

# Transient Thermal Flow and Thermal Stress Analysis Coupled NASTRAN and SC/Tetra

Qin Yin Fan

Software CRADLE Co., Ltd.

## ABSTRACT

In SAE paper 2004-01-1345, author focused on how to use a steady state temperature result obtained by a CFD analysis to conduct a thermal-stress analysis easily with 3 data transfer methods (*Direct Conversion Method, Surface Mapping Method and Volume Mapping Method*). The advantage and disadvantage of the 3 methods were compared in that paper and a steady state analysis of an engine exhaust manifold was used to show the accuracy, flexibility, efficiency and practicality.

In this paper, how to simplify data transfer in a transient coupling is introduced. In transient couplings, 3 troublesome operations always happen. They are 1) multi-steps coupling, 2) change of element from low to high order, and 3) estimation of mid node temperature loads of high order elements. In this paper, CFD (Computational Fluid Dynamics) software adopted is SC/Tetra [1], which generates *hybrid mesh* consisting of tetrahedron, hexahedron, and prism as well as pyramid elements. Meanwhile, structure analysis software used in the study is NASTRAN, in NASTRAN second order tetrahedron or second order hexahedron must be used independently. So in the typical case, the sequences normally are: generating only tetrahedrons in solid part in CFD mesh, and then at every time step, transferring solid part temperature result of CFD to structure elements, executing thermal conduct analysis for getting mid node temperature loads and at last defining constrain conditions and performing thermal stress analysis. It is of great significance to simplify all these operations by preparing an input file as explained in the following paragraphs. Further more, author shows how to choose a few steps of transient temperature results as the loads of thermal stress analysis from a long list of CFD transient results to estimate the local maximum transient thermal stress. In this paper a T pipe and an exhaust manifold are used to describe simplification of data transfer procedures.

## INTRODUCTION

Transient compressible thermal flow and thermal stress coupling analyses are not only difficult in theory but also has cumbersome executive procedures. Accuracy of transient coupling of thermal flow and thermal stress is

affected by transient thermal flow calculation, which is again always affected by turbulent model and compressibility of the flow. When compressibility is not important, a steady state velocity field can be built at first and then a transient thermal field can be built later based on the previous velocity field. Because time step of a thermal field can always be bigger than time step of a flow field, a separated solution of flow and temperature can be useful for saving much computer CPU time. On the contrary, when compressibility cannot be neglected, clearly it is difficult to find a suitable time step for both flow and temperature in a transient calculation. Meanwhile, initial conditions and boundary conditions that are always not perfectly clear will affect the result greatly. Equally important, that the precedent results will affect subsequent results in the transient progress must be took into consideration, namely, the pressure wave backward effects to the results can not be left out. Although it is not difficult to find that a transient thermal flow induces a maximum temperature difference between some typical pair points and may make a maximum thermal stress there at a certain moment, but accurate answer is not easy to estimate. At last, it is still important to find experimentally a set of suitable boundary conditions and initial conditions to get reasonable result. All these are discussed in the following sections with concrete examples.

To simplify tangled operations, the interface tool of SC/Tetra has been developed that the output from the results of SC/Tetra can be high order tetrahedral mesh with mid node temperature loads. The mid-node loads are interpolated by shape function of the element with corner node temperature. The output data can include geometry as well as loads for the first time step but only loads without geometry data for next time steps. What is more, the data can be output as any format of any structure analysis software besides NASTRAN. The tool, as it turned out, is very accurate and efficient for multi-step transient coupling and can cut many cumbersome procedures of mesh type and temperature loads transformations.

## SOFTWARE

In this investigation, we have used a CFD code, SC/Tetra developed by authors [1]. To solve the thermal stress field, we have used commercial structure system NSTRAN2004 developed by MSC. NASTRAN is capable of solving for temperature fields. These solvers are chosen because they are typical structure analysis software widely known by engineers and are readily available to any engineer, so that the contents introduced here can be made more universal. That is to say the concepts introduced here can be used for coupling between any CFD and any structure applications developed by oneself or of commercial software.

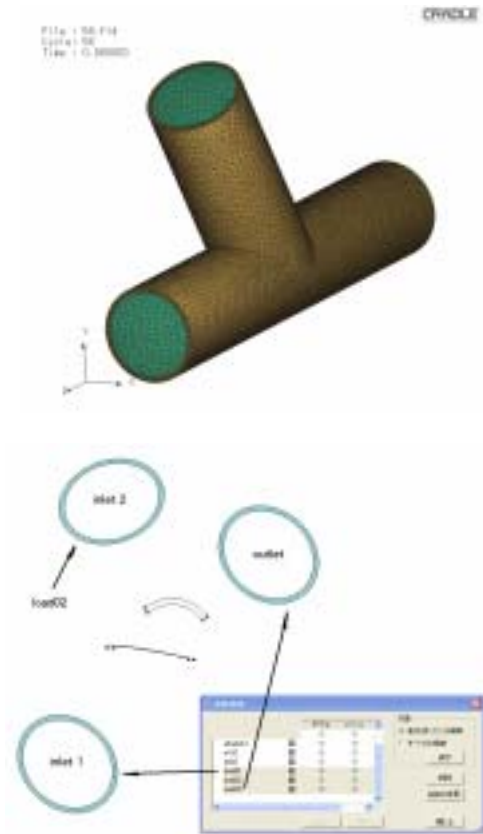
For comparison, tetrahedron mesh is used only in solid domain and hybrid mesh is used only in fluid domain for all calculations of this study. This kind of mesh is easy to create in ST/Tetra and easy to convert to other CAE codes.

