

Comparison of Coupled Analysis of Thermal Flow and Thermal Stress

Qin Yin Fan

Software Cradle Co., Ltd.

Tatsuhito Matsushima

Software Cradle Co., Ltd.

Abstract:

Using the temperature distribution of solid parts calculated by a flow-solid coupled CFD analysis, thermal-stress analyses are conducted. Four approaches are investigated. 1) The mesh and the temperature distribution of FLOTRAN are used in a thermal-stress analysis directly. While this is the simplest and most direct approach, designing a suitable mesh for FLOTRAN is generally difficult for complex models such as an engine exhaust manifold. 2) SCRYU/Tetra, a FVM based CFD software, is used instead of FLOTRAN to solve for the temperature and flow field. SCRYU/Tetra uses a mesh consisting of tetrahedron, hexahedron, prism and pyramid, or a *hybrid mesh*. Because the hybrid mesh is compatible between ANSYS element type 95 and SCRYU/Tetra, not only the temperature distribution of solid part but also the mesh can be re-used for thermal-stress analysis in ANSYS. 3) Flow and temperature field is solved using SCRYU/Tetra and resulting film coefficient between solid and fluid and the ambient flow temperature is interpolated to the corresponding surface of a mesh generated for thermal-stress analysis. The temperature distribution within the solid part is re-calculated and the thermal-stress analysis is done using ANSYS. 4) Flow and temperature field is solved using SCRYU/Tetra and resulting temperature field is interpolated to the nodes of the mesh for thermal-stress analysis. Thermal stress analysis is done using ANSYS.

Above 4 methods are compared in this paper with a simple cross pipe model. Approach 4) is found to be most reasonable in terms of computational flexibility and efficiency. Finally as an application, a complex engine exhaust manifold is analyzed to show the accuracy, flexibility and efficiency of the combined use of SCRYU/Tetra and ANSYS.

Introduction

Rapid advancements in simulation technology and computer hardware have made the CAE a very common tool for engineering. Commercial software with sophisticated GUI has made, for example, structural analysis and computational fluid dynamics (CFD) a well-established procedure. However if the problem spans both stress analysis and CFD, the analysis procedure may be less established. This typically occurs because a stress analysis system and a CFD system tend to be developed independently from each other and much less attention is paid to the interface between them.

To overcome the problem, ANSYS provides a consistent method to treat the structural analysis and flow field analysis using FLOTRAN. However this combination of solvers may not always be the best choice for a given set of problems. For example, the treatment of a very complicated geometry not suited for FLOTRAN may be necessary or a function not equipped with FLOTRAN may be needed.

In this paper, we have investigated different approaches to couple a stress analysis system and a CFD system to analyze a thermal stress field. The key method of the coupling is how to convert field data obtained in the CFD stage to input data for stress analysis. The approaches are fairly straightforward and should be useful for applications in different fields of study.

Software

In our investigation, we have used two different CFD systems: FLOTRAN and SCRYU/Tetra [1]. To solve for the thermal stress field, we have used ANSYS. All three applications are capable of solving for temperature fields. These solvers are chosen because they are readily available to the authors and the authors have considerable experience in the software.

FLOTRAN is a part of ANSYS and gives a consistent procedure for fluid-coupled thermal stress analysis within one system. FLOTRAN supports hexahedral, pyramidal and tetrahedral elements for its computational mesh. However it does not support prism elements. A hexahedral mesh can be generated if one chooses to construct the computational mesh manually. However if the geometry is sufficiently complicated and an automatic mesh generator needs to be used, a tetrahedral mesh becomes the practical choice for FLOTRAN.

SCRYU/Tetra is a CFD system based on finite volume method [2] and is an independent system from ANSYS. It supports hexahedral, prismatic, pyramidal and tetrahedral elements, i.e. a hybrid mesh, for its computational mesh. A computational mesh for SCRYU/Tetra is typically constructed using SCRYU/Tetra preprocessor by generating a tetrahedral mesh first and then inserting prism layers along the no-slip walls. Prism layers are inserted in order to treat the boundary layer of the flow properly. The resulting computational mesh typically contains tetrahedral, prismatic and pyramidal elements. SCRYU/Tetra has an interface with ANSYS and its results can be used with minimal effort as input conditions to ANSYS.

ANSYS supports a wide range of elements. From these, we have used type 45, 72, 92 and 95 elements to solve for the stress fields. Type 45 and 72 element are the first order elements and type 92 and 95 elements are the second order elements. More details of these elements will be discussed later. Sparse direct method is used to solve for the thermal stress field.

As we will show, there are some restrictions in the latest version of the software we have used. These restrictions are restrictive enough to have affected the choice of test cases we have investigated. In future versions of the software, some of the restrictions may be removed and test cases we had to exclude may become possible to evaluate. Thus for the record, we list the versions of the software we have used: FLOTRAN and ANSYS Version 5.7 and SCRYU/Tetra Version 3.

T-type pipe calculation

The test geometry investigated in this section is shown in Fig. 1. The detail of the computational mesh will be discussed later. If one applies a simple boundary condition that consists of a constant ambient temperature and a constant film coefficient on the inner and outer surface of the solid part of the pipe, then temperature field and thermal stress field can be solved directly using ANSYS. In this case no flow field needs to be solved. However, if turbulent heat transfer inside the pipe is to be evaluated more accurately, the flow field inside the pipe must be solved together with the temperature field. A better accuracy is expected if we solve the flow field outside the pipe. However this requires the definition of more specific conditions outside the pipe and will defocus our purpose. Hence we will consider only the flow field inside the pipe.

We define the test problem as follows. The inner diameter of the main pipe is 2.0m and that of the branch pipe is 1.4m. The thickness of the pipe is 0.1m. The working fluid is air and is treated as incompressible. All properties of air are treated as constant. The density is 1.2050 kg/m^3 , viscosity is $1.8135\text{e-}05 \text{ kg/m}\cdot\text{s}$, thermal conductivity is $0.02574 \text{ W/m}\cdot\text{K}$, and specific heat is $1004.0 \text{ J/Kg}\cdot\text{K}$. The material of the pipe is iron. The density is 7871.40 kg/m^3 , Young's modulus is $210\text{e}9 \text{ N/m}$, Poisson's ratio is 0.3, thermal conductivity is $81.168 \text{ W/m}\cdot\text{K}$ and the specific heat is $439.2 \text{ J/Kg}\cdot\text{K}$.

