CML TFC Code (beta version) User's Manual

Jinglin Zheng and David B. Bogy

Computer Mechanics Laboratory

Department of Mechanical Engineering

University of California

Berkeley, CA 94720

May 2009

1 Introduction

The CML TFC code includes a set of macro files written in APDL (ANSYS Parametric Design Language), which can be directly executed in the commercial finite element software, ANSYS. These macros use an iterative approach [1] to solve the flying-attitude and thermal deformation of a slider with micro-heaters built within (Thermal Flying-height Control or TFC slider). Fig. 1 shows the flowchart of this solution procedure. This process starts from solving the generalized Reynolds equation to obtain the pressure profile in the air bearing and the spacing in the head-disk interface (HDI). The heat transfer coefficients on the air bearing surface (ABS) are calculated and input to the finite element model of the slider. A coupled-field analysis is then conducted to calculate the deformation of the slider. Considering the effects of the ABS deformation on the flying-attitude, the original ABS shape is updated according to this deformation and used for a new round of spacing and pressure calculations. This procedure continues until the flying-attitudes obtained from two consecutive iterations have a difference less than some specified value. In this procedure, the CML quick4 solver is adopted to solve the generalized Reynolds equation using the finite volume method [2]. The commercial finite element software, ANSYS, is used to execute the electro-thermal-structural analysis. The CML TFC code runs on a windows-platform computer with certain ANSYS products (modules for electric, thermal and structural analysis) installed.
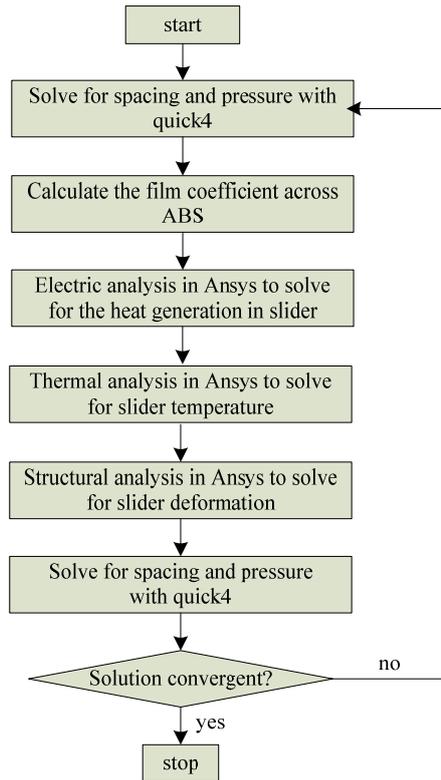
Fig. 1 Flowchart of the TFC solution procedure

2 Related files and structure

2.1 File structure
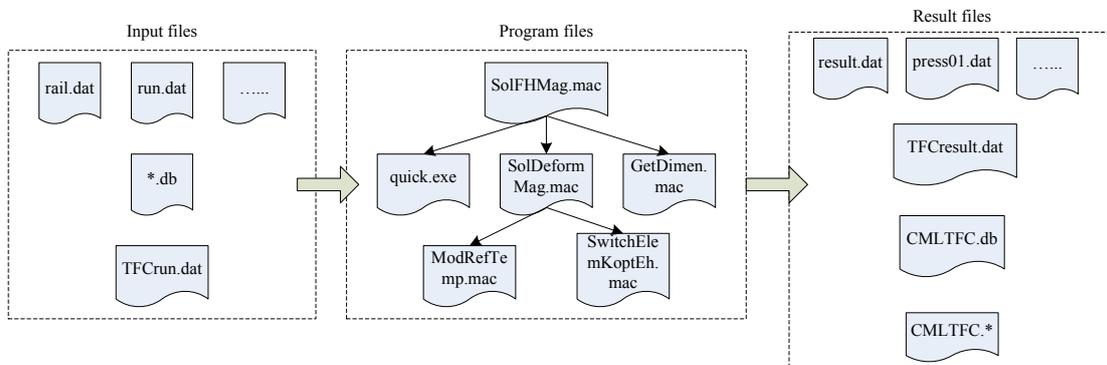
Fig. 2 File structure of the TFC code

Fig. 2 shows a complete list of files and the relationships between them in the CML TFC code.

The left block contains input files that define the problem to be solved. The files on the first row define the geometry of the air bearing surface, the suspension load, initial flying attitudes, etc. The *.db* on the second row is an ANSYS database file defining the geometry of the slider. *TFCrun.dat* on the third row defines parameters

associated with the thermal flying height control.

The middle block contains all of the program files responsible for executing the TFC calculations.

The right block contains 3 groups of output files. The first row includes standard output files generated by the quick4 solver. TFCresult.dat on the second row records the converging process of this iterative procedure. *CMLTFC.db* on the third row is the solved FE model with the geometry, loads and solutions all saved. *CMLTFC.** (*CMLTFC.log*, *CMLTFC.err*, etc.) on the fourth row include the standard files created by ANSYS for each job executed.

2.2   Input files

2.2.1   *TFCrun.dat*

A sample file of TFCrun.dat is shown in Fig. 3. This file defines the TFC problem to be solved. The meanings of parameters defined in this file are explained as follows.

