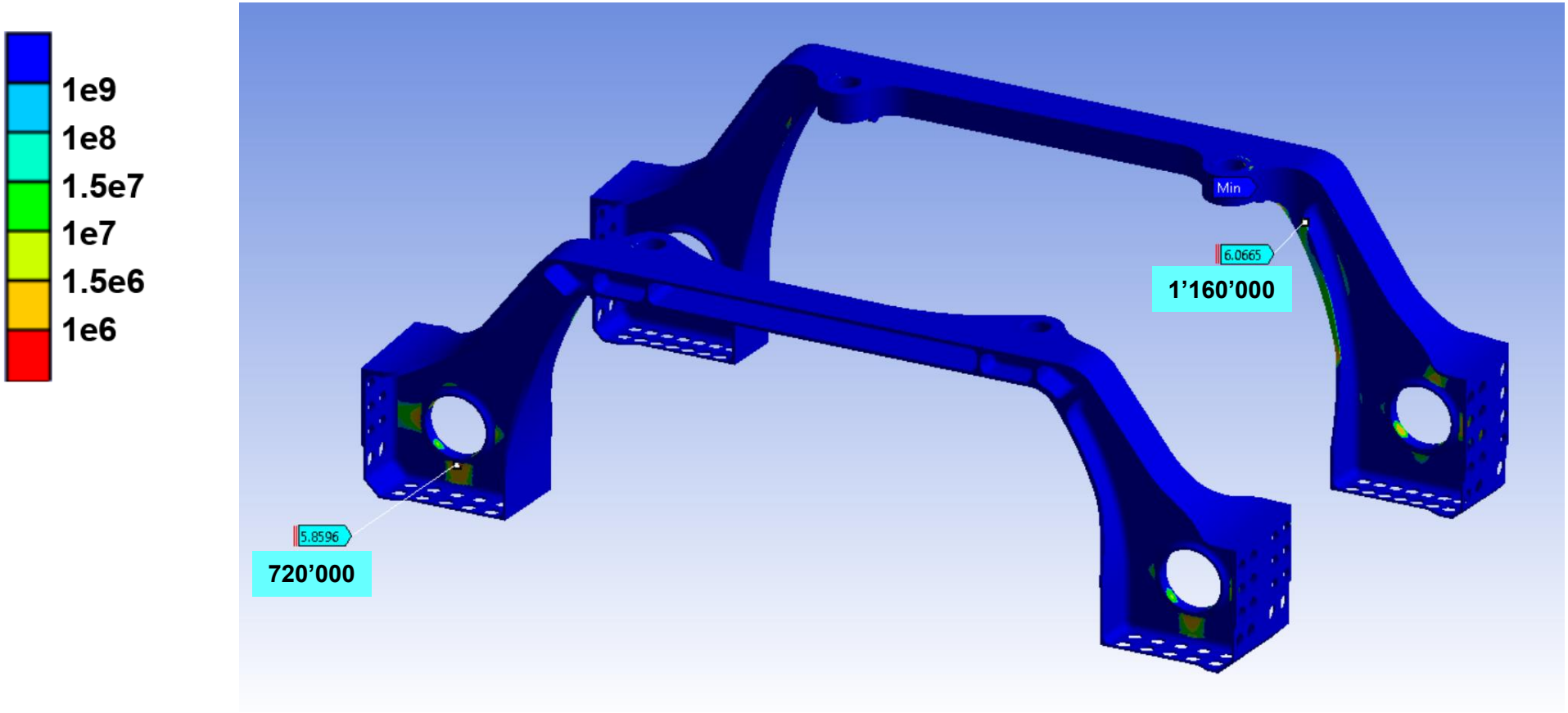
8. Run the dummy thermal analysis



8. Post process the “Temperature” (Log-Life)



8. Post process the “Temperature” (Log-Life)



MultiFat

Fatigue Analysis Tool

Quick Tutorial Ansys

END