Anhang L – Input Files

1. Aufbau des Gesamtmodells

```
**********************************************************************
**                    Allgemeines File zur Modellerstellung          **
**********************************************************************
*Heading
** Job name: Modell_Gesamt Model name: Modell_1_pl
** Generated by: Abaqus/CAE 6.12-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**EINHEITEN in SI-Einheiten [m], [kg], [s]
**
**********************************************************************
**                    Erstellung der Parts                           **
**********************************************************************
**
** Die Einzelnen Sets sowie Nodes werden hier nicht dargestellt
**
** PARTS
**
*Part, name=Gesamt
*Node
1,  203.601913,  248.721283,  92.4912262
 ...
3531259,  3795, 22315, 85321
**
*Nset, nset=Ausbruch
*Nset, nset=Dolomit
*Nset, nset=Gips
*Nset, nset=Tonreich
*Nset, nset=ALL, generate
*Nset, nset=Ausbau_Stollen_SET
*Nset, nset=Ausbau_Kammer_SET
*Nset, nset=_Ausbau_Stollen_Skin, internal
*Nset, nset=_Ausbau_Stollen_Skin, internal
```

```
*Nset, nset=_Ausbau_Kammer_Skin, internal
*Nset, nset=Ausbau_Stollen_PIC
*Nset, nset=Ausbau_Kammer_PIC
*Nset, nset=GOK
*Nset, nset=BC-1
*Nset, nset=BC-2
*Nset, nset=BC-3
**
*Elset, elset=Ausbruch
*Elset, elset=Gips
*Elset, elset=Tonreich
*Elset, elset=ALL, generate
*Elset, elset=Ausbau_Stollen_SET
*Elset, elset=Ausbau_Kammer_SET
*Elset, elset=_Ausbau_Stollen_Skin, internal, generate
*Elset, elset=_Ausbau_Kammer_Skin, internal, generate
*Elset, elset=Ausbau_Stollen_PIC, generate
*Elset, elset=Ausbau_Kammer_PIC, generate
*Elset, elset=GOK
*Elset, elset =BC-1
*Elset, elset =BC-2
*Elset, elset =BC-3
**
** Das Elset Elastisch_PEEQ wurde für „Distortet Elements“ angelegt
**
*Elset, elset=Elastisch_PEEQ → Distortete element easdf
**
** Section: Elastisch
*Solid Section, elset=Elastisch_PEEQ, material=Elastisch
**
** Section: Ton
*Solid Section, elset=Tonreich, material=Ton
** Section: Gips
*Solid Section, elset=Gips, material=Gips
** Section: Dolomit
*Solid Section, elset=Dolomit, material=Dolomit
** Section: Ausbau
*Shell Section, elset=Ausbau_Stollen_PIC, material=Ausbau
0.1, 5
** Section: Ausbau
*Shell Section, elset=Ausbau_Kammer_PIC, material=Ausbau
0.1, 5
**
*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Gesamt-1, part=Gesamt
*End Instance
**
```

```
*End Assembly
**
```