- *db file name*: this line defines the name of the .db file that contains the 3-D slider model. The program will load the model according to the file name defined here. You can use any valid names for your model **other than CMLTFC** since this name is reserved for the output .db file of the this program. **Do not include the file extension (.db) on this line.**

- *Thermal model - model type:* this line defines the type of thermal models to be used to describe the heat transfer on the air bearing surface. Currently, two thermal models are provided in the TFC code.

  - *1*: Heat transfer model proposed by L. Chen in [3] is used.
  - *2*: Heat transfer model proposed by D. Chen in [4] is used.

The difference between these two models is mainly in the definition of the mean free path of air molecules in the air bearing. Please refer to the two references listed above for more details

```
db file name
***********************************
slider

thermal model
***********************************
ModelType
2

ansys model
***********************************
unit    orient
2       1
pad1    pad2
55      57
refTemp(C)
25

heater
***********************************
power(mW)  res(ohms)
100        95

mapping
***********************************
xmin       xmax
0.75       1
ymin       ymax
0.33       0.67
nx         ny
201        201

constraints
***********************************
backfixed
0

loads
***********************************
pres
0

cooling
***********************************
back     side        (W/(m.m.k))
2000     100


convergence
***********************************
NFH(nm)     Max. num. of Iteration
0.05        10
```

Fig.3 Sample TFCrun.dat

- *ANSYS model - unit:*
  - *1*: MKS units are used
  - *2*: μMKSV units are used

  Refer to ANSYS Coupled-Field Guide to see the specific units used in each unit system.

- *ANSYS model - orient*: this parameter defines the orientation of the model. **Currently, it should always be set to *1*.**

- *ANSYS model - pad1, pad2*: area number of the heater pads. VOLT constraints are applied on these two areas using the ANSYS command *DA*.

- *ANSYS model - refTemp*: the reference temperature of materials used in the thermal strain calculations in ANSYS. To be consistent with *run.dat*, the unit is set to be ^{o}C.

- *Heater - power*: power applied to the heater in mW.

- *Heater - res*: heater resistance in ohm. The voltage is calculated using $(res \cdot power)^{0.5}$ and applied on *pad1*. Accordingly, *0* is applied on *pad 2*.

- *Mapping - xmin, xmax, ymin, ymax*: these four parameters define an area on the air bearing surface on which you apply the deformation calculated from ANSYS to modify the shape of the ABS. Using these parameters, you are allowed to modify a portion of the ABS where the protrusion profile caused by heating has significant effects on the slider's flying-attitude. The following example shows how to use this set of parameters.

  For a *0.85mm×0.7mm* slider, suppose the desired mapping area is *0.2mm×0.3mm,* and it is located at the trailing edge center as shown in Fig. 4, Then, *xmin=(0.85-0.2)/0.85=0.76*, which is the x-coordinate of the left edge of this area normalized by the slider's length. Similarily, *xmax=0.85/0.85=1*, which is the x-coordinate of the right edge of this area normalized by the slider's length; *ymin=0.2/0.7=0.29*, which is the y-coordinate of the inner edge of this area normalized by the slider's width; *ymax=(0.2+0.3)/0.7=0.71*, which is the y-coordinate of the outer edge of this area normalized by the slider's width.
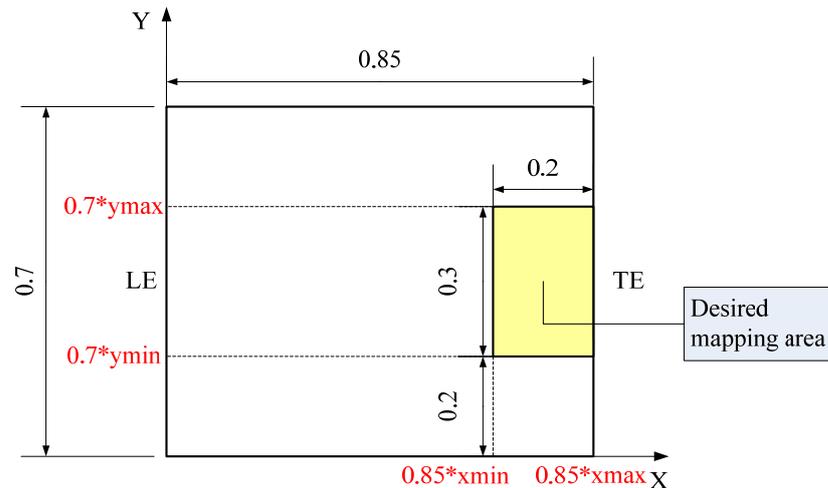
Fig. 4 Definition of *xmin, xmax, ymin,* and *ymax*

- *Mapping - nx, ny*: these two parameters define the density of the grid in the mapping area. *nx* is the total number of grid points in the mapping area along the x-axis, *ny* is the total number of grid points in the mapping area along the y-axis. Once *nx* and *ny* are defined, a uniform rectangular grid will be imposed onto the mapping area. The solved displacements *UZ* on the nodes in the FE model are linearly interpolated to obtain the displacements on this uniform rectangular grid, as shown in Fig. 5.
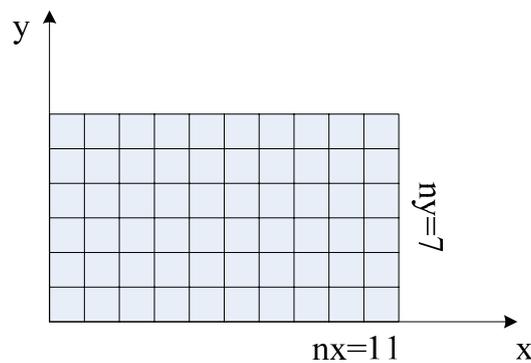
Fig. 5 Uniform grid to map UZ to

- *Constraints - backfixed*: this parameters defines whether or not to apply the back-fixed structural constraints for the structural analysis in ANSYS.