Because SC/Tetra is a CFD system based on finite volume method [2] [3] and supports hexahedral, prismatic, pyramidal and tetrahedral elements, i.e. a *hybrid mesh*. Generating a tetrahedral mesh first and then inserting prism layers along the no-slip walls typically construct a computational mesh of SC/Tetra. Prism layers are inserted in order to treat the boundary layer of the flow properly. To avoid the inserting of prism layers along the no-slip walls in the solid side, the resulting mesh typically may contain first order tetrahedral, prismatic hexahedron and pyramidal elements in the fluid domain and only tetrahedron mesh in the solid domain.

SC/Tetra has an interface with other structure systems and its results can be used with minimal effort as input load conditions to them. With low order element and temperature loads at corner nodes from SC/Tetra, in NASTRAN solver, temperature load can be added to every mid node by running a thermal analysis to get the temperatures of the corner nodes automatically. So when NASTRAN is used, only element type change and boundary condition definitions must be made in NASTRAN preprocessor. That means when NASTRAN is used, after element order change, a thermal analysis must be executed to create a temperature distribution for defining mid node temperature loads for subsequent thermal stress analysis. All these are very troublesome, especially for multiple time steps of a transient calculation. This is why the utility tool of SC/Tetra has been developed.

Present version of SC/Tetra tool can output high order tetrahedral mesh with all mid node temperature loads as well as corner nodes from a result file. For NASTRAN the output is just high order elements instead of degenerated high order elements for other commercial structure analysis software. Mid node loads are interpolated by shape function of the elements with corner node temperatures. With the data output by CFD including constraints defined in SC/Tetra preprocessor, thermal stress calculation is started and thermal stress is output. All these are in one step and very simple.

To check the accuracy of the data transformation, comparisons have been made for the following steady state calculations. The tests used a T type pipe and another exhaust manifold model as in the next section. For the first calculation, low tetrahedron meshes and only corner temperature loads are read into structure analysis. And then, in structure analysis software, meshes are changed into high order meshes and the mid node temperature loads are estimated by thermal analysis. For the second one, all transformation and interpolation are executed in SC/Tetra transformation tool. Both tests got the same results with NASTRAN or other commercial structure analysis software for every test model.



**Fig.1 T type pipe model (size in X: -0.2 to 0.2, Y: -0.2 to 0.97 and Z: 0 to 2), mesh (above, 125224 elements and 32433 nodes in all, 35766 elements in solid) and stress constrain boundary conditions (below)**

As we will show, there are some restrictions in the transform tool of SC/Tetra. The first restriction is that constraints defined in SC/Tetra preprocessor for structure analysis must be full fix loads, and the load name must be defined as laod01, laod02, and so on. The second restriction is that the element shape in the solid domain must be low order tetrahedron in fluid thermal analysis. But the elimination of these restrictions is not difficult.

## T-TYPE PIPE CALCULATION

### CALCULATING CONDITIONS AND RESULTS

The geometry, mesh and monitor points of T pipe investigated in this section are shown in figure 1. If one applies a simple boundary condition that consists of a constant ambient temperature and a constant film coefficient on both inner and outer surfaces of the solid part of the pipe, then thermal CFD analysis can be simplified as a thermal analysis. However, if turbulent heat transfer inside the pipe is to be evaluated more accurately, at least the thermal flow field inside the pipe must be solved together with the solid temperature field [4]. Over and above that, a transient flow forms a transient heat transfer coefficient on the solid surface and the value of coefficient varies in time and space. Theoretically to estimate the residual thermal stress, effect of the temperature change in time to final result must be considered and calculations must take huge CPU time for industrial problems. As a consequence, deformation of solid part induced by thermal stress is hypothesized to be small and not affect the fluid, also the effect of the history of temperature change be neglected.

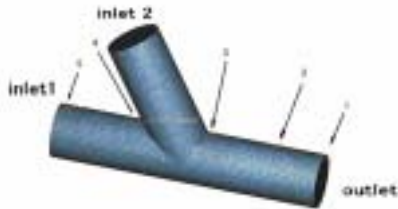


Fig. 2 Position of monitor point pairs of T type pipe model

Point	X (m)	Y (m)	Z (m)
1a	0.000000	0.400000	1.000000
1b	0.000000	0.390000	1.000000
2a	0.000000	0.380000	1.000000
2b	0.000000	0.370000	1.000000
3a	0.000000	0.360000	1.000000
3b	0.000000	0.350000	1.000000
4a	0.000000	0.340000	1.000000
4b	0.010000	0.195000	0.000000
5a	0.000000	0.180000	0.000000
5b	0.000000	0.195000	0.400000

Table 1 Coordinate value of monitor points

For all that, in T pipe example, five monitor point pairs (ten points) are chosen on the symmetrical plane (Fig. 2). In every one of the 5 pairs, one point is on the out surface and another point is on the inner surface of the pipe and their temperature difference is traced. The coordinate values of the ten points are listed in table 1. When the absolute value of the temperature difference of a point pair arrives at a maximum value, the temperature distribution of the solid part at the time is used as

temperature load and a steady state thermal stress distribution is calculated. After all, every point pair needs one stress calculation.