2. Plastische Berechnung inkl. Parameterstudie

```
**********************************************************************
**                  Materialparameter und Materialmodell             **
**                  die Materialparameter sind im Kapitel            **
**                  VI NUMERISCHE BERECHNUNGEN  angegeben            **
**********************************************************************
**
** MATERIALS
**
*Material, name=Ausbau
*Density
1.,
*Elastic
 5e+09, 0.2
*Material, name=Dolomit
*Density
2400.,
*Elastic
 5.36e+10, 0.297
*Mohr Coulomb
 30.59, 0.01,   0.
 …
 30.59, 0.01,  28.
*Mohr Coulomb Hardening
 4.21e+06, 0., 0.
… Siehe  VI NUMERISCHE BERECHNUNGEN
 4.21e+06, 0.,28.
*Material, name=Elastisch
*Density
2300.,
*Elastic
 1.015e+10, 0.358
*Material, name=Gips
*Density
2300.,
*Elastic
 1.015e+10, 0.358
*Mohr Coulomb
 31.11, 0.01,   0.
….
  18., 0.09,  28.
*Mohr Coulomb Hardening
 6.3e+06, 0., 0.
 ….
 100000., 0.,28.
*Tension Cutoff
520000., 0., 0.
….
```

```
520000., 0.,28.
*Material, name=Ton
*Density
2300.,
*Elastic
 6.63e+09, 0.358
*Mohr Coulomb
 31.11, 0.01,   0.
....
  18., 0.09,  28.
*Mohr Coulomb Hardening
  6.3e+06, 0., 0.
....
  100000., 0.,28.
*Tension Cutoff
520000., 0., 0.
....
520000., 0.,28.
**
**********************************************************************
**                      Randbedingungen                               **
**********************************************************************

** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet30, 2, 2
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet31, 1, 1
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet32, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Gesamt-1.ALL, 1.
**
**********************************************************************
**                      Berechnungssteps                              **
**********************************************************************
**
**
*Initial Conditions, Type=stress, File=C:\_Gschwandtner\Modell_1.odb, Step=1, Inc=1
**
** STEP: Grav
**
*Step, name=Grav, unsymm=YES
*Geostatic
```

```
**
** LOADS
**
** Name: Grav   Type: Gravity
*Dload
, GRAV, 10., 0., 0., -1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, remove
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
*Model Change, remove
Gesamt-1.Ausbau_Stollen_PIC,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
*Print, solve=NO
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, POR, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S, SAT, VOIDR
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Ausbruch
**
*Step, name=Ausbruch, unsymm=YES
*Static
0.2, 1., 1e-05, 1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, add
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
*Model Change, add
Gesamt-1.Ausbau_Stollen_PIC,
** Interaction: Ausbruch
```

```
*Model Change, remove
Gesamt-1.Ausbruch,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Mittelwert
**
*Step, name=Mittelwert, unsymm=YES
*Static
0.02, 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-2   Type: Temperature
*Temperature
Gesamt-1.ALL, 1.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
```

```
*End Step
** ----------------------------------------------------------------
**
** STEP: Median
**
*Step, name=Median, unsymm=YES
*Static
0.02, 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-3   Type: Temperature
*Temperature
Gesamt-1.ALL, 2.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: min_1
**
*Step, name=min_1, unsymm=YES
*Static
0.02, 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-4   Type: Temperature
*Temperature
Gesamt-1.ALL, 3.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
```

```
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: min_2
**
*Step, name=min_2, unsymm=YES
*Static
0.02, 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-5   Type: Temperature
*Temperature
Gesamt-1.ALL, 4.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------------------------------
** STEP: min_3 bis STEP: min_25 gleich wie zuvor
** ----------------------------------------------------------------------------------------
**
** STEP: min_26
**
*Step, name=min_26
```

```
*Static
0.02, 1., 1e-05, 1.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-29   Type: Temperature
*Temperature
Gesamt-1.ALL, 28.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```

3. Gesamtstabilität unter Berücksichtigung der Laugung des Deckgebirges durch Niederschlagswässer

3.1. File zur Ermittlung der Temperaturverteilung

```
**********************************************************************
**                    Allgemeines File zur Modellerstellung          **
**********************************************************************
**
*Heading
** Job name: Temp_1 Model name: TEMP
** Generated by: Abaqus/CAE 6.12-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**EINHEITEN in SI-Einheiten [m], [kg], [s]
**
**********************************************************************
**                    Erstellung der Parts wie zovor                 **
**********************************************************************
**
**----------------------------------------------------------------------------------------
**
```

```
** Koordninatensysteme für die ermittlung der Temperaturverteilung
*Orientation, name=Ori-4
      1.,        0.,        0.,        0.,        1.,        0.
**
**********************************************************************
**                    Materialparameter und Materialmodell            **
**                    die Materialparameter sind im Kapitel           **
**                    VI NUMERISCHE BERECHNUNGEN  angegeben           **
**********************************************************************
**
** MATERIALS
**
*Material, name=Ausbau
*Conductivity, type=ORTHO
0.,0.,0.
*Density
1.,
*Elastic
 5e+09, 0.2
*Specific Heat
1090.,
*Material, name=Dolomit
*Conductivity, type=ORTHO
0.,   0.,10000.
*Density
2400.,
*Elastic
 5.36e+10, 0.297
*Specific Heat
1090.,
*Mohr Coulomb
 30.59, 0.01
*Mohr Coulomb Hardening
 4.21e+06,0.
*Material, name=Gips
*Conductivity, type=ORTHO
0.,   0.,10000.
*Density
2300.,
*Elastic
 1.015e+10, 0.358
*Specific Heat
1090.,
*Mohr Coulomb
 31.11, 0.01
*Mohr Coulomb Hardening
 6.3e+06,0.
*Material, name=Ton
*Conductivity, type=ORTHO
0.,   0.,10000.
*Density
2300.,
```