  - *0*: No structural constraints will be applied by the program. In this case, you need to define your own structural constraints in the **.db*

file. The program will use this set of constraints for structural analysis.

- **1:** A set of structural constraints as shown in Fig. 6 are to be applied on the FE model for structural analysis. The two corners at the leading edge on the back of the slider will be fixed. The back of the slider will be constrained in the height direction. The deformation added to the ABS will be the original displacement at each grid point minus the minimum displacement in the mapping area. This option is suitable for models without any structural constraints.
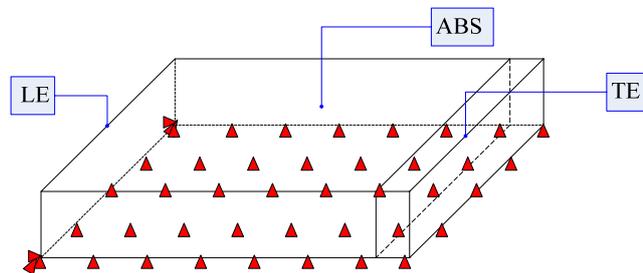
Fig. 6 Back-fixed constraints

- **Loads – pres:** determines whether or not to apply the air bearing pressure as pressure loads in the FE model.
  - **0:** No pressure loads are applied.
  - **1:** The air bearing pressure calculated by the quick4 solver will be applied to the air bearing surface in the FE model as pressure loads. In this case, the deformation caused by the air bearing pressure is accounted for in the structural analysis.
- **Cooling - back:** a uniform convection coefficient with the value defined here will be applied at the back of the slider in the FE model.
- **Cooling – side:** a uniform convection coefficient with the value defined here will be applied at the four sides of the slider in the FE model.
- **Convergence – NFH:** When the nominal FH calculated from two iterations has a difference less than or equal to this parameter, the program will have converged and will exit.
- **Convergence – Max Num. of Iteration:** When the total number of

iterations is greater than this value, the program will exit regardless of convergence.

2.2.2 *rail.dat*

This file contains information of the air bearing geometry. Refer to the *CMLair User's Manual* for a detailed description of this file. *Rail.dat* can be directly generated from the *CMLair* user interface. **The overall size (length and width) of the air bearing design should match with the size of the FE model built in ANSYS.**

2.2.3 *run.dat*

This file contains information on the suspension load, initial flying-attitudes, radii, etc. Refer to *CMLair User's Manual* for a detailed description of this file. Although run.dat allows multiple radii, altitudes and RPMs, **only the last calculated flying-attitude will be used for the TFC simulation**. The temperature and pressure defined in *run.dat* are used as ambient temperature and pressure in the TFC simulation. *Run.dat* can be directly generated from *CMLair* user interface.

In most cases, *rail.dat* and *run.dat* contain all information necessary for the quick4 solver. However, extra input files may be needed under certain circumstances. For example, if the *ioldg* option is set to be 1, *x.dat* and *y.dat* will also be needed as input files. **Since the CML TFC code incorporates the quick4 solver within, make sure you have all requested files by quick4 as input files.**

2.2.4 **.db*

This is a database file created in ANSYS defining the slider's geometry. Do not name it as **CMLTFC.db** so that it won't be overwritten by the output files of this program. Certain requirements should be satisfied for the CML TFC code to be compatible.

(1) The model contains a block-shaped slider. **The overall size of the model (length and width) should match with the size of the air bearing design.**

(2) The material properties are well defined for the steady-state electro-thermal-structural analysis.

(3) The model is oriented as shown in Fig. 7 **in the global Cartesian coordinate system**, i.e., the ABS is oriented upward along the z-axis, and, the slider length

direction should be arranged along the x-axis, with the x-coordinate increasing from the leading edge to the trailing edge.
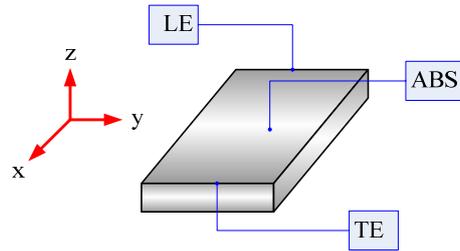
Fig. 7 Required orientation of the FE model

(4) The model should be meshed. A 3-step sequential coupled-field analysis is executed (refer to Fig. 8) in ANSYS, hence, the degrees of freedom (DOF) of every element type are modified by calling **KEYOPT** at the beginning of each stage. The allowed element types include: (i) coupled-field elements: SOLID5, SOLID98; (ii) contact elements: TARGE170, CONTA173, CONTA174, CONTA175. The element types listed above are dealt with according to their definitions of key options.