Case	Time Step(s)	End Time(s)	Cycle	CPU time (Hours)	Tmax (p5b)
250	250	50000	200	00.19	571.935
25	25	50000	2000	01.22	571.939
20	20	25000	1249	01.89	570.371
10	10	35000	3500	02.20	571.816
01	1	35000	35000	23.00	571.820
001	0.1	35000	350000	207.752	569.189

Table 2 CPU time and time step size of various cases (monitor point is 5b)

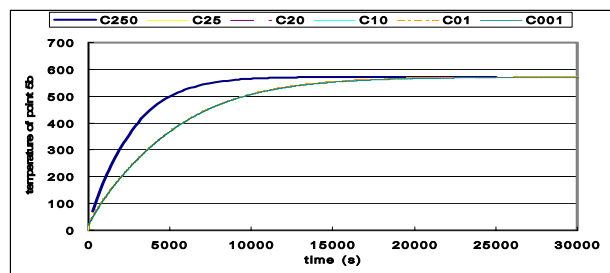


Fig. 3 Temperature variation of point 5b to time of all cases

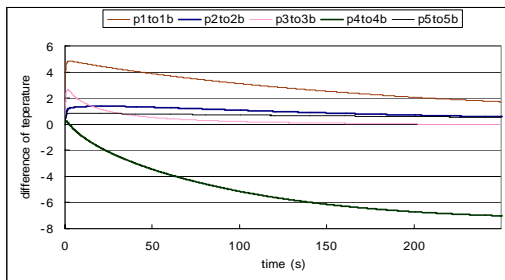
Point pairs	Case01			Case10		
	Cycle	Time(s)	Tmax	Cycle	Time(s)	Tmax
1 and 1b	26	26.0	4.8397	4	40	4.876
2 and 2 b	220	220.0	1.3859	23	230	1.3821
3 and 3b	17	17.0	2.6591	4	40	2.4649
4 and 4b	3220	3220.0	7.1520	325	3250	7.147
5 and 5b	34	34.0	0.8588	6	60	0.8551

Table 3 Time when the temperature difference arrives at the maximum of Case01, Case10 and their value, Note: As an example, point 1a is at the position at the out surface and Point 1b is in the related position at the inner surface in Fig.2 at the position 1, and the temperature difference T is the absolute value of the 2 points

The detail conditions of T pipe problem are as follows. The length and out diameter of the main pipe are 2 m and 0.40 m and those of the branch pipe are 1 m and 0.40 m respectively. The thickness of the pipe is 0.02m. The working fluid is incompressible air and all its properties are treated as constant. The density of air is 1.205 kg/m<sup>3</sup>, viscosity is 1.8135×10<sup>-5</sup> kg/m·s, thermal conductivity is 0.02574 W/m·K, and specific heat is 1004.0 J/kg·K. The material of the pipe is iron. The density of iron is 7871.40 kg/m<sup>3</sup>, Young's modulus is 210×10<sup>9</sup> N/m, Poisson's ratio is 0.3, thermal conductivity is 81.168 W/m·K, the specific heat is 439.2 J/kg·K and expansion coefficient at 293 k is 16.6e-6 1/K.

In figure 2, at inlet 1 near point 5, inlet velocity and temperature are set to 3 m/s and 833 K respectively. In the same way, at inlet 2 they are set to 10 m/s and 1123 K. Pressure is set to 0 Pa at the outlet near point 1. Ambient temperature is set to 293 K and the film coefficient between the outer surface of the pipe and the ambient atmosphere is set to 0.5 W/m<sup>2</sup> ·K. Adiabatic condition is set on all ends of the pipe.

In thermal stress analysis, all nodes on the plane of inlet 1, inlet 2 and those on the plane of outlet are all constrained completely as shown at the bottom in figure 1. They are defined in the SC/Tetra preprocessor with the fluid boundary conditions together. Pressure force exerted on the inner surface of the pipe from the fluid is ignored.



**Fig. 4 Changes to time of the temperature difference of the monitor point pairs of case 01**

Time Step	Time(s)	Displacement	Equivalent Stress
2	0	0 - 0.000746	0.231 - 26.1e7
4	40	0 - 0.000970	0.112 - 33.7e7
6	60	0 - 0.001191	8.746624 - 41.1e7
23	230	0 - 0.00298	0.136 - 102e7
325	3250	0 - 0.018515	2.64 - 607.0e7
Steady state		0 - 0.024093	1.78 - 801.0e7
4 ( NASTRAN )	40	0-0.000832	1.27-28.5e7

**Table 4 Thermal stress and displacement of time step 4, 6, 23 and 325 at time 4 s, 6 s, 23 s and 325 s of case 10 in the above 5 lines of software 1. Steady state results of software 1 for comparison and results at time step 4 of NASTRAN in the last 2 lines**

