



# INITSTATE EXAMPLE

ROBERTO SILVA

[www.esss.com.br](http://www.esss.com.br)

- This example is originated from a thermal-structural analysis
- One reason for performing a thermal-structural study of the welding process is to evaluate the residual stress field.
- Recall that residual stresses are summed to the operational stresses.

$$\sigma_{TOTAL} = \sigma_{DESIGN} + \sigma_{RESIDUAL}$$

- Since the thermal-structural study can be computationally very complex (large files), an efficient approach is to write a special initial state file, that can be read on other simulations.

- The first step is to create an initial state file (\*.ist) to be read.
  - File can be manually by the user

```

/CSYS,0
! ELEM ID    ELEM INTG    LAY/CELL    SECT INTG    SX    SY    SZ    SXY    SYZ    SXZ
  1 ,        1,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        2,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        3,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        4,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        5,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        6,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        7,        ,        ,        100,    0,    0,    0,    0,    0
  1 ,        8,        ,        ,        100,    0,    0,    0,    0,    0

```

- File can be automatically written by ANSYS, using the INISTATE command (shown next).
- The automatic creation is recommended, since it provides more precise results, but a file can be manually created if a previous simulation is not available.

- **INISTATE** command syntax for creation of files.

**INISTATE, WRITE, 1, , , , *CSID*, *Dtype***

- **CSID** defines reference coordinate system for results.
  - 0 : uses the Global Cartesian System
  - -1 or MAT : based on material coordinate system
  - -2 or ELEM : based on element's coordinate system
- **Dtype** defines which result will be written.
  - S : output stresses
  - EPEL : output elastic strain
  - EPPL : output plastic strain

- **INISTATE** is used again for reading the initial state file.

## **INISTATE, READ, *Fname, Ext***

- In WorkBench, initial state file has the file.ist default name.
- The same procedure for copying results files in submodeling and thermal-structural simulations can be used.

**/COPY, file, ist, , file, ist, ..\..\**

**Use this at the initial state model, at the Solution folder**

**/COPY, file, ist, ..\..\, file, ist**

**Use this at the main model, at the Environment folder**

- If a reference analysis is unavailable, initial state can be entered directly by two **INISTATE** instructions.

**INISTATE, SET, DTYP, *Data Type***

**INISTATE, DEFINE, *Elid, Eint, Klayer, ParmInt, Cxx, Cyy, Czz, Cxy, Cyz, Cxz***

*Data Type* is the type of result (as the previous labels shown)

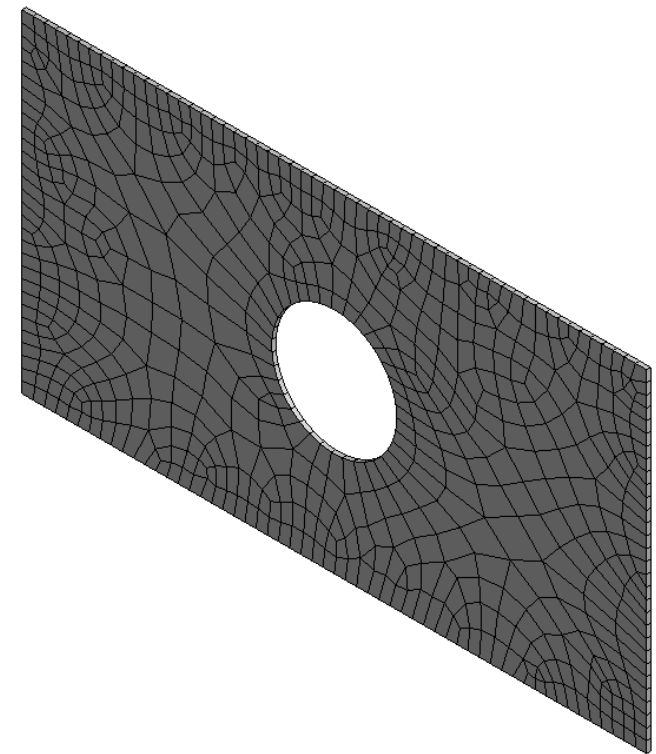
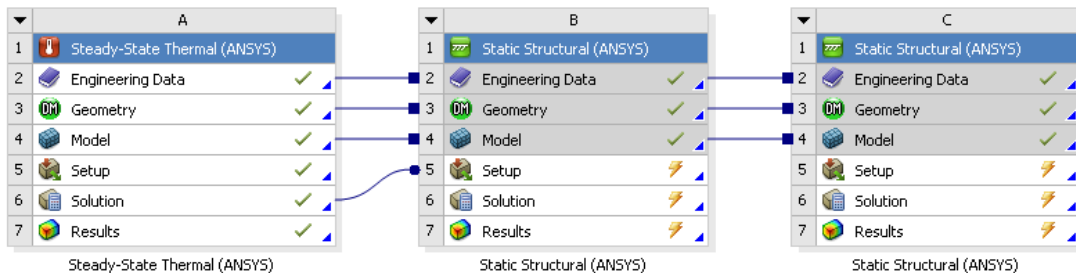
*Elid* is the element number (ALL can be used)