For most air bearing designs, the heat transfer coefficient on the ABS may vary sharply in a small area, especially at the trailing edge. A properly meshed model is essential for a successful solution in ANSYS.

2.3 Program files

2.3.1 *SolFHMag.mac*

This macro serves as the main program for the CML TFC code. It first exits the current ANSYS processor, and starts a new job titled CMLTFC. This macro iteratively calls *quick.exe* and the ANSYS coupled-field analysis defined in *SolDeformMag.mac*. It also decides when to exit the solving process. The flowchart of this macro is shown in Fig. 8.
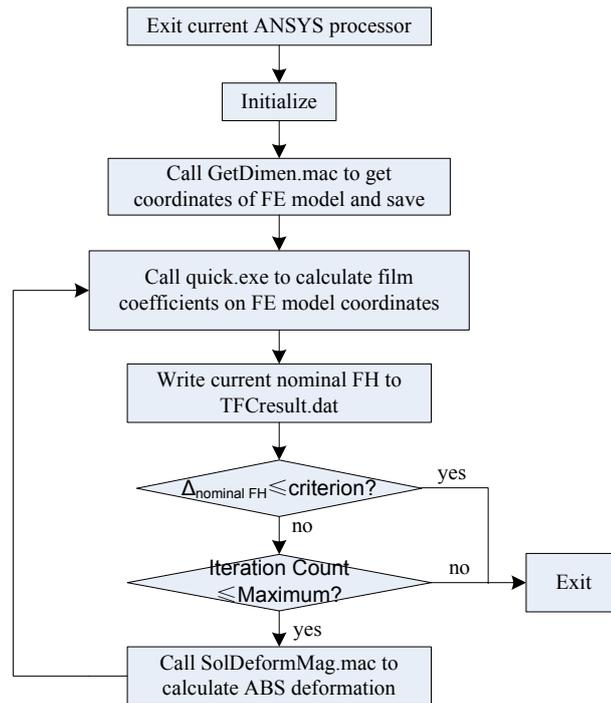
Fig. 8 Flowchart for *SolFHMag.mac*

2.3.2　quick.exe

This compiled executable file solves for the stead-state flying attitude of a slider with a certain ABS design. It is essentially the quick4 solver of CMLair. Besides the standard output files of the quick4 solver, *quick.exe* also creates a formatted file containing the film coefficients across the air bearing surface, which is ready to be imported and loaded into the FE model.

2.3.3　*SolDeormMag.mac*

This macro defines a coupled-field analysis in ANSYS using the sequential method. Basically, it starts by applying voltages across the heater pads and calculates the Joule heating, then it carries out a thermal analysis to obtain the temperature over the slider body, and it ends up with a structural analysis for the deformation of the slider body. It also does a linear interpolation to convert the solved *UZ* on the finite element mesh onto a quick-solver-accepted rectangular grid. The details of this macro and essential commands used to apply the loads and constraints are briefly described in Fig. 9.

```
                    ┌─────────────┐
                    │  Initialize │
                    └─────────────┘
                           │
                           ▼
               ┌───────────────────────┐
               │   Modify DOF of       │
               │   elements  as VOLT   │
               └───────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Use DA command to define VOLT │
           │   constraints on heater pads  │
           └───────────────────────────────┘
                           │
                           ▼
                    ┌─────────────┐
                    │    solve    │
                    └─────────────┘
                           │
                           ▼
               ┌───────────────────────┐
               │   Modify DOF of       │
               │   elements as TEMP    │
               └───────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Read the heat generation results from last │
           │            analysis           │
           └───────────────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │  Use SF command to apply cooling │
           │  coefficients on the ABS and the other 5 │
           │     sides of the slider body  │
           └───────────────────────────────┘
                           │
                           ▼
                    ┌─────────────┐
                    │    solve    │
                    └─────────────┘
                           │
                           ▼
               ┌───────────────────────┐
               │   Modify DOF of       │
               │   elements as UX, UY, │
               │           UZ          │
               └───────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Read the temperature results from last │
           │            analysis           │
           └───────────────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Use SF command to apply pressure loads │
           │      on the ABS if required   │
           └───────────────────────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Apply structural constraints if required │
           └───────────────────────────────┘
                           │
                           ▼
                    ┌─────────────┐
                    │    solve    │
                    └─────────────┘
                           │
                           ▼
           ┌───────────────────────────────┐
           │ Use linear interpolation to obtain UZ on │
           │ the specified rectangular grid and save as │
           │           Usergeom01.dat      │
           └───────────────────────────────┘
                           │
                           ▼
               ┌───────────────────────┐
               │  Save as Deformed.db  │
               └───────────────────────┘
```

Fig. 9 Flowchart of *SolDeformMag.mac*

2.3.4   *GetDimen.mac*

This macro is called by *SolFHMag.mac* at the beginning of the program to obtain the coordinate definitions in the FE model so that *quick.exe* is able to produce the film coefficients on consistent coordinates. You don't need to call this macro directly.