	Tetra	Pyramid	Prism	Hexa
NASTRAN	○		○	○
ABAQUS	○			○
ANSYS	○	○	○	○
I-DEAS	○		○	○

**Table 5 “○” shows the supported element shape. And NASTRAN, ABAQUAS, ANSYS and I-DEAS are registered trademarks of their owners**

From here, sequences of T pipe solution are described. First, the meshes of both fluid and solid parts are generated in the preprocessor of SC/Tetra. And the boundary and initial conditions of thermal fluid analysis, also the constrain boundary conditions of thermal stress analysis (not including temperature loads) are defined in SC/Tetra preprocessor. Second, CFD system uses all the meshes and only the boundary and initial conditions of thermal fluid analysis to solve a thermal flow field. In fluid analysis, k-ε turbulent model is used to model the effect of turbulence. Third, from the SC/Tetra result files in the solid part, the utility tool of SC/Tetra converts the low tetrahedron meshes into high order meshes and outputs the temperature distribution in the solid part of the pipe as temperature load as well as the not yet used constrain boundary conditions of thermal stress defined in the preprocessor of SC/Tetra. All loads and constrains of mid nodes are included. At last thermal stress field is solved using structure system NASTRAN or other commercial structure analysis software.

Without doubt, time step size of fluid calculation must be pay attention. When velocity field reaches steady state more quickly than temperature field, velocity field can be solved at first and temperature field can be calculated separately late based on the steady state velocity field to spare much CPU time. In the no-coupled thermal analysis, time step can be defined much bigger than ones in the velocity field calculation. To find the possible biggest time step, a very large time step should be used at first and then half of the time step, one forth of it... the suitable time step size will be easily found after the results keep unchanged. As an example, the time steps used for T pipe are listed in Table 2. The first time step is 250 s, and then 25 s, 20 s, 10 s and 1 s. Temperature changes at point 5b of all cases of T pipe are shown in Fig.3. The results of time step 1, 0.1 and 0.001 are on the same curve. That means the time step size variation in the range of 0.1~1 s does not affect the result. The time step size 1 s is found to be suitable after 4 case tests and all the test CPU time is about 5.5 hours. When test begins from a size of 0.1 s, 207 hours must be wasted for only the first case. Here, a very simple but important rule must be remembered, that test must begin from a very big time step size to find a suitable time step size, but not from a small one.

For spare space, only some results of case 01 and 10 are shown in detail. The maximum temperature difference and the time when temperature difference reaches at a maximum value for all five-point pairs are shown in Table 3. The curves of temperature difference variation of all the five point pairs of case01 are plotted in Fig.4. The range of displacement and von Mises stress at five different times when the temperature difference reaches at a maximum value for all the five point pairs of case10 are shown in Table 4. Only contours at time 4 s of NASTRAN (case10) are shown in Fig. 5. The difference of results between NASTRAN and other commercial structure analysis software in Table 4 can be considered the difference of postprocessor and little different accuracy of the solvers. Note the method testing time step size used here in table

2 can be only used for incompressible flow, the reason is explained in detail in the next section.

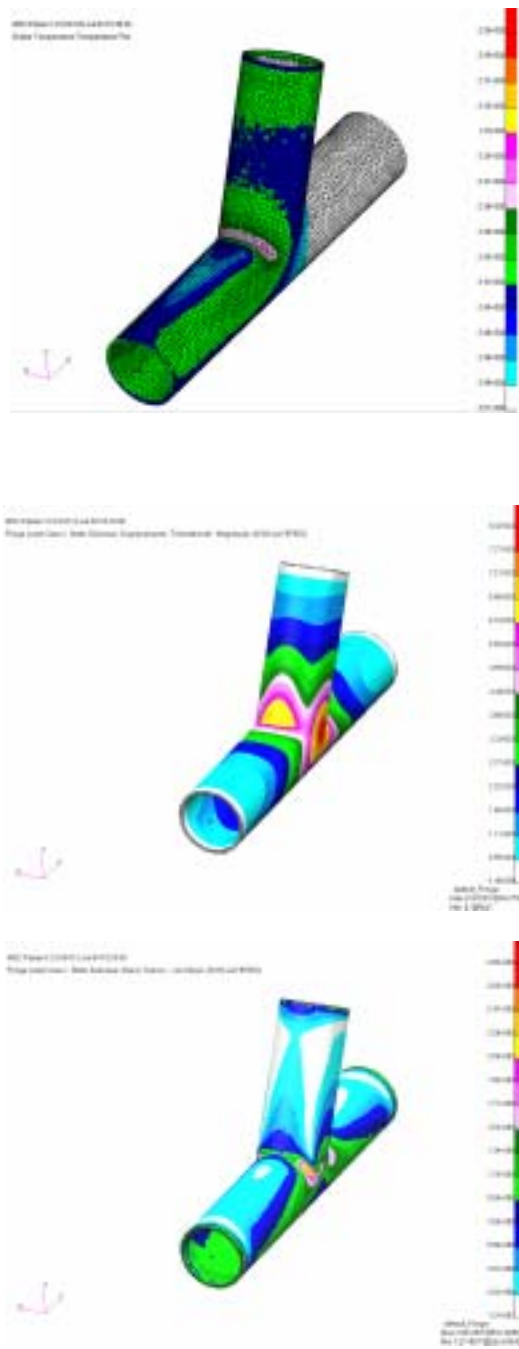


Fig. 5 Temperature loads (first), deformation contour (second), and von Mises stress (third) of case 10 at time step 4 of 40 s of NASTRAN, and postprocessor used PARTRAN