At inlet 1 in Fig. 1, Inlet velocity and temperature are set to 0.7 m/s and 773 K, respectively. In the same way, at inlet 2 they are set to 0.4 m/s and 573 K. Pressure is set to 0 Pa at the outlet. The ambient temperature is set to 546K and the film coefficient between the outer surface of the

pipe and the ambient temperature is set to $5 \text{ W/m}^2 \cdot \text{K}$. Adiabatic condition is set on each end of the pipe.

In thermal stress analysis, all nodes on the plane of inlet 1 and those on the plane of outlet are constrained in the normal direction. Nodes on the plane of inlet 2 are constrained completely. Pressure force exerted on the inner surface of the pipe from the fluid is ignored.

The test problem can be solved as follows. First, the flow and temperature field is solved using either FLOTRAN or SCRYU/Tetra. The $k-\epsilon$ model is used to model the effect of turbulence. Second, using the temperature distribution in the solid part of the pipe, thermal stress field is solved using ANSYS.

One important issue here is how to convert the temperature field obtained in the CFD analysis to the input data for thermal stress analysis. The first consideration to be made is whether the same mesh should be used in both analyses.

If the same mesh is used, the temperature values at the nodes can be used directly as the input data for thermal stress analysis. No interpolation of data is necessary because FLOTRAN and SCRYU/Tetra store all field values at the nodes of the mesh and ANSYS requires nodal values as its input temperature data. We call this method as direct conversion method.

One restriction in using this method is that FLOTRAN and ANSYS do not accept all element shapes in the input mesh as shown in Table 1. For example, element type 45 of ANSYS does not support pyramid elements. Element type 72 and 92 support only tetrahedral elements. Thus if the mesh used in CFD analysis contains pyramid elements in the solid part, the element type 45 cannot be used and the use of element type 95 must be considered. Here, note that if pyramid elements are contained *only* in the fluid part of the mesh, the element type 45 can still be used because these elements can be simply deleted or inactivated in the thermal stress analysis.

Table 1. Supported element shapes.

| | FLOTRAN Type 142 | SCRYU/Tetra | ANSYS Element Type | | | |
|-------------|---------------------|-------------|--------------------|----|----|----|
| | | | 45 | 95 | 72 | 92 |
| Tetrahedron | O | O | O | O | O | O |
| Pyramid | O | O | X | O | X | X |
| Prism | X | O | O | O | | |
| Hexahedron | O | O | O | O | | |

If a different mesh is used in the thermal stress analysis, the temperature values at the nodes of the mesh be interpolated to the nodes of the mesh for thermal stress analysis. Thus some introduction of error is expected in the interpolation process. On the other hand, because the mesh can be different for CFD and stress analysis, a suitable mesh can be chosen for each analysis stage without affecting the other stage.

In the case a different mesh is used, two interpolation methods can be considered to analyze the thermal stress field. The first method is to interpolate the turbulent film coefficient and corresponding ambient fluid temperature on the inner surface of the pipe. The turbulent film coefficient can be evaluated from the temperature law of the wall [3] and the ambient fluid temperature can be taken as the temperature at the node one element thickness inside from the surface. Then together with boundary conditions on other surfaces of the pipe, the temperature field of the solid part can be recalculated using ANSYS. Using resulting temperature field, the thermal stress field is analyzed. We call this method as surface mapping method. Element type 70 is used in our study to solve for the temperature field in ANSYS.

One advantage of surface mapping method is that the temperature field in the solid part of the pipe does not necessarily need to be solved in the CFD stage because the turbulent film coefficient and the ambient fluid temperature can be computed only from the fluid part of the solution. Hence, it is possible to use a mesh that spans only the fluid part in the CFD stage and to use a mesh that spans only the solid part in the thermal stress analysis stage, though a few iterations between stages might be necessary to obtain a converged solution. The iteration will not be necessary if the fluid temperature, hence the ambient fluid temperature, can be taken as constant and no special effect depending on temperature is considered. In fact, in this case the temperature field need not be solved in the CFD stage because turbulent film coefficient can be computed only from the velocity field and the fluid properties [3].

The second method is to interpolate temperature defined at the nodes of the CFD mesh to the nodes in the stress analysis mesh. We call this method as volume mapping method. Because no recalculation of temperature in ANSYS is necessary, the volume mapping method is a more straightforward method compared to the surface mapping method. However it is necessary in the CFD stage to compute the temperature field in the solid part of the pipe.

Investigated cases:

Using the methods defined in the last section, we have investigated four cases:

- 1) The flow field and the temperature field are solved using FLOTRAN and the thermal stress field is solved using ANSYS. The mesh is generated using ANSYS preprocessor and is shown in Fig. 1. The fluid part of the mesh contains tetrahedral and pyramid elements and the solid part contains tetrahedral, pyramid and hexahedral elements. Because no interpolation facility is available between FLOTRAN and ANSYS, the same mesh is used in FLOTRAN and ANSYS except that the element type number is converted from 142 to 95. Note that element type 45, 72 and 92 cannot be used because the solid part of the mesh contains pyramids.
- 2) The flow field and the temperature field are solved using SCRYU/Tetra and the thermal stress field is solved using ANSYS. The computational mesh is generated using SCRYU/Tetra preprocessor and is shown in Fig. 2. The fluid part of the mesh contains tetrahedral, pyramid and prism elements and the solid part contains only tetrahedral elements. The solid part of the mesh is reused in ANSYS to compute the thermal stress field. Element type 95 is used in ANSYS.
- 3) The flow and temperature field obtained in Case 2 are used again. In ANSYS, the mesh shown in Fig. 3 is used. To interpolate the temperature data obtained using SCRYU/Tetra to the mesh for ANSYS, the surface mapping method is used. That is, the turbulent film coefficient on the inner surface of the pipe and the ambient fluid temperature is interpolated to the corresponding surface of the ANSYS mesh. SCRYU/Tetra system is equipped with this function. Using the data as the boundary condition, the temperature distribution in the solid part of the pipe is recalculated and subsequently the thermal stress is analyzed. In ANSYS, element type 70 and 95 are used to calculate the temperature and the thermal stress field, respectively.
- 4) Same as 3) except that the volume mapping method is used. In this case, the nodal temperature data itself is interpolated from SCRYU/Tetra mesh to the nodes of the ANSYS mesh and no temperature calculation is done on the ANSYS side. SCRYU/Tetra system is equipped with functions to interpolate temperature and other fields to a different mesh. In ANSYS, the mesh shown in Fig. 3 is used. Thermal stress field is analyzed using element type 95.