```
*Elastic
 6.63e+09, 0.358
*Specific Heat
1090.,
*Mohr Coulomb
 31.11, 0.01
*Mohr Coulomb Hardening
 6.3e+06,0.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Gesamt-1.ALL, 1., 1., 1., 1., 1.
**
**********************************************************************
**                    Berechnungssteps                              **
**********************************************************************
**
** STEP: Heat
**
*Step, name=Heat, inc=100000
*Heat Transfer, end=PERIOD, deltmx=10.
1., 250000., 0.01, 2500.,
**

**********************************************************************
**                    Randbedingung als Temperaturbelastung         **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Temperature
*Boundary
Gesamt-1.GOK, 11, 11, 100.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
*Output, history, frequency=0
*End Step
```

3.2. File zur Ermittlung der Gesamtstabilität

```
**********************************************************************
**                    Allgemeines File zur Modellerstellung         **
**********************************************************************
**
```

```
*Heading
** Job name: Temp_1 Model name: TEMP
** Generated by: Abaqus/CAE 6.12-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**EINHEITEN in SI-Einheiten [m], [kg], [s]
**
**********************************************************************
**                    Erstellung der Parts wie zovor                  **
**********************************************************************
**
**----------------------------------------------------------------------------------------
**
**********************************************************************
**                    Materialparameter und Materialmodell            **
**                    die Materialparameter sind im Kapitel           **
**                    VI NUMERISCHE BERECHNUNGEN  angegeben           **
**********************************************************************
**
** MATERIALS
**
*Material, name=Elastisch
*Density
2300.,
*Elastic
 1.015e+10, 0.358
**
*Material, name=Ausbau
*Density
1.,
*Elastic
 5e+09, 0.2
*Material, name=Dolomit
*Density
2400.,
*Elastic
 5.36e+10, 0.297
*Mohr Coulomb
 30.59, 0.01,   0.
...
 30.59, 0.01, 100.
*Mohr Coulomb Hardening
 4.21e+06,  0.,  0.
...
 4.21e+06,  0.,100.
*Material, name=Gips
*Density
2300.,
*Elastic
  1.053e+10, 0.358,    0.
...
 2.48133e+07, 0.358,  100.
*Mohr Coulomb
```

```
 26.3, 0.01,   0.
…
 18.14, 0.01, 100.
*Mohr Coulomb Hardening
 2.89e+06,  0.,  0.
…
 10175.9,  0.,100.
*Tension Cutoff
520000., 0., 0.
…
520000., 0., 100.
*Material, name=Ton
*Density
2300.,
*Elastic
   6.63e+09, 0.358,    0.
…
 2.48319e+07, 0.358,  100.
*Mohr Coulomb
 26.3, 0.01,   0.
 …
 18.14, 0.01, 100.
*Mohr Coulomb Hardening
 2.89e+06,  0.,  0.
…
 10175.9,  0.,100.
*Tension Cutoff
520000., 0., 0.
…
520000., 0., 100.
**
**********************************************************************
**                        Randbedingung                               **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
_PickedSet30, 2, 2
** Name: BC-2 Type: Displacement/Rotation
*Boundary
_PickedSet31, 1, 1
** Name: BC-3 Type: Displacement/Rotation
*Boundary
_PickedSet32, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Anfangstemp   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Gesamt-1.ALL, 0.
```

```
**
**********************************************************************
**                        Berechnungssteps                            **
**********************************************************************
**
*Initial Conditions, Type=stress, File=C:\_Gschwandtner\Modell_1.odb, Step=1, Inc=1
**
** STEP: Grav
**
*Step, name=Grav, unsymm=YES
*Geostatic
**
** LOADS
**
** Name: Grav   Type: Gravity
*Dload
, GRAV, 10., 0., 0., -1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, remove
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
*Model Change, remove
Gesamt-1.Ausbau_Stollen_PIC,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
*Print, solve=NO
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, POR, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S, SAT, TEMP, VOIDR
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Ausbruch
**
*Step, name=Ausbruch, unsymm=YES
*Static
```