At this point, author wants to go further and claim the close relevancy of the paper to a previous paper of author [3]. In that paper, how to convert the temperature field obtained in a CFD analysis into an input data for thermal stress analysis has been discussed. But mesh type change from low order into high order has not been

mentioned. As complement, the method used above in this section can be considered as a part of **direct conversion method**. The restriction of this method is only that structure system and CFD system must accept the same element in solid part. Preprocessor of SC/Tetra can generates hybrid mesh in the fluid part and just tetrahedron mesh in the solid part at the same time automatically.

In the same paper [3], another 2 methods **Surface Mapping Method** and **Volume Mapping Method** are discussed. They must be enhanced by adding high order element mapping. That is to say, the mid node temperature of high order elements must be interpolated using the corner node temperature in the result of fluid analysis, when high order element is used in the structure analysis. Table 5 can be used for reference to choose the mapping method. Note, it is better to refer the paper [3] to find detail definitions of all the 3 methods, because repeating of these definitions is beyond the scope of this paper.

### PRACTICAL EXHAUST MANIFOLD MODEL

Fluid in last section is considered as incompressible flow, but in practical exhaust manifold and so on even compressibility of subsonic flow affects the result significantly [3]. In this section, compressible thermal fluid of another complicated model is used for investigation.

The model of exhaust manifold is shown in figure 6. It has 507931 elements (285896 in solid) and 126611 nodes. The size of the model is in X: -0.690775 ~ -0.256671 m, in Y: -0.114349 ~ 0.0739997 m and in Z: -2.55461e-010 ~ 0.148652 m.

The calculation conditions of exhaust manifold are as following. The mass flow 0.07236 kg/s and temperature 1223 K are defined at the inlet exit and pressure 10000 Pa is defined at the outlet exit. The ambient temperature is set to 293 K and the film coefficient between the outer surface of the exhaust manifold and ambient atmosphere is set to 13.5 W/m<sup>2</sup> ·K. The compressibility of air is considered defining its gas constant as 287, viscosity as 1.8135×10<sup>-5</sup> kg/m·s, thermal conductivity as 0.02574 W/m·K, and specific heat as 1004.0 J/kg·K where density is induced from ideal gas equation. The density of solid part is 7871.4, kg/m·s, thermal conductivity is 29.3 W/m·K, specific heat is 439.2 J/kg·K and expansion coefficient at 293 k is 16.6e-6 1/K. The constrain loads (below of figure 6) of thermal stress analysis are defined in the SC/Tetra preprocessor and just transferred into thermal stress analysis with the temperature results when fluid analysis is finished.

Because compressible flow is a very complex physical phenomenon, it is always necessary to check the results by experiments carefully. With verifications, we can distinguish the numerical oscillations induced by small time step from the physical vibration and find correct initial and boundary conditions to get a reasonable result.

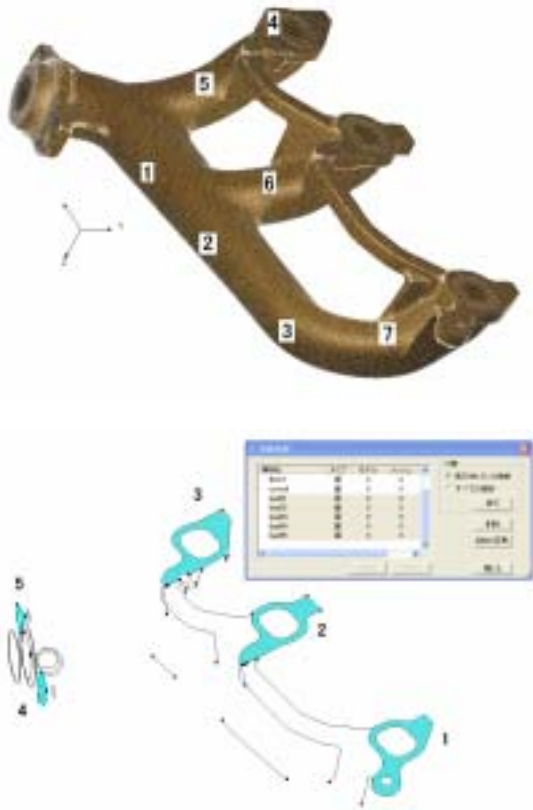


Fig. 6 Model of exhaust, position of its monitor point pairs (above) and constrains of thermal stress analysis (bottom)