The sizes of the computational meshes are shown in Table 1.

Table 2 Number of elements in computational mesh. Both solid and fluid elements are counted in the mesh size in CFD column.

| Case | Mesh size | |
|------|-----------|-------|
| | CFD | ANSYS |
| 1 | 67624 | 19630 |
| 2 | 51531 | 16116 |
| 3 | | 59439 |
| 4 | | 59439 |

Results:

No difficulty in convergence in solving the discretized linear equations is observed in the cases in this section. To solve for temperature fields in ANSYS, PBCGM method is selected because the PGMR method is recommended for solving the energy equation for ill-conditioned conjugate transfer problems. In general, the Preconditioned BiCGStab method (PBCGM) requires less memory than the PGMR method. It is also recommended for extremely ill-conditioned conjugate heat transfer problems.

In SCRYU/Tetra, fluid equations are solved by SIMPLEC method. Discretized momentum equations are solved by the BiCGStab method and the continuity equations are solved by AMG method. These are the default solution algorithms of SCRYU/Tetra solver.

The CFD part of calculation results of Case 1 and Case 2 are shown in Fig. 4 and Fig. 5, respectively. The appearance of velocity vector in Case 1 is different from that in Case 2 partly because no prism layers are used along the wall of the pipe and consequently no vectors are plotted near the wall. The temperature distribution on the outer surface of the pipe is also different between Case 1 and Case 2 significantly near inlet 1. Because no experimental data is available in our test problem, we cannot really tell which result is more accurate. However, the high temperature near inlet 1 in Case 1 seems unrealistic.

Another problem in Case 1 is that computation time to solve for the flow field takes longer: it takes 120 minutes in Case 1, compared to 4.3 minutes in Case 2. The drastic difference in computational time comes because no convergence is attained in Case 1 within the predefined limit of 300 cycles, while a convergence is attained in 47 cycles in Case 2.

In Fig. 6, we compare the temperature distribution obtained in SCRYU/Tetra in Case 2 and that obtained using ANSYS in Case 3. Because the boundary conditions for Case 3 are generated from the result of Case 2, the results in Fig. 6 should be identical. The Figure shows the temperature distributions are similar, but small differences are observed near inlet 1 and in the bottom part of the pipe. We attribute the source of difference to the accuracy of interpolation of surface data and to the difference in the discretization methods used in SCRYU/Tetra and ANSYS.

In Fig. 7, we show the result of thermal stress analysis. Equivalent stress and displacement distributions are shown for each case. As shown in the figure, equivalent stress and displacement distributions are similar in all cases. It is rather surprising that a similar result is obtained in Case 1 with other cases though temperature distribution is considerably different as shown in Fig. 4 and Fig. 5. No effect of high temperature distribution near inlet 1 is seen in Fig. 7. One reason for this is that the nodes on inlet 1 are allowed to move freely in the plane of inlet 1.

The results in Fig. 7 (b), (c) and (d) are very similar and shows that a reasonable accuracy can be obtained by using all three methods of transferring temperature fields from SCRYU/Tetra to ANSYS. The results in Case 3 and Case 4 are nearly identical and shows that the surface mapping method and the volume mapping method can work equally well.

The equivalent stress distribution is rougher in Case2, probably because the quality of the tetrahedral mesh generated using SCRYU/Tetra is not as suitable as that of a mesh generated using ANSYS preprocessor. This can be known because smoother equivalent distribution is obtained in Case 1 where the coarseness of the mesh is comparable with that of Case 2.

This situation can be improved by using different types of elements as shown in Fig. 8. The element type 92 gives the closest result compared to Cases 3 and Case 4 in Fig. 7, where a finer mesh is used for thermal stress analysis. Element type 45 gives a result different from other results even if a finer mesh is used and should not be used with a mesh containing tetrahedral elements. The result also shows that the second order elements (type 92 and 95) give better results compared to the first order elements (type 45 and 72).

Practical exhaust manifold model

A more complicated model is used for additional investigation. The computational mesh to solve for flow field and associated thermal stress field is shown in Fig. 9. Fig. 9 (a) shows the fluid part and the solid (cast iron) part and Fig. 9 (b) shows the fluid path. The volume of the fluid part of the mesh is 0.00327 m^3 . The mesh is created using SCRYU/Tetra preprocessor. The fluid part of the mesh contains 192537 elements consisting of tetrahedral, pyramid and prism elements and the solid part contains 47566 tetrahedrons. The mesh in Fig. 9 (c) contains 155707 tetrahedral elements and is used for thermal stress analysis in ANSYS using the volume mapping method.

For current geometry, the flow field is solved using SCRYU/Tetra. FLOTRAN is not used because the last pipe example suggests that the computation take too much time.

Boundary conditions are as follows. The mass flow rate at each inlet is 0.0628 kg/s and inlet fluid temperature is 1023 K . The outlet pressure is 138.7 kPa . A filter condition is applied in the cylinder part of the mesh to model a weak pressure loss caused by a filter. The film coefficient on the outer wall of the exhaust manifold is set to $2.9 \text{ W/m}^2\cdot\text{K}$ and ambient temperature is set to 573 K . Node movements are not allowed at the holes in the flange.

As a preliminary investigation, the flow field and the temperature field is solved treating the gas as an incompressible fluid and as an ideal gas. The same property as in the pipe example is used for air treated as an incompressible fluid and iron. When air is treated as an ideal gas, the gas constant is set to $287 \text{ J/Kg}\cdot\text{K}$.

Fig. 10 shows the calculation result for the case air is treated as an ideal gas. The contour plot of Mach number shows that the Mach number becomes as high as 0.7. This implies that the air has to be treated as an incompressible fluid. The effect of compressibility comes up drastically in the temperature distribution as shown in Fig. 11. When air is treated as incompressible, the effect of temperature drop as the gas accelerates is not taken into account. This results in a more uniform temperature distribution of the exhaust manifold.

