Project 2 - Report: Iso-Parametric Q4 Element for Heat Transfer

Piotr Liebersbach

November 29, 2018

Contents

For Project 2, heat transfer was modeled using the finite element method. In particular, 4 node quadrilateral elements were used to discretize a domain, and solve the heat equation. The element was used to extend FEMCOD for the heat transfer analysis, and the resulting system of equations was solved using its triple factoring method, as well as its skyline storage scheme. This element was implemented via a subroutine called ISQUAD. The element was tested with a patch test, and then used to model heat transfer in a plate with a circular hole in the middle. The results from the code were analyzed and show a temperature distribution in the plate that is expected from intuition. Supporting material can be found in the Appendix.

1 Q4 Element: Background

The modeled element is an iso parametric four node quadrilateral element. An iso parametric element is a more general quadrilateral element; a bilinear rectangle is limited to strictly rectangular shapes and this limits its applicability to simple geometries. An iso parametric element alleviates this limitation by allowing for more general parallelogram shapes. The physical element is mapped to a reference space, in which it is represented by a square. The element has four nodal degrees of freedom, and each one is returned by a shape function that is linear in η and ξ directions. This reference space is used to develop the finite element formulation for the element to be used in forming the stiffness matrix. Because of the use of physical and reference coordinates, shape functions and their derivatives are mixed between the two coordinate systems. This leads to complicated expressions in forming the stiffness matrix, which involve integrating products of shape function derivatives. Because of the definition of the element geometry, these expressions are rational functions, and cannot be integrated analytically. Therefore numerical integration is used to evaluate the shape function derivatives at points inside the element, and this method is used to form the element stiffness matrix. Gauss Quadrature is used for the numerical integration to obtain the stiffness matrix, as it uses less integration points than other methods for similar orders of accuracy. Even with the numerical integration though, the stiffness matrix cannot be integrated exactly. However, standard practice in finite elements has shown that good results can be obtained if squares of strains are integrated exactly ($\underline{B}^T \underline{B}$). To accomplish this a four point Gauss Quadrature rule is employed. Using this approach the stiffness matrix for the element is evaluated, and is assembled into the global stiffness matrix. The implementation of this procedure for calculating the element stiffness matrix in FEMCOD is given in the Appendix.

2 Patch Test

The validity of the ISQUAD subroutine that calculates the element stiffness matrix was validated via a four element patch test. The patch represents a rectangular domain with two sides (opposite each other) at two fixed temperatures, and the other two sides as insulated. From heat transfer theory, specifically conduction, heat will flow from the hot side to the cold side, varying linearly through the domain. Halfway through the domain, the temperature is expected to be the average of the two wall temperatures.

The first patch test employs four square elements, which can be seen in the Appendix. In this test, the temperatures at the left hand nodes are fixed at $100°K$, while the right hand nodes are fixed at $0°K$. The three nodes in the middle of the patch are unconstrained, and are adiabatic. Since these nodes are exactly in the middle of the patch, it is expected that the temperature at these nodes will be $50°K$. Solving this patch with FEMCOD, the results show that the middle nodes are exactly at $50°K$. This result can be read from the results listed in the Appendix, as well as by looking at the temperature contour of the patch created from the result. This contour is given in Figure 1, and shows a linear temperature distribution from left to right, with average temperature in the middle of the patch. Because