Case	LOOP, UNDR	Time step t (s)	Transient Courant Number CL (average / max)
1	Steady state	~	~
2	1, 0	~	1.0 Fixed
2c	50, 0	~	0.5 Fixed
6	50, 0	0.00001	0.065541-0.79815 / 0.76-14.3144
7	50, 0	0.0001	1.263-8.010 / 10.746-151.394
8	50, 0	0.00005	0.54 ~ 4.1 / 4.89 ~ 78.23
9	50, 0	0.000001	0.00476 ~ 0.4091 / 0.08764 ~ 2.594
10	5, 0	0.0001	1.259-8.048 / 10.0-152.0
11	5, 0.7-0.8	0.01	350-549 / 5935-8585
12	5, 0.7-0.8	1.0	37377-58659 / 573415-765968
13	5, 0.7-0.8	50.0	2.1-3.6e6 / 3.4-4.5e6

Table 6 Cases of exhaust manifold. LOOP=n means the maximum loop in one time step is n. UNDR=0 means the under relaxation default value are not changed, and the default values are 0.9 for velocity and pressure, 0.6 for turbulence. And when UNDR=0.7-0.8, the default value of under relaxation is changed into 0.7 for pressure and 0.8 for velocity

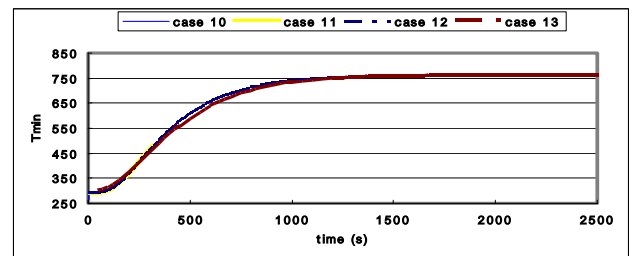
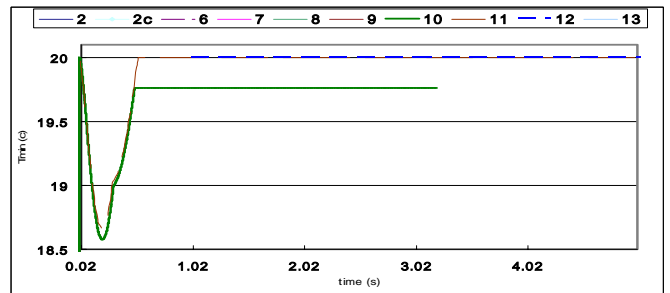
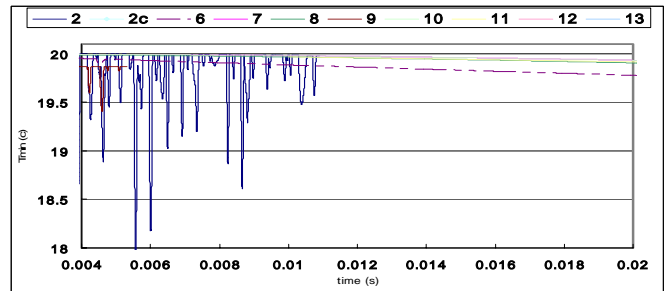
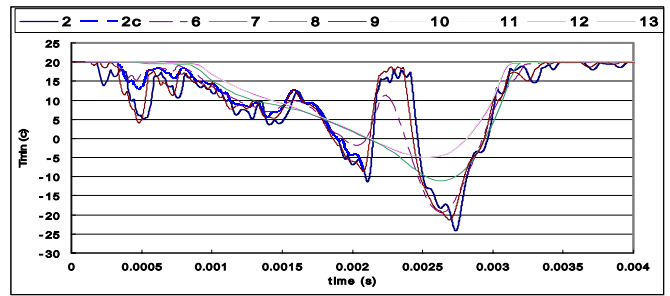


Fig. 7 Minimum temperature in solid part of cases of exhaust in the range of time 0~0.004, 0.004~0.02, 0.2~4.7 and 0~2500 s from top to bottom as (a), (b), (c) and (d)

The executed cases of exhaust in the section are shown in table 6. Fixed time step size or fixed Courant number is used for all these cases. Further a very small time step size is used in case 9. All these cases show that different time step size gives different transient compressible result. In spite of that necessary of verification with experiments cannot be neglected, following several conclusions can be summarized comparing calculated results. At first, a constant size is always better than a constant Courant number, because change of the time step size can affect the calculation as a kind of change of boundary condition or initial condition. This is why in Fig. 7(a) the result of case 2c is close to case 6, but case 2 and 9 has many

smaller waves than case 2c and 6. This can be also explained that the time step size of case 2c (Courant number is fixed in 0.5) changes less than that of case 2 (Courant number is fixed in 1.0). Second, very small time step requires much CPU time and induces pressure oscillations. [5][6] The waves of case 2 in Fig. 7(b) can be considered as oscillations induced by both small time step and big change of time step, and waves of case 9 is induced by just small time step. Third, the air flows into the inlet at 1223 K and the initial temperature of the domain is 293 K. That means there is a shock of temperature at the start time and the shock may induce smooth waves of the curves in Fig. 7(a) and the small smooth vibrations carry on to about 0.004~0.005 s. After that the small smooth vibration disappears in Fig 7(b). At last all the cases have a smooth change to reach at the maximum value of 1023K (750-C in the figures).