A further preliminary investigation is carried out by changing the heat conductivity of the iron as a function of temperature. However, this effect is small and a very similar result as in the right figure of Fig. 11 is obtained. Hence this topic will not be discussed here any further.

Taking the result of the preliminary study into account, thermal stress field is analyzed by treating air as an ideal gas. Two cases are investigated. In the first case thermal stress field is analyzed using the direct conversion method, i.e., based on the same mesh as one used in the CFD stage. We call this case as Case 5. In the second case, the temperature field is interpolated to the mesh shown in Fig. 9 (c) using the volume mapping method. We call this case as Case 6. In both cases element type 95 is used to analyze the thermal stress field in ANSYS.

The results of thermal stress analyses are shown in Fig. 12. The result shows that the displacement distributions of two different cases are nearly identical and that the direct conversion method and the volume mapping method can work equally well.

Discussion

The investigations above suggest that the combination of SCRYU/Tetra and ANSYS can work as well as the combination of FLOTRAN and ANSYS for thermal stress analyses. If the geometry under analysis is very complicated, in fact the use of SCRYU/Tetra may be favored. The first reason for this is that SCRYU/Tetra preprocessor can automatically generate a hybrid mesh with prism layers inserted along the no-slip walls. On the other hand, only tetrahedral mesh can be generated for FLOTRAN using the ANSYS automatic mesh generator. In general, the lack of boundary layer elements has a bad effect on the accuracy of a CFD analysis. Secondly, the convergence of the flow field may be. Thirdly, the mapping methods can be used if SCRYU/Tetra is used.

In Cases 2, 3 and 4 and Cases 5 and 6, we are able to obtain nearly identical result using mapping methods as that of direct conversion method. The error caused by the mapping appears to be small. The mapping methods give a great advantage if a fine resolution is required in different parts of the geometry in CFD stage and stress analysis stage. In such cases, a considerable saving in computer memory and calculation time can be expected.

The memory and CPU requirements are summarized in Table 3 for a typical mesh containing 100,000 elements. The table shows that SCRYU/Tetra is considerably faster than FLOTRAN if the number of iterations before the calculation converges is the same.

Table 3. Expected memory and CPU time requirements for a typical computational mesh containing 100,000 elements. CPU time for CFD is the time to advance 100 cycles. The CPU time is estimated using a PC with Pentium III 866 MHz CPU. Note that considerable deviation can result depending on the geometry, mesh quality, shapes of the elements used in the mesh and other factors.

| | FLOTRAN Type 142 | SCRYU/Tetra | ANSYS Element Type | | | |
|-----------------|---------------------|-------------|--------------------|------|----|----|
| | | | 45 | 95 | 72 | 92 |
| Memory (Mbytes) | - | 25 | 62 | 1516 | - | - |
| CPU Time (sec) | 5915 | 692 | 699 | 3152 | - | - |

The table also shows that the second order elements consume more time and memory compared to the first order elements, though our investigation show that the accuracy of the second order elements is better. These factors should be taken into account when meshes for CFD and thermal stress analysis are designed.

Our investigation suggests that element type 92 gives a smoother result compared to element type 95. Thus if the memory and CPU time requirements allow, we recommend the use of element type 92. It gives a reasonable result for relatively small number of elements. One drawback of using element type 92 is that only tetrahedral elements can be used in the mesh. However this does not cause a problem because in SCRYU/Tetra preprocessor one can specify in which part of the mesh the prism layer should be inserted. This allows one to leave the solid part of the mesh strictly consisting of tetrahedral elements.

The Case 3 and Case 4 show that the surface mapping method can work as well as the volume mapping method. However the surface mapping method is more cumbersome, because one has to set up the thermal analysis problem in ANSYS. Another problem arises when the quality of the CFD mesh is not good and the prism layer thickness cannot be considered uniform. In such a case the definition of the film coefficient and the ambient temperature becomes somewhat vague. For these reasons we recommend that the volume mapping method is used as far as the computer resource allows a CFD including the temperature analysis of the solid part.

The volume mapping method is sufficiently flexible that SCRYU/Tetra can be coupled with other systems such as I-DEAS, NASTRAN and ABAQUS. In the same way ANSYS can be coupled with other CFD systems.

Conclusion

Methods to analyze flow-coupled thermal stress fields are investigated using FLOTRAN, SCRYU/Tetra and ANSYS. It is found that:

- 1) The combination of SCRYU/Tetra and ANSYS can be more efficient, flexible and accurate compared to the combination of FLOTRAN and ANSYS.
- 2) The volume mapping method is the favored method to couple CFD and thermal stress analysis.

Reference:

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. H. Tennekes and J. L. Lumley, *A first course in turbulence*, MIT Press, Massachusetts, 1972.

Figures

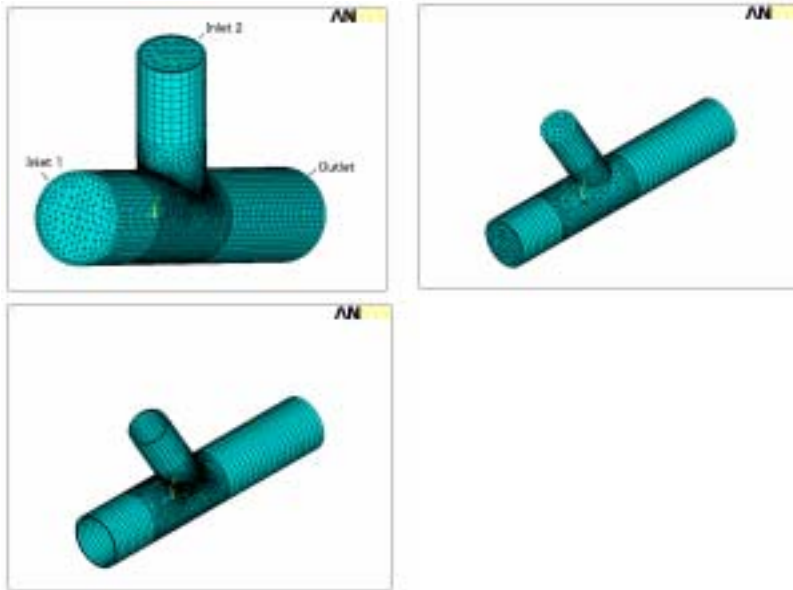


Fig. 1 Computational mesh for Case 1. Left: fluid and solid part. Middle: fluid part. Right: solid part.