```
0.2, 1., 1e-05, 1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, add
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
*Model Change, add
Gesamt-1.Ausbau_Stollen_PIC,
** Interaction: Ausbruch
*Model Change, remove
Gesamt-1.Ausbruch,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S, TEMP
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Heat
**
*Step, name=Heat, unsymm=YES
*Static
0.1, 250000., 0.001, 250000.
**
** PREDEFINED FIELDS
**
** Name: Anfangstemp   Type: Temperature
*Temperature, op=NEW
** Name: Temp_Field   Type: Temperature
*Temperature, op=NEW, file=C:\_Gschwandtner\Heat\Temp_1.odb
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
*Print, solve=YES
**
```

```
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
LE, PE, PEEQ, S, TEMP
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```

4. Gesamtstabilität unter Berücksichtigung der des Laugungspotentials des unterirdischen Sees in der unteren Sohle

4.1. File zur Ermittlung der Temperaturverteilung

```
**********************************************************************
**                    Allgemeines File zur Modellerstellung          **
**********************************************************************
**
*Heading
** Job name: Laugung Model name: 02_TEMP_US
** Generated by: Abaqus/CAE 6.12-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**EINHEITEN in SI-Einheiten [m], [kg], [s]
**
**********************************************************************
**                    Erstellung der Parts wie zovor                 **
**********************************************************************
**
**----------------------------------------------------------------------------------------
**
*Nset, nset=Gips_TEMP_Down
*Nset, nset=Gips_NT
*Nset, nset=GipsT
*Nset, nset=Tonreich_TEMP_Down
*Nset, nset=Tonreich_NT
*Nset, nset=TonT
*Nset, nset=TonNT
*Nset, nset=BC_Temp
**
*Elset, elset =Gips_TEMP_Down
*Elset, elset =Gips_NT
*Elset, elset =GipsT
*Elset, elset =Tonreich_TEMP_Down
```

```
*Elset, elset =Tonreich_NT
*Elset, elset =TonT
*Elset, elset =TonNT
*Elset, elset =BC_Temp
**
**Koordninatensysteme für die ermittlung der Temperaturverteilung
*Orientation, name=Ori-4
      1.,        0.,        0.,        0.,        1.,        0.
3, 0.
** Section: Gips_NT
*Solid Section, elset=GipsNT, orientation=Ori-4, material=Gips_NT
,
** Section: Dolomit_NT
*Solid Section, elset=Dolomit, orientation=Ori-4, material=Dolomit_NT
,
** Section: Ton_NT
*Solid Section, elset=TonNT, orientation=Ori-4, material=Ton_NT
,
** Section: Gips_T_Down
*Solid Section, elset=GipsT, orientation=Ori-4, material=Gips_TEMP_DOWN
,
*Orientation, name=Ori-5
      1.,        0.,        0.,        0.,        1.,        0.
3, 0.
** Section: Ausbau
*Shell Section, elset=Ausbau_Kammer_PIC, material=Ausbau, orientation=Ori-5
0.1, 5
** Section: Ton_T_Down
*Solid Section, elset=TonT, orientation=Ori-4, material=Ton_TEMP_DOWN
,
*Orientation, name=Ori-6
      1.,        0.,        0.,        0.,        1.,        0.
3, 0.
**
**********************************************************************
**                    Materialparameter und Materialmodell            **
**                    die Materialparameter sind im Kapitel           **
**                    VI NUMERISCHE BERECHNUNGEN  angegeben           **
**********************************************************************
**
** MATERIALS
**
*Material, name=Ausbau
*Conductivity
0.,
*Density
1.,
*Elastic
 1e+10, 0.2
*Specific Heat
1090.,
*Material, name=Dolomit_NT