2.3.5 *ModRefTemp.mac*

This macro is called by *SolDeformMag.mac* at the beginning of the coupled-field analysis to modify the reference temperature used to calculate the thermal strains. It is to make sure the reference temperature of each material is consistent with the value defined in *TFCrun.dat*. You don't need to call this macro directly.

2.3.6 *SwitchElemKoptEh.mac*

This macro is called by *SolDeformMag.mac* at the beginning of each stage of the coupled-field analysis. It is to make sure the elements have the right DOFs at each stage of the analysis.

2.4 Output files

2.4.1 Output files from *quick.exe*

quick.exe produces all of the standard output files of the CMLair quick4 solver, such as *result.dat, press01.dat,* etc. For information contained in these files, refer to the user's manual for *CMLair*. In addition, the file named *FilmCoefficient01.dat* tabulates the air bearing cooling coefficients applied into the FE model. Fig. 10 gives a sample of this file. The first row of *FilmCoefficient01.dat* gives the coordinates in the FE model along the slider's length direction, and the first column lists the coordinates in the FE model along the slider's width direction.

Coordinate in length
direction in the FE model

```
0.000000000000000E+00  0.200000000285042E-04  0.192752000000003E+01
-0.289999999925419E-04  0.101150169248552E+04  0.101150813744200E+04
 0.126809710000000E+02  0.684652214277668E+04  0.684685249674165E+04
 0.224108908090903E+02  0.104783937337930E+05  0.104802323729805E+05
 0.299549710000000E+02  0.104787920909275E+05  0.104818171918852E+05
 0.331319710000000E+02  0.104789562901922E+05  0.104826433469147E+05
 0.348919710000000E+02  0.104790463455936E+05  0.104831328051322E+05
 0.370199710000000E+02  0.104791543661216E+05  0.104837407591103E+05
 0.394989710000000E+02  0.104792790103458E+05  0.104843913693569E+05
 0.423139710000000E+02  0.104794189910598E+05  0.104851729197280E+05
 0.454449710000000E+02  0.104795727394404E+05  0.339674650500113E+05
```

Coordinate in width
direction in the FE model

Film coefficients

Fig. 10 Sample *FilmCoefficient01.dat*

2.4.2 *CMLTFC.db*

This file is the originally input **.db* file with all loads applied and solution saved. Therefore, you can obtain all post-processing results provided by ANSYS through this file, including the displacements, temperature, stress, heat flux, etc.

2.4.3 *TFCresult.dat*

This file contains information regarding the TFC iterations. **It is updated at the end of each TFC iteration.** You can access this file throughout the running process to learn the progress of the solution. The file is comprised of two parts: information of the converging process and a summary of results, as shown in Fig. 11. The first part lists the nominal FH calculated from iterations that are finished. The second part contains information about the total number of iterations, the difference in the nominal FH of the last two iterations, whether the solution is considered to be converged and the UZ at the reference point if you set the *Constraints – backfixed* option to be *1*. Here, if the solution converges in the specified maximum iteration number, a message of *solution converged* will be written in this file. Otherwise, a message of *solution not converged* will be given in this file.

```
This file contains information about current TFC simulation.          Nominal FH from
                                                                      each iteration
Nominal flying-height of iteration        1  is    18.81893 nm

Nominal flying-height of iteration        2  is    18.65077 nm

Nominal flying-height of iteration        3  is    18.65514 nm

Total number of iterations:        3
Difference in Nominal FH(nm):    0.00437
Solution converged!
uz at reference point(nm):     0.00000                    Result summary
```

Fig. 11 Sample *TFCresult.dat*

2.4.4 *CMLTFC.**

CMLTFC is used as the job name for the TFC analysis in ANSYS. Thus ANSYS creates a series of files titled *CMLTFC* containing the essential running information. For example, you may want to check *CMLTFC.err* for any warning and error messages generated during the TFC analysis in ANSYS. Refer to ANSYS help for details of these files.

2.4.5 *Usergeom01.dat*

This file contains information about the deformation at the mapping area specified in *TFCrun.dat*. It is of the same format as *User Defined Geometry* defined in *CMLair*. Refer to *CMLair User Manual* for the format of this file.

3. Installation and Setup

The CMLTFC code is written in APDL (ANSYS Parametric Design Language). It is a series of commands packaged in macros which can be directly called from ANSYS. The execution of CMLTFC code requires no more than executing a macro in ANSYS. The following steps should be taken for setting up a TFC analysis:

(1) Copy all program files to the current working directory of your ANSYS. A complete set of program files includes: *SolFHMag.mac*, *quick.exe*, *SolDeformMag.mac, GetDimen.mac, ModRefTemp.mac, SwitchElemKoptEh.mac.* The functions of these files and the relationships between them were introduced in section 2.3**.**

(2) Copy all input files to the current working directory of your ANSYS. Necessary input files are stated in section 2.2.

(3) Modify *TFCrun.dat* to define the parameters for the TFC analysis. The meanings of all parameters defined in *TFCrun.dat* are stated in section 2.2.1. While modifying the parameters, make sure the format of this file is well maintained so that the program can read this file without difficulties.