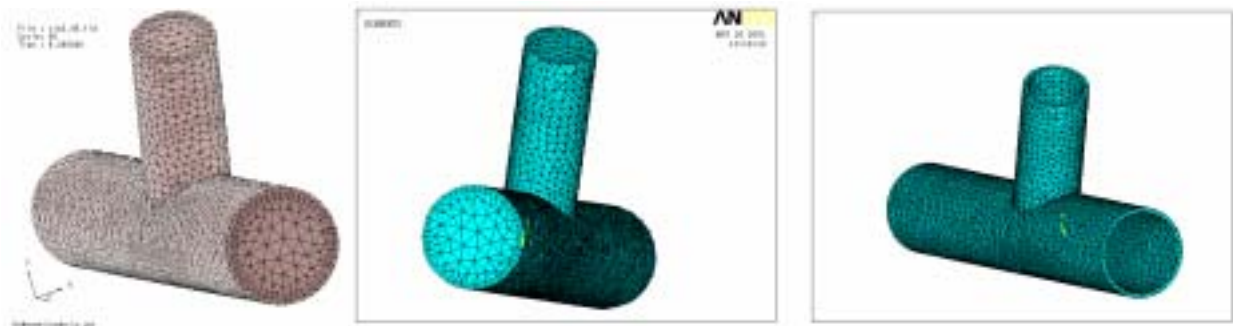


Fig. 2 Computational mesh for Case 2. Left: fluid and solid part. Middle: fluid part. Right: solid part. Note that prism elements are used in the fluid part to resolve boundary layer.

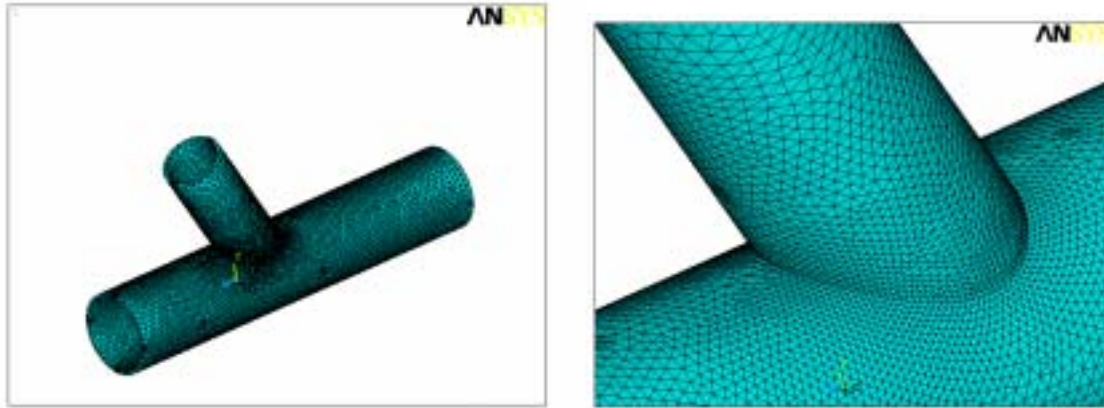


Fig. 3 Computational mesh for thermal stress analysis for case 3 and case 4. Only the solid part of the pipe is meshed. The mesh contains only tetrahedral elements.

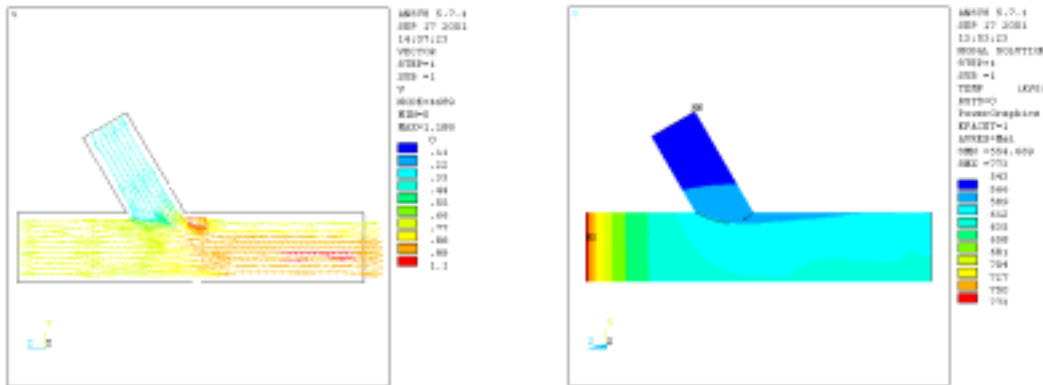


Fig. 4 The CFD result of Case 1. Left: velocity field on center plane. Right: temperature distribution on the outer surface of the pipe.

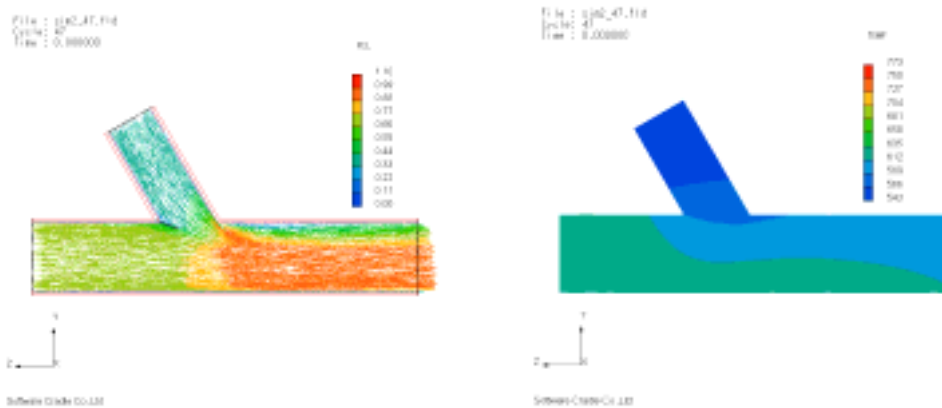


Fig. 5 The CFD result of Case 2. Left: velocity field on center plane. Right: temperature distribution on the outer surface of the pipe.