```

```
*Conductivity
0.,
*Density
2400.,
*Elastic
 5.36e+10, 0.297,    0.
…
 5.36e+10, 0.297,  100.
*Specific Heat
1090.,
*Mohr Coulomb
 30.59, 0.01,   0.
…
 30.59, 0.01, 100.
*Mohr Coulomb Hardening
 4.21e+06,  0.,  0.
…
 4.21e+06,  0.,100.
*Material, name=Gips_NT
*Conductivity
0.,
*Density
2300.,
*Elastic
  1.053e+10, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
    100.,  0.,160.
*Material, name=Gips_TEMP_DOWN
*Conductivity, type=ORTHO
10000.,10000., 5000.
*Density
2300.,
*Elastic
  1.053e+10, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
```

```
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
     100.,  0.,160.
*Material, name=Ton_NT
*Conductivity
0.,
*Density
2300.,
*Elastic
   6.63e+09, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
     100.,  0.,160.
*Material, name=Ton_TEMP_DOWN
*Conductivity, type=ORTHO
10000.,10000., 5000.
*Density
2300.,
*Elastic
   6.63e+09, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
     100.,  0.,160.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Gesamt-1.ALL, 1., 1., 1., 1., 1.
**********************************************************************
**                      Berechnungssteps                              **
**********************************************************************
**
```

```
** STEP: Heat
**
*Step, name=Heat, inc=100000
*Heat Transfer, end=PERIOD, deltmx=10.
1., 1000., 0.01, 500.,
**
**********************************************************************
**                    Randbedingung als Temperaturbelastung          **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Temperature
*Boundary
Gesamt-1.BC_Temp, 11, 11, 10.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Contact Output
HTL,
*End Step
** ----------------------------------------------------------------
**
** STEP: Heat_2
**
*Step, name=Heat_2
*Heat Transfer, end=PERIOD, deltmx=10.
1., 1000., 0.01, 500.,
**
**********************************************************************
**                    Randbedingung als Temperaturbelastung          **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Temperature
*Boundary, op=NEW
** Name: BC-2 Type: Temperature
*Boundary, op=NEW
Gesamt-1.BC_Temp, 11, 11, 50.
**
** OUTPUT REQUESTS
**
```

```
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Contact Output
HTL,
*End Step
** ----------------------------------------------------------------
**
** STEP: Heat_3
**
*Step, name=Heat_3
*Heat Transfer, end=PERIOD, deltmx=10.
1., 1500., 0.01, 500.,
**
**********************************************************************
**                    Randbedingung als Temperaturbelastung          **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-2 Type: Temperature
*Boundary, op=NEW
** Name: BC-3 Type: Temperature
*Boundary, op=NEW
Gesamt-1.BC_Temp, 11, 11, 100.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Contact Output
HTL,
*End Step
** ----------------------------------------------------------------
**
** STEP: Heat_4
**
*Step, name=Heat_4
*Heat Transfer, end=PERIOD, deltmx=10.
```

```
0.1, 15000., 0.05, 5000.,
**
**************************************************************************
**                    Randbedingung als Temperaturbelastung               **
**************************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-3 Type: Temperature
*Boundary, op=NEW
** Name: BC-4 Type: Temperature
*Boundary, op=NEW
Gesamt-1.BC_Temp, 11, 11, 150.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Contact Output
HTL,
*End Step
```