(4) Launch your ANSYS to the current ANSYS working directory and run the main program, *SolFHMag.mac*. GUI (Graphic User Interface) operations and commands are provided in ANSYS for running a macro. You can use either of the methods listed below to start a TFC analysis:

(i) Using command lines: type

*USE, SolFHMag.mac

in the single line input window provided by the ANSYS User Interface.

(ii) Using GUI operations: Go to the utility menu and choose *Macro->Execute macro*. In the edit box for *name of macro to be executed*, type *SolFHMag* and then click *OK*.

(5) All output files are exported to the current ANSYS working directory. Refer to section 2.4 for an introduction to the output files. Save these files properly before running a new analysis in the same folder.

4. Example TFC analysis

This section contains an example of a TFC analysis with the FE model and ABS design provided in the package. The FE model named MyModel.db only contains the substrate, overcoat and a small block-shaped heater for simplicity. Yet it is effective for illustrating the usage of the CML TFC code.

4.1 Running the example

You can run a TFC analysis on the provided design by copying the program files and input files to your ANSYS working directory as shown in section 3. You can keep the settings of the provided *TFCrun.dat* unchanged. In the example *TFCrun.dat,*

- The field *db file name* is set to be *MyModel* which is the name for the provided FE model.

- The field *unit* is set to be *2*, which means μMKSV units are being used.

- Fields *pad1* and *pad2* are set to be *16* and *15*, which means constant voltage constraints will be applied on area *16* and *15*.

- The field *RefTemp* is set to be *25*, which is the same as the ambient temperature setting in the *run.dat*. The thermal strain will be calculated with 25^oC as the reference temperature.

- The field *power* and *res* are set to be 5 and *5,* respectively, which means a voltage of *0.158v* will be applied on *pad1* and *0v* will be applied on *pad2*.

- The mapping parameters set the area for the mapped protrusion to be the last *1/4* in the ABS length and the middle *1/3* in the ABS width. The density of grid points will be *201* by *201*.

- The field *backfixed* is set to be *1* so that the structural constraints in Fig. 6 are applied. The slider will have no global bending in this case.

- The field *pres* is set to be 0, such that the pressure-induced deformation on the air bearing surface is not considered.

- The fields *cooling-back* and *cooling-side* are set to be *100,* which means a

uniform convection coefficient of *100W/(k·m²)* will be applied on the other five sides of the slider.

- The field *convergence-NFH* is set to be *0.01* so that a difference of less than *0.01nm* in the nominal FH will be considered as a sign of convergence.

- The field *Max. num. of Iteration* is set to be *10*, which means the program will stop when the total of iterations reaches *10*, regardless of convergence.

4.2 Convergence of the solution

The CML TFC code is tested here with the provided slider design. The solution converges in four cycles. Fig. 12 shows the protrusion profiles from the 2nd and 3id iterations Fig. 13 is the generated *TFCresult.dat*, which shows the convergence of the nominal FH.
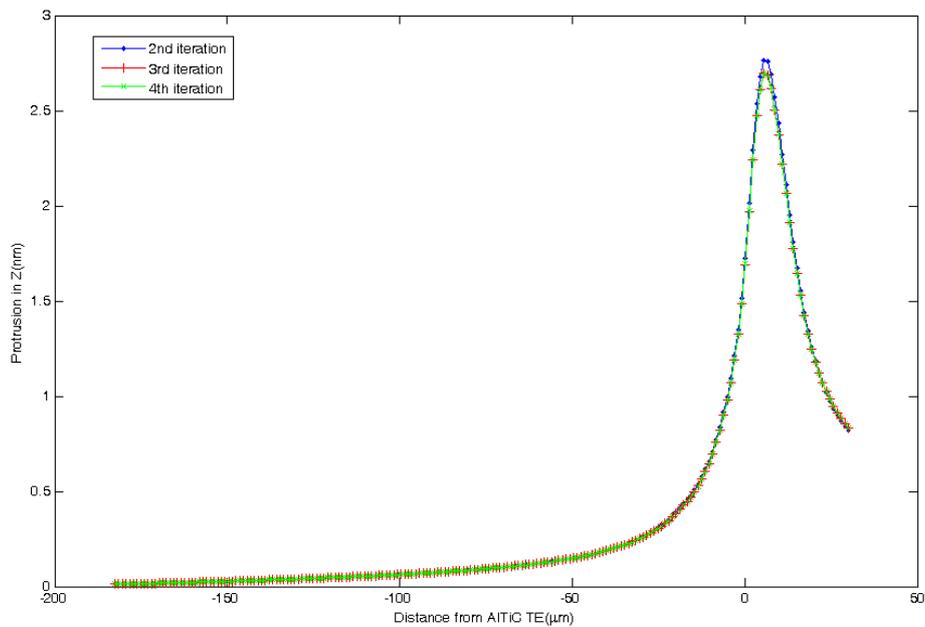
Fig. 12 Convergence of protrusion profile

```
This file contains information about current TFC simulation.

Nominal flying-height of iteration     1  is    6.87179 nm

Nominal flying-height of iteration     2  is    7.51617 nm

Nominal flying-height of iteration     3  is    7.49745 nm

Nominal flying-height of iteration     4  is    7.49764 nm

Total number of iterations:        4
Difference in Nominal FH(nm):    0.00019
Solution converged!
uz at reference point(nm):     0.11454
```