Point	X m	Y m	Z m
1a	-4.0e-1	-7.0e-2	9.65e-2
1b	-4.0e-1	-6.8e-2	9.70e-2
2a	-5.0e-1	-6.19e-2	9.52e-2
2b	-5.0e-1	-5.95e-2	9.25e-2
3a	-5.9e-1	-4.76e-2	9.09e-2
3b	-5.9e-1	-4.94e-2	8.53e-2
4a	-3.5e-1	2.37e-2	-0.00000
4b	-3.5e-1	2.19e-2	4.94e-3
5a	-3.6e-1	-9.98e-3	5.00e-2
5b	-3.6e-1	-1.71e-3	4.82e-2
6a	-4.7e-1	-1.00e-2	5.00e-2
6b	-4.7e-1	7.10e-4	4.53e-2
7a	-6.3e-1	-8.45e-3	4.86e-2
7b	-6.3e-1	5.65e-3	3.78e-2

Table 7 Coordinate values of monitor points of exhaust manifold

All the same, the boundary conditions and initial conditions are more difficult to define than in incompressible flow, at least the shock at the very beginning of the calculation discussed above must be verified, because it may not exist really. Similarly, when velocity is defined at the inlet, a shock induced by velocity will affect the result greatly and the shock of the velocity may not exit in real physical phenomenon too.

Turning now in this paragraph to compare temperature variations, in figure 7(a) and (b), the decrease of temperature is less than -30 degrees from 0~0.02 s, the temperature increases to 1023 K (750-C) from 0.02~2500 s, so the temperature difference change after 0.02 s is much bigger than the change at the early time range such as 0~0.02 s, so the vibration until 0.02 s can be neglected and a big time step here 0.01~1 s of case 11 and 12 can be considered as suitable sizes of time step to investigate the effect of transient flow to solid part.

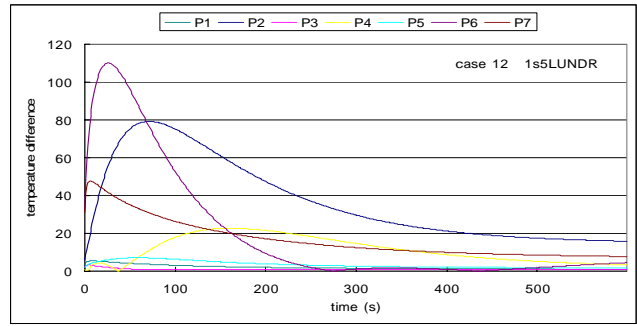


Fig. 8 Change to time of maximum temperature difference of point pairs of exhaust (case 12)

P pair	1	2	3	4	5	6	7
T	5.36	3.49	5.11	47.59	13.55	84.73	25.48
Time(s)	1.65	1.27	45.08	5.72	56.95	1.42	186.42
Cycle	165	127	4508	572	5695	142	18642
T	5.60	79.15	3.27	22.7	7.113	110	47.65
Time(s)	9	73	6	162	59	28	8
Cycle	9	73	6	162	59	28	8
T	3.54	1.85	3.17	37.73	8.99	60.99	21.87
Time(s)	50	50	250	50	50	50	250
Cycle	1	1	5	1	1	1	5

Table 8 Maximum temperature difference of point pairs of case 11, 12 and 13 from top to bottom of exhaust are shown here

Here three time steps of 0.01 s, 1.0 s and 50.0 s are used to investigate variation of maximum temperature difference of point pairs in exhaust. In table 8, the maximum temperature difference, the related time and cycle of the 3 cases are shown and in Fig. 8 the temperature difference variations to time of point pairs of case 12 are plotted. Again for spare space of the paper only the temperature loads, the deformation contour and von Mises stress of exhaust manifold at time 28 s of case 12 are shown in figure 9. At time of 28 s, the temperature difference between monitor point 6a and 6b reaches at the maximum value of 110 k. As in table 8, it is very clear that time steps affect the results. This means suitable time step size, correct boundary conditions and initial conditions are still important for getting an accurate result, even a reasonable procedure and a convenient interface of thermal-flow-stress coupled calculation are prepared.

## DISCUSSION

At last, the most important conclusion is that transient thermal flow induces a maximum thermal stress sometime at any point. In the three cases here, monitor points are chosen experientially. Although, all the maximum values are not bigger than that in a steady state calculation at the same point, under certain conditions, bigger transient

stress must be able to obtain. In table 9, von Mises stress of one point for both models in the paper is listed and the von Mises stress values of steady state and transient calculations are compared. In T pipe the von Mises stress of point 1a and 1b (figure 2) of case 01 (table 2) are almost the same for both of steady state and transient case. For exhaust manifold, point 6a and 6b (figure 6) are compared; the value of von Mises stress of steady state is about 30 times bigger than that of transient calculation of case 12 (figure 9).