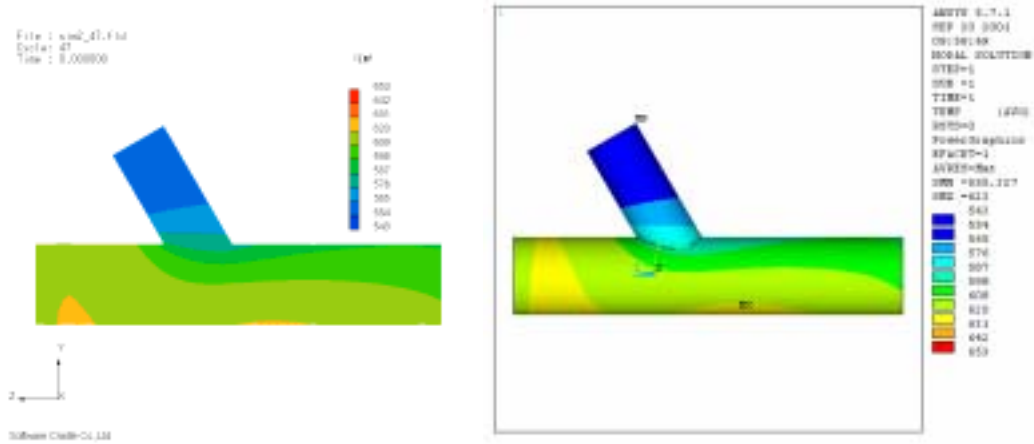
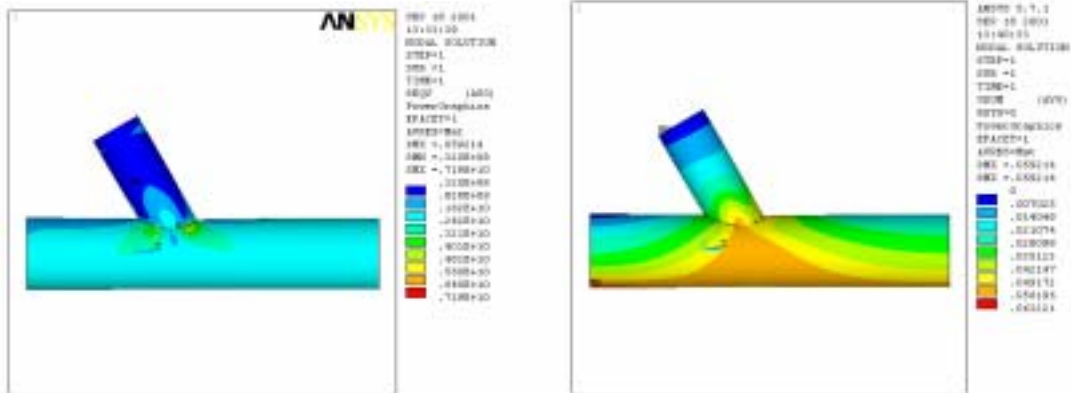
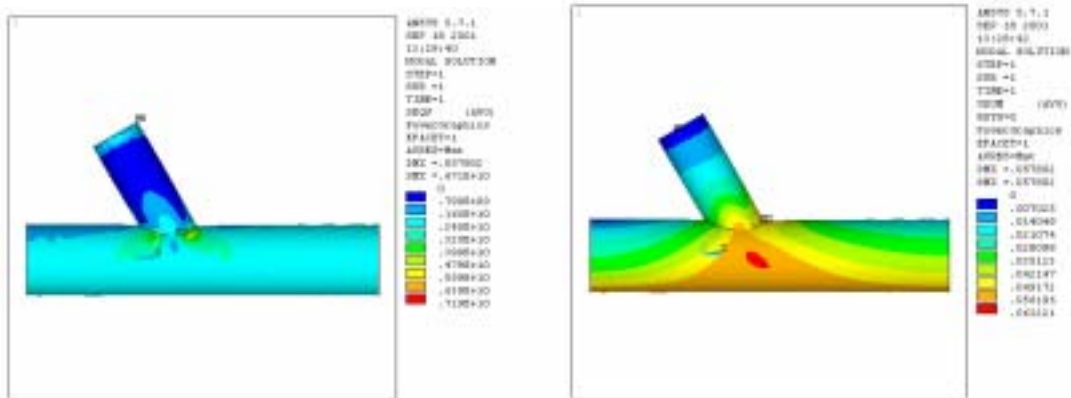


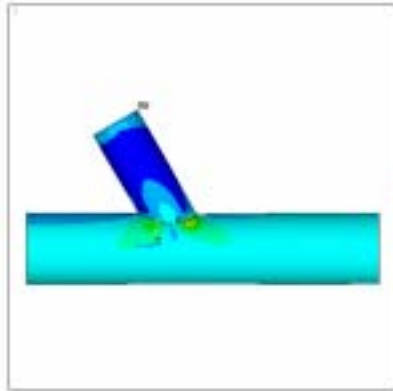
Fig. 6 Temperature distribution on the outer surface of the pipe. Left: Case 2. Right: Case 3 (obtained using ANSYS).