Fig. 13 *TFCresult.dat* of the example problem

4.3 Plotting the ABS cooling coefficient with *FilmCoefficient01.dat*

You can use the generated file *FilmCoeffcient01.dat* to plot the applied film coefficient on the air bearing surface (Fig. 14).
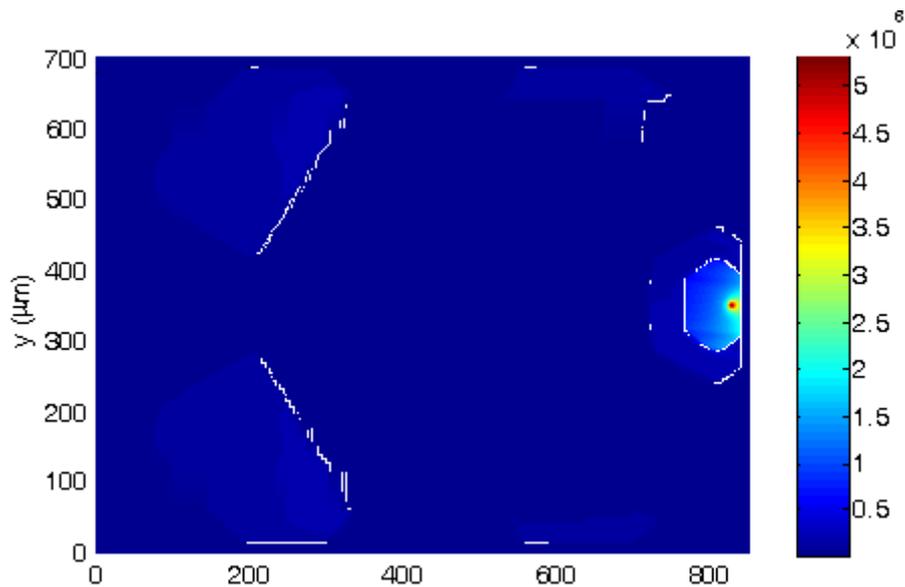
Fig. 14 Display using *Filmcoefficient.txt*

4.4 Plotting the protrusion profile with *Usergeom01.dat*

You can directly use the provided file *Usergeom01.dat* to display the deformation around the transducer (as shown in Fig. 15).
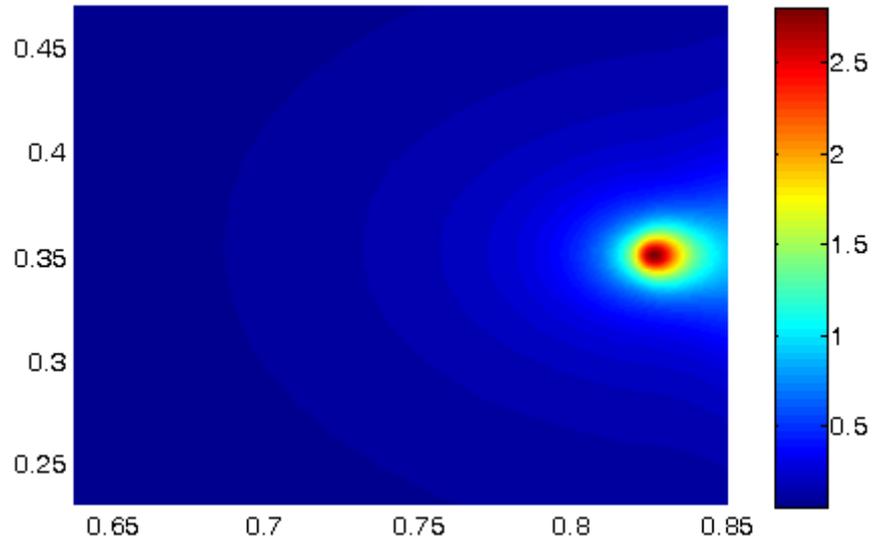
Fig. 15 Plot deformation with *Usergeom01.dat*