*Eint* is the Gauss integration point

*ParmInt* is the section integration point for shells and beams

*Cxx ... Cxz* are the component values

- Example: consider a plate with a hole.
  - First, a thermal study is defined to calculate temperature field.
  - A structural study is carried on, evaluating stresses due to temperature distribution.
  - Residual thermal stresses are used as initial condition for the model, when subjected to a tensile load.

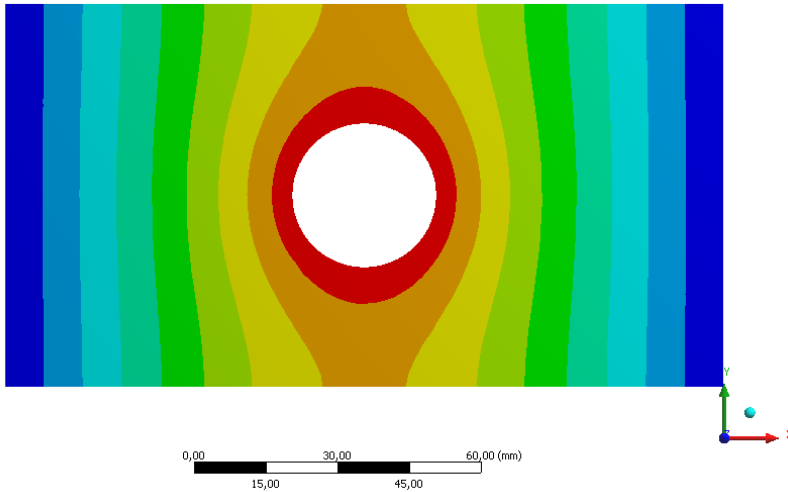


- Initial stress state evaluation.

A: Steady-State Thermal (ANSYS)

Temperature  
Type: Temperature  
Unit: °C  
Time: 1

50 Max  
47,778  
45,556  
43,333  
41,111  
38,889  
36,667  
34,444  
32,222  
30 Min



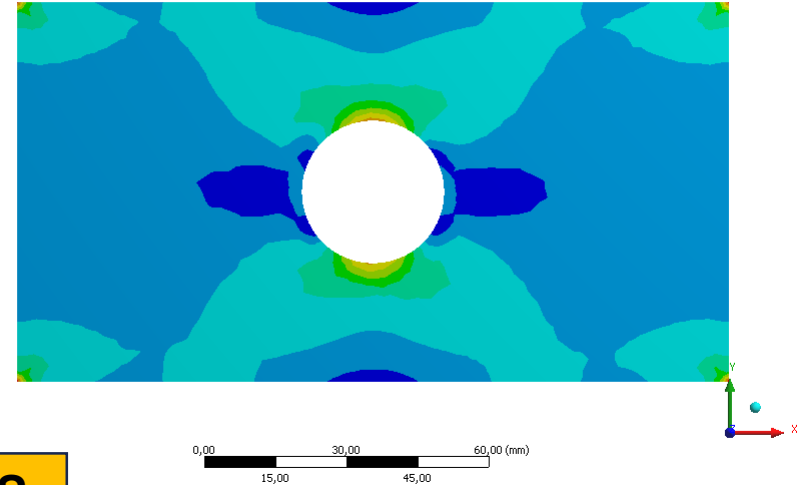
ANSYS  
v12.1

Temperature distribution from thermal study

B: Static Structural (ANSYS)