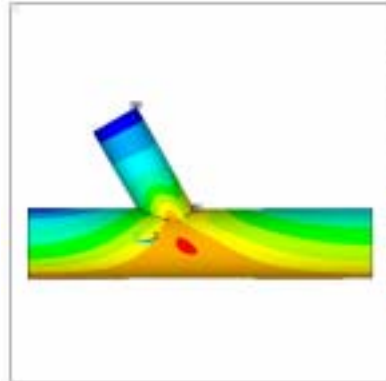
(a) Case 1



(b) Case 2

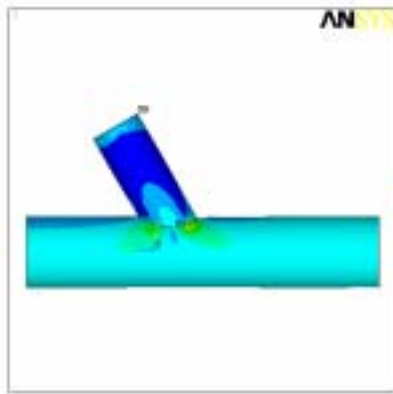


4800 5.7.1
 STEP 27 2001
 20.00e+00
 GLOBAL SOLUTION
 STEP=1
 SUB=1
 TIME=1
 SUB=1400
 SUB=0
 PUNCH=QBRI0
 PSCALE=1
 PVAL=Max
 MIN = 0.0000
 MAX = 6980.00
 0
 7000.00
 5000.00
 3000.00
 1000.00
 0
 -1000.00
 -3000.00
 -5000.00
 -7.18E+03

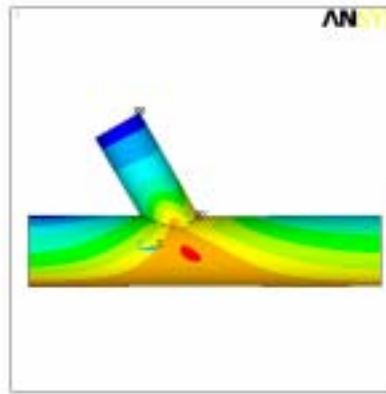


4800 5.7.1
 STEP 27 2001
 20.00e+00
 GLOBAL SOLUTION
 STEP=1
 SUB=1
 TIME=1
 SUB=1400
 SUB=0
 PUNCH=QBRI0
 PSCALE=1
 PVAL=Max
 MIN = 0.0000
 MAX = 0.000110
 0
 0.000010
 0.000020
 0.000030
 0.000040
 0.000050
 0.000060
 0.000070
 0.000080
 0.000090
 0.000100
 0.000110

(c) Case 3



4800 5.7.1
 STEP 27 2001
 20.00e+00
 GLOBAL SOLUTION
 STEP=1
 SUB=1
 TIME=1
 SUB=1400
 SUB=0
 PUNCH=QBRI0
 PSCALE=1
 PVAL=Max
 MIN = 0.0000
 MAX = 6980.00
 0
 7000.00
 5000.00
 3000.00
 1000.00
 0
 -1000.00
 -3000.00
 -5000.00
 -7.18E+03



4800 5.7.1
 STEP 27 2001
 20.00e+00
 GLOBAL SOLUTION
 STEP=1
 SUB=1
 TIME=1
 SUB=1400
 SUB=0
 PUNCH=QBRI0
 PSCALE=1
 PVAL=Max
 MIN = 0.0000
 MAX = 0.000110
 0
 0.000010
 0.000020
 0.000030
 0.000040
 0.000050
 0.000060
 0.000070
 0.000080
 0.000090
 0.000100
 0.000110

(d) Case 4

Fig. 7 The result of thermal stress analysis. Left: equivalent stress distributions. Right: displacement distributions.

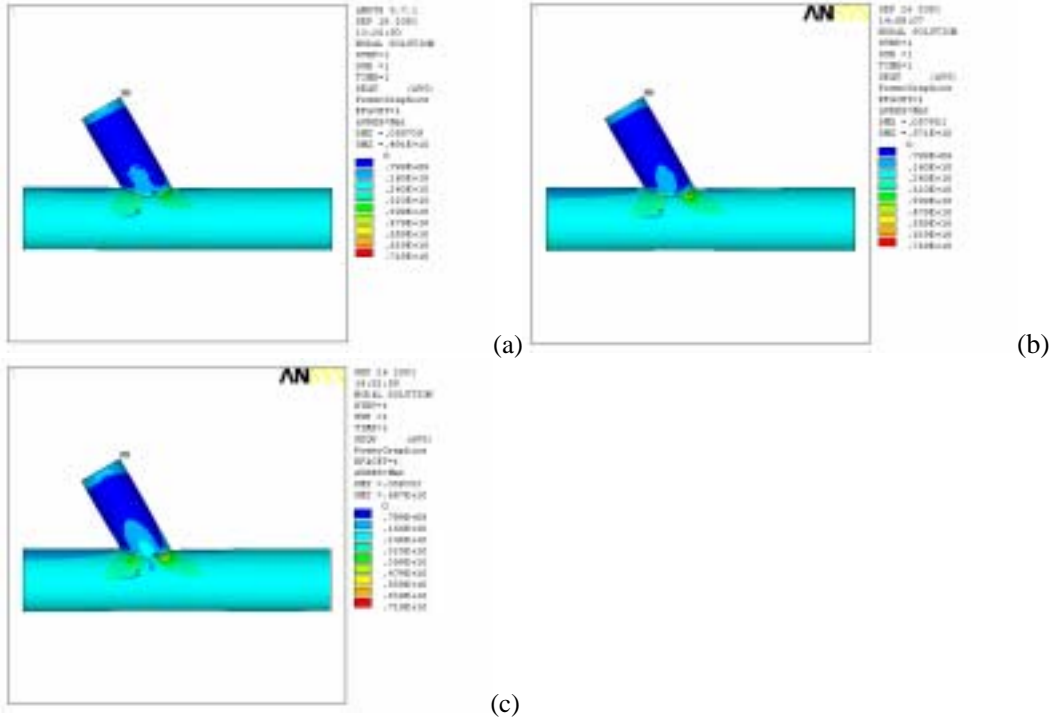


Fig. 8 The equivalent stress distribution. Conditions are same as Case 2 except that in the thermal stress analysis stage, element type 45, 72 and 92 are used in (a), (b), (c), respectively.