4.5 Displaying various results with *CMLTFC.db*

Since ANSYS has a large set of post processing functions, it may be more convenient to view and analyze the results in ANSYS. Figs. 16 and 17 show the body temperature and displacement *UZ* over the slider body created by ANSYS using *CMLTFC.db*.
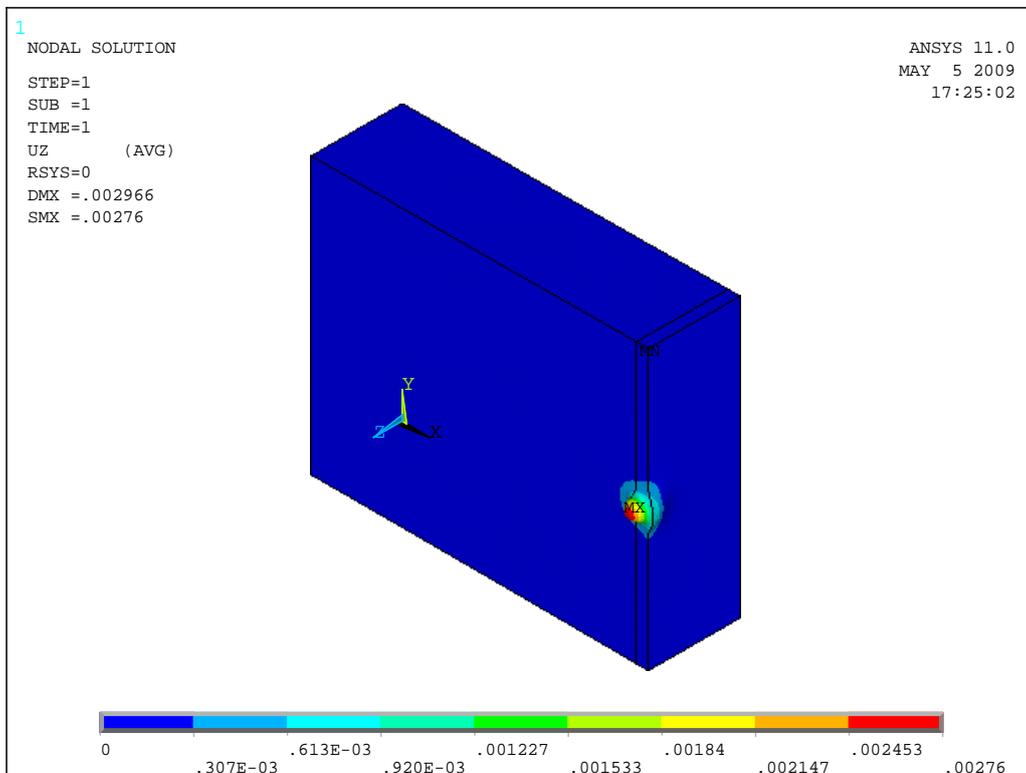
Fig. 16 UZ plot in ANSYS
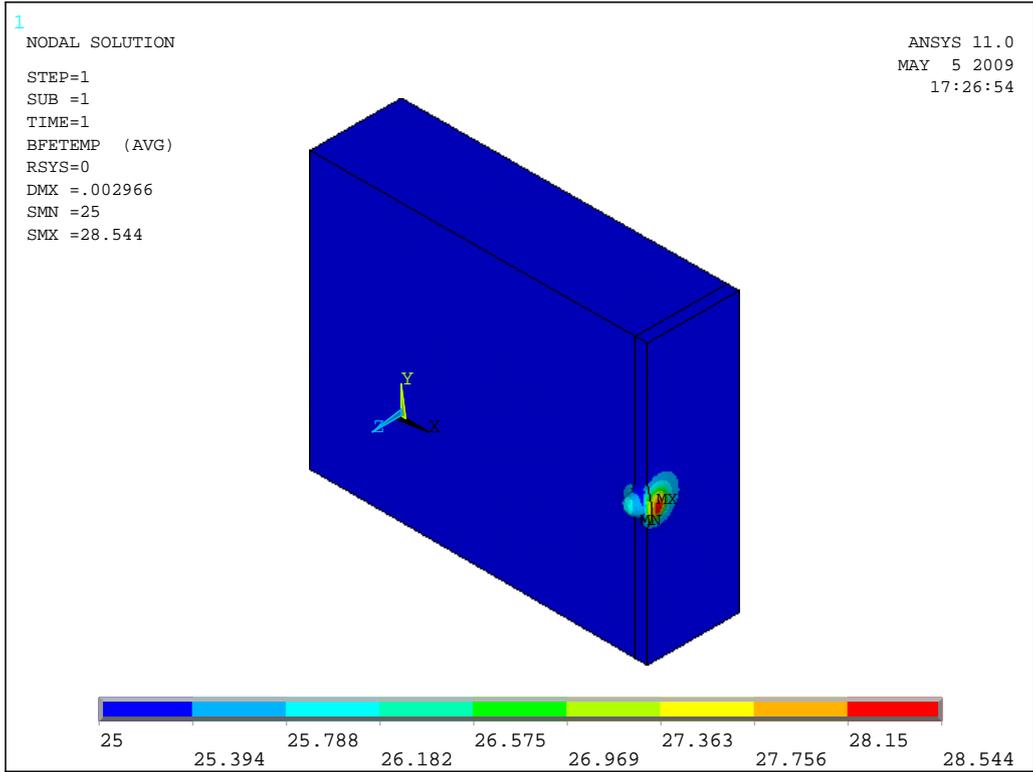
Fig. 17 Temperature distribution plotted by ANSYS

Reference

[1]   J. Juang, et. al., Alternate air bearing slider design areal density of 1 Tb/in2, *IEEE Trans. Magn.*, vol. 42, no.2, pp. 241-246, Feb. 2006

[2]   B. J. Cox, D. B. Bogy, The CML Air Bearing Design Program (CMLAir), Version 7 User Manual, Dec 2007

[3]   L. Chen, et. al., thermal dependence of MR signal on slider flying state, *IEEE Trans. Magn.*, vol. 36, no.2, pp. 2486-2489, Sep. 2000

[4]   D. Chen, N. Liu, D. B. Bogy, A Phenomenological Heat Transfer Model for the Molecular Gas Lubrication System in Hard Disk Drives, CML report 2008-12.

Acknowledgement