4.2. File zur Ermittlung der Gesamtstabilität

```
**************************************************************************
**                    Allgemeines File zur Modellerstellung               **
**************************************************************************
**
*Heading
** Job name: Gesamtmodell_Laugung_U2 Model name: 02_Mech_US
** Generated by: Abaqus/CAE 6.12-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**EINHEITEN in SI-Einheiten [m], [kg], [s]
**
**************************************************************************
**                    Erstellung der Parts wie zovor                      **
**************************************************************************
**
**--------------------------------------------------------------------------------------------
**
**************************************************************************
**                    Materialparameter und Materialmodell                **
**                    die Materialparameter sind im Kapitel               **
**                    VI NUMERISCHE BERECHNUNGEN  angegeben               **
```

```
**************************************************************************
**
** MATERIALS
**
*Material, name=Ausbau
*Conductivity
0.,
*Density
1.,
*Elastic
 1e+10, 0.2
*Specific Heat
1090.,
*Material, name=Dolomit_NT
*Conductivity
0.,
*Density
2400.,
*Elastic
 5.36e+10, 0.297,    0.
…
 5.36e+10, 0.297,  100.
*Specific Heat
1090.,
*Mohr Coulomb
 30.59, 0.01,   0.
…
 30.59, 0.01, 100.
*Mohr Coulomb Hardening
 4.21e+06,  0.,  0.
…
 4.21e+06,  0.,100.
*Material, name=Gips_NT
*Conductivity
0.,
*Density
2300.,
*Elastic
  1.053e+10, 0.358,    0.
…
     10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
  13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
 …
      100.,  0.,160.
*Material, name=Gips_TEMP_DOWN
```

```
*Conductivity, type=ORTHO
0., 0., 0.
*Density
2300.,
*Elastic
  1.053e+10, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
     100.,  0.,160.
*Material, name=Ton_NT
*Conductivity
0.,
*Density
2300.,
*Elastic
   6.63e+09, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
…
     100.,  0.,160.
*Material, name=Ton_TEMP_DOWN
*Conductivity, type=ORTHO
0., 0., 0.
*Density
2300.,
*Elastic
   6.63e+09, 0.358,    0.
…
    10000., 0.358,  160.
*Specific Heat
1090.,
*Mohr Coulomb
 26.3, 0.01,   0.
…
 13., 0.01, 160.
```

```
*Mohr Coulomb Hardening
  2.89e+06,  0.,  0.
...
     100.,  0.,160.
**
**********************************************************************
**                        Randbedingung                               **
**********************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
Set-1, 1, 1
** Name: BC-2 Type: Displacement/Rotation
*Boundary
Set-2, 2, 2
** Name: BC-3 Type: Displacement/Rotation
*Boundary
Set-3, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
Set-4, 0.
**
**********************************************************************
**                        Berechnungssteps                            **
**********************************************************************
**
**
*Initial Conditions, Type=stress, File=C:\_Gschwandtner\Heat_UntereS\Modell_1.odb, Step=1, Inc=1
**
**
** STEP: Grav
**
*Step, name=Grav
*Geostatic
**
** LOADS
**
** Name: Gravity   Type: Gravity
*Dload
, GRAV, 10., 0., 0., -1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, remove
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
```

```
*Model Change, remove
Gesamt-1.Ausbau_Stollen_PIC,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, POR, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, SAT, TEMP, VOIDR
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Ausbruch
**
*Step, name=Ausbruch, unsymm=YES
*Static
0.01, 1., 1e-05, 1.
**
** INTERACTIONS
**
** Interaction: Ausbau_Kammer
*Model Change, add
Gesamt-1.Ausbau_Kammer_PIC,
** Interaction: Ausbau_Stollen
*Model Change, add
Gesamt-1.Ausbau_Stollen_PIC,
** Interaction: Ausbruch_1
*Model Change, remove
Gesamt-1.Ausbruch,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, TEMP
```

```
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Laugung_1
**
*Step, name=Laugung_1, unsymm=YES
*Static
0.01, 1000., 0.001, 1000.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-1   Type: Temperature
*Temperature, op=NEW
** Name: Predefined Field-2   Type: Temperature
*Temperature, op=NEW, file=C:\_Gschwandtner\Heat_UntereS\Laugung.odb, binc=1, estep=1,
einc=14
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, TEMP
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Laugung_2
**
*Step, name=Laugung_2, unsymm=YES
*Static
0.01, 1000., 0.001, 1000.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-2   Type: Temperature
```

```
*Temperature, op=NEW
** Name: Predefined Field-3   Type: Temperature
*Temperature, op=NEW, file=C:\_Gschwandtner\Heat_UntereS\Laugung.odb, bstep=2, estep=2, einc=23
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, TEMP
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Laugung_3
**
*Step, name=Laugung_3, unsymm=YES
*Static
0.01, 1000., 0.001, 1000.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-3   Type: Temperature
*Temperature, op=NEW
** Name: Predefined Field-4   Type: Temperature
*Temperature, op=NEW, file=C:\_Gschwandtner\Heat_UntereS\Laugung.odb, bstep=3, estep=3, einc=30
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, TEMP
*Contact Output
CDISP, CSTRESS
```

```
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Laugung_4
**
*Step, name=Laugung_4, unsymm=YES
*Static
0.01, 20000., 0.001, 20000.
**
** PREDEFINED FIELDS
**
** Name: Predefined Field-4   Type: Temperature
*Temperature, op=NEW
** Name: Predefined Field-5   Type: Temperature
*Temperature, op=NEW, file=C:\_Gschwandtner\Heat_UntereS\Laugung.odb, bstep=4, estep=4,
einc=50
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
CF, NT, RF, U
*Element Output, directions=YES
E, LE, PEEQ, PEEQMAX, S, TEMP
*Contact Output
CDISP, CSTRESS
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
```