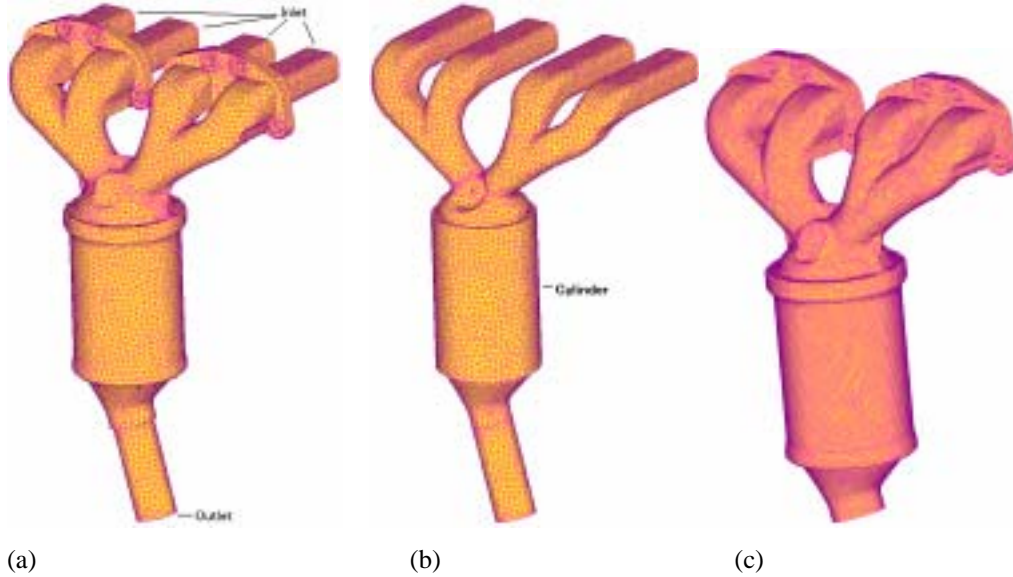


Fig. 9 Computational mesh of practical model. (a) Solid part and fluid path. (b) Fluid path. (c) Mesh used for thermal stress analysis.

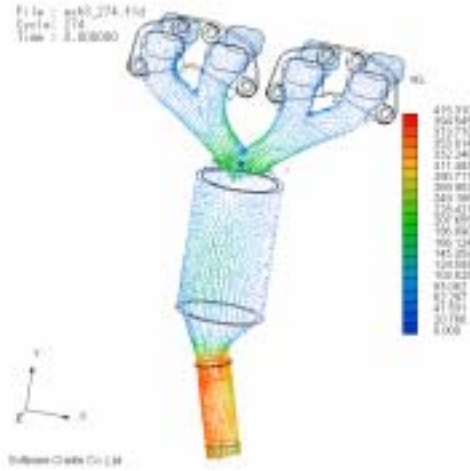
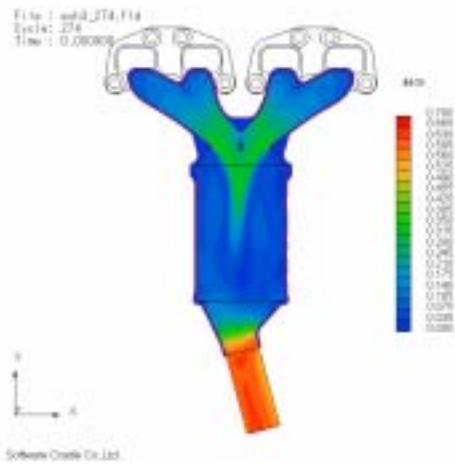


Fig. 10 Result of exhaust manifold calculation for the case air is treated as an ideal gas. Left: Contour plot of Mach number in the center cross section of the filter cylinder. Right: flow pattern

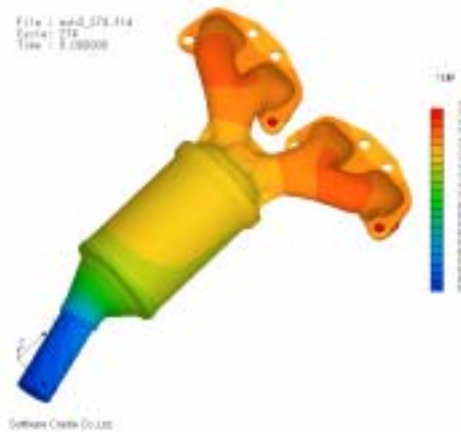
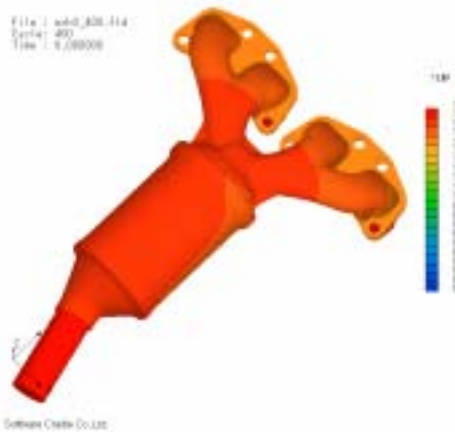


Fig. 11 The temperature distribution of exhaust manifold. Left: air is treated as an incompressible fluid. Right: air is treated as an ideal gas. Temperature scale is in centigrade.

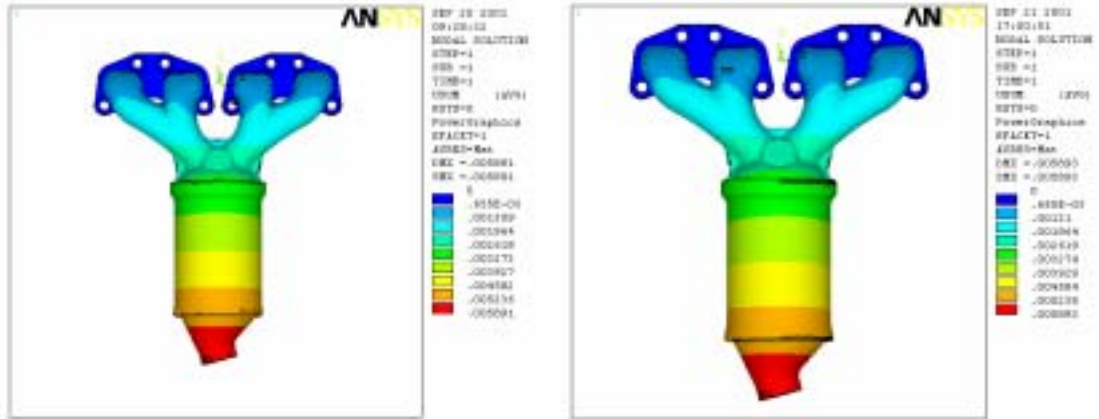


Fig. 12. Displacement distribution of practical exhaust model. Left: Case 5. Right: Case 6.