Equivalent Stress  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 1

178,6 Max  
159,09  
139,58  
120,08  
100,57  
81,068  
61,562  
42,057  
22,551  
3,0459 Min



ANSYS  
v12.1

Stress distribution

Static Structural (B5)

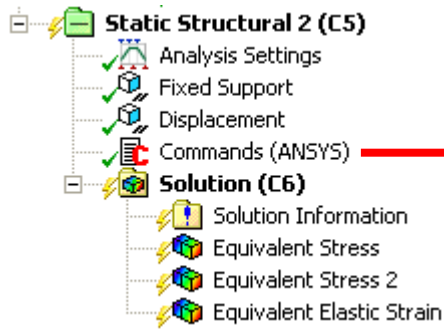
- Analysis Settings
- Fixed Support
- Commands (ANSYS)
- Imported Load (Solution)
- Solution (B6)
  - Solution Information
  - Equivalent Stress
  - Equivalent Elastic Strain
  - Commands (ANSYS)

```
INISTATE, WRITE, 1, , S
```

```
/COPY, file, ist, , file, ist, ..\..\
```



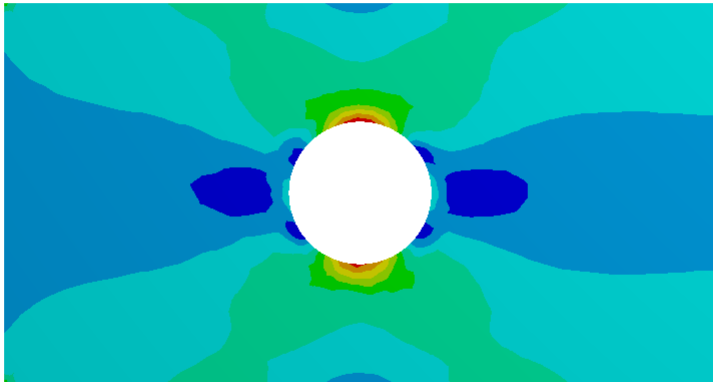
- Initial stress state reading.



```
/COPY, file, ist, ..\..\, file, ist  
INISTATE, READ, file, ist
```

C: Static Structural (ANSYS)  
Equivalent Stress 2  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 2

78,465 Max  
70,215  
61,965  
53,715  
45,465  
37,215  
28,965  
20,715  
12,465  
4,2155 Min



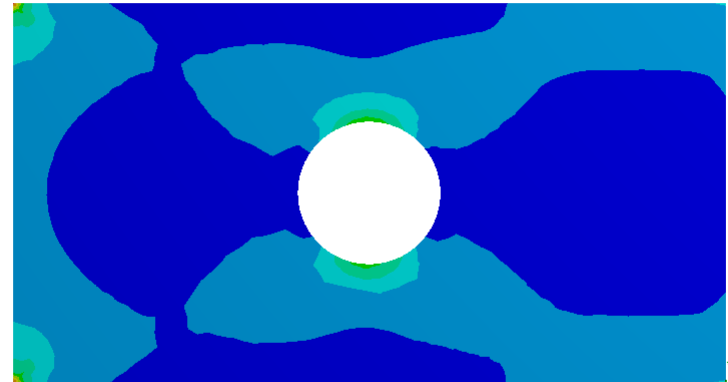
	Time [s]	<input checked="" type="checkbox"/> Minimum [MPa]	<input checked="" type="checkbox"/> Maximum [MPa]
1	1,	0,	0,
2	2,	4,2155	78,465

Without initial state

ANSYS  
v12.1

C: Static Structural (ANSYS)  
Equivalent Stress 2  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 2

134,28 Max  
119,39  
104,49  
89,594  
74,697  
59,801  
44,905  
30,008  
15,112  
0,21521 Min



	Time [s]	<input checked="" type="checkbox"/> Minimum [MPa]	<input checked="" type="checkbox"/> Maximum [MPa]
1	1,	2,7912	176,63
2	2,	0,21521	134,28

With initial state

ANSYS  
v12.1