Model	x	y	z	von Mises stress	
				S	T
T-pipe	-4.7E-3	0.4	1.0	4.9E+8	4.9E+8
	-5.6E-3	0.39	1.0	5.9E+8	5.9E+8
Exhaust	-0.47	-9.4E-3	0.049	1.3E+9	4.3E+8
	-0.47	0	0.046	1.5E+9	4.9E+8

Table 9 Comparison of von Mises stress of transient and steady state calculations at some points, S means steady state and T means transient

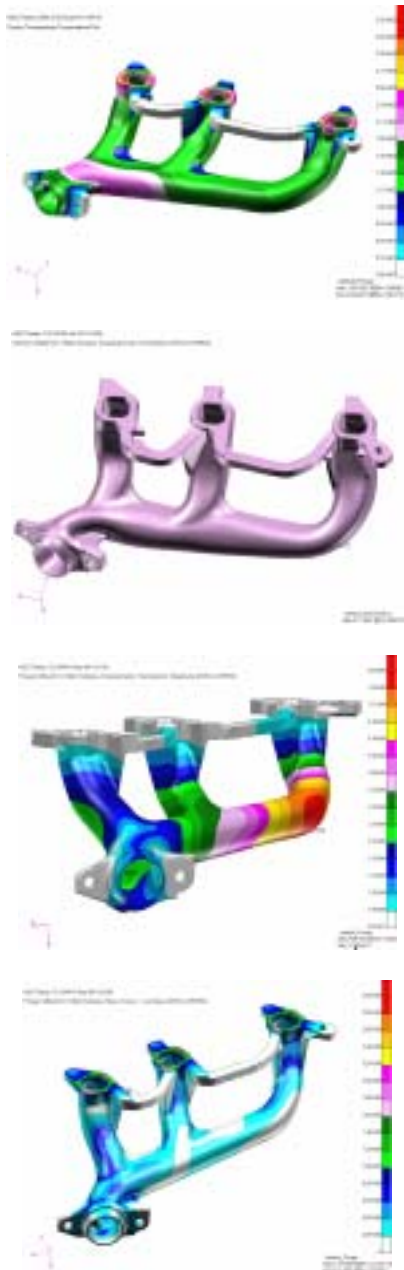


Fig. 9 Temperature loads (first), deformation (second), deformation contour (third) and von Mises stress (forth) of case 12 of exhaust manifold at time step 28 (28 s)

## CONCLUSION

Transient thermal flow and thermal stress coupled analysis is investigated in detail to establish a reasonable method for it. With interpolation outputting high order mesh and mid node temperature loads, complex coupled simulation can be made much easier than normal way step by step, it is found that:

1. When compressibility of the fluid can be neglected, the thermal deformation has little effect to the flow and the effect of the history of the temperature change is not important in the stress calculation, uncoupled method can be used. Transient temperature field can be built on a steady state velocity field to save much CPU time. And when monitored temperature difference of point pairs reaches at a maximum value, the temperature distribution can be used as thermal stress temperature loads to estimate the transient thermal stress at a time like this.
2. Even in a subsonic flow in many practical problems, compressibility of the flow affects greatly the temperature distribution, eventually the thermal stress, such as in an exhaust manifold and all that. [3] When Compressibility must be concerned with, the simulation becomes complex. At first the time step size is difficult to determine, second the fluid and thermal calculations must be coupled. Third, the initial conditions and boundary conditions are always not perfectly clear. Some of them must be measured with experiments. Anyhow, the effect of thermal deformation to the flow and the effect of the temperature change in time to the stress calculation can be considered as small; the method retold in the paper is still an effectual measure to estimate the thermal stress. That is to say, taking temperature as load at the time when the temperature difference of the monitor point pair reaches at a maximum value. So the thermal analysis can be made separately after the thermal-fluid coupled calculation.
3. In all cases of the paper, the biggest thermal stress in a transient process is not bigger than the maximum thermal stress in a steady state one. Nevertheless, under certain conditions, big transient stress can be obtained and may be an important design parameter.
4. Thermal-fluid-stress coupled simulation is limited by computer resource not only now and also in the near future. So how to apply transient residual stress

analysis considering the temperature time trace to industry products is still an important research object. [7][8]

## REFERENCES

1. T. Matsushima, "An automatic mesh generator based CFD system to be used as a design tool", SAE technical paper 2001-01-0037.
2. S. V. Patankar, Numerical heat transfer and fluid flow, Hemisphere Publishing Co., Taylor & Francis Group, New York, 1980.
3. Qinyin Fan, Masayuki Kuba, Junichi Nakanishi, Coupled Analysis of Thermal Flow and Thermal Stress of an Engine Exhaust Manifold, SAE technical paper 2004-01-1345.
4. H. Tennekes and J. L. Lumley, A first course in turbulence, MIT Press, Massachusetts, 1972.
5. Rhie, C. M. and Chow, W. L., Numerical study of the turbulent flow past an airfoil with trailing edge separation. AIAA Journal, Vol. 21, 1983.
6. Wen Zhong Shen, "Improved Rhie-Chow Interpolation for Unsteady Flow Computations", AIAA Journal, Vol.39 NO.12 page 2406.
7. Qinyin FAN, A Research on Coupled Transient Analysis of Thermal Flow and Thermal Stress, SAE technical paper 2005-01-0516
8. Qinyin FAN, Coupling of Transient Analysis of Thermal Flow and Thermal Stress for T pipe, SAE technical paper 2006-01-0589

## CONTACT

Authors can be contacted by the following e-mail address:  
fan@cradle.co.jp