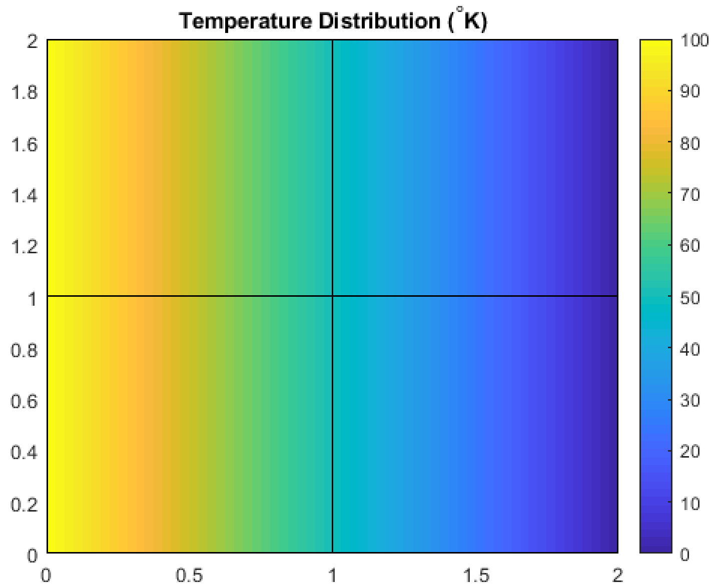

Figure 1: Temperature field in patch test 1.

the patch is made from undistorted square elements, it is expected that the contours will be straight and uninform which can also be seen.

A second patch was also performed to test for distorted elements. In this patch test, the central node was moved toward the north east corner of the patch, while all the nodes remained in the same place. This second patch test can also be found in the Appendix. The set up of the second patch was also similar to the set up of the first patch. Since the center node is no longer in the center, the three central nodes will not have average temperature, but will instead have temperatures linearly interpolated by distance. The contour of temperature for the second patch can be seen in Figure 2. It is expected that the center node will be the coldest of the three central nodes, since it is closest to the cold side, and that the two nodes (2 & 8) on the top and bottom will be at higher temperatures. Since node 5 is closer to node 8, it is expected that node 8 will colder than node 2, which can be seen in the nodal solution. These trends can also be seen in the temperature contour in Figure 2.

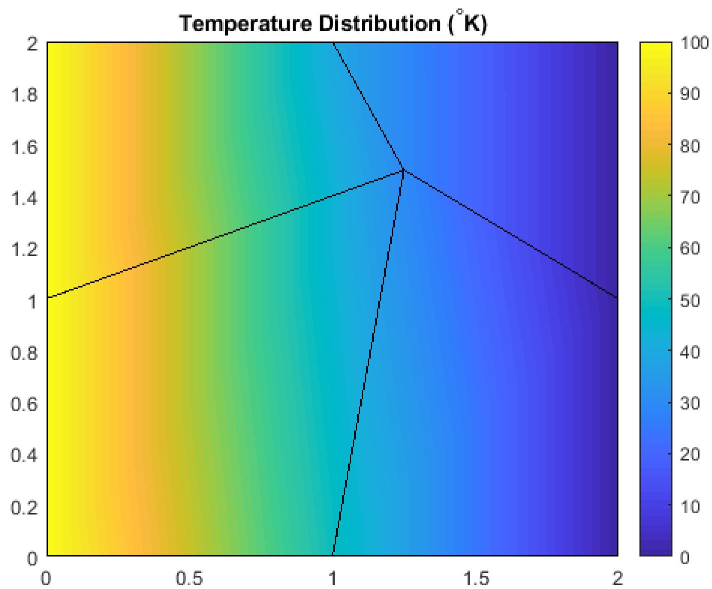

Figure 2: Temperature field in patch test 2.

3 Problem: Heat Transfer in Plate with a Hole

With the successful completion of the patch test, the element implementation was then tested with a bigger problem. For this task, heat transfer was modeled in a plate with a hole in the middle. Due to symmetry of the model, only a quarter section of the plate was modeled. The resulting mesh of the model can be found further in the Appendix. The model consists of 36 quadrilateral elements, and 49 nodes. The numbering scheme can be found with the mesh in the Appendix. A heat flux is applied to the boundary of the circular hole with a magnitude of $5000W/m^2$, using consistent nodal loads. The temperature across the bottom and left walls are unconstrained and insulated, and the temperature across the top and right walls are set to $300°K$. The thermal conductivity of the material is set to $100W/m^2K$, and the elements have unit thickness.

For the temperature field, it is expected that the top and rigPht walls will be at the prescribed temperature of $300°K$. The circular hole will be at a uniform temperature, and this is expected to be the hottest surface in the entire plate. In addition, this plate has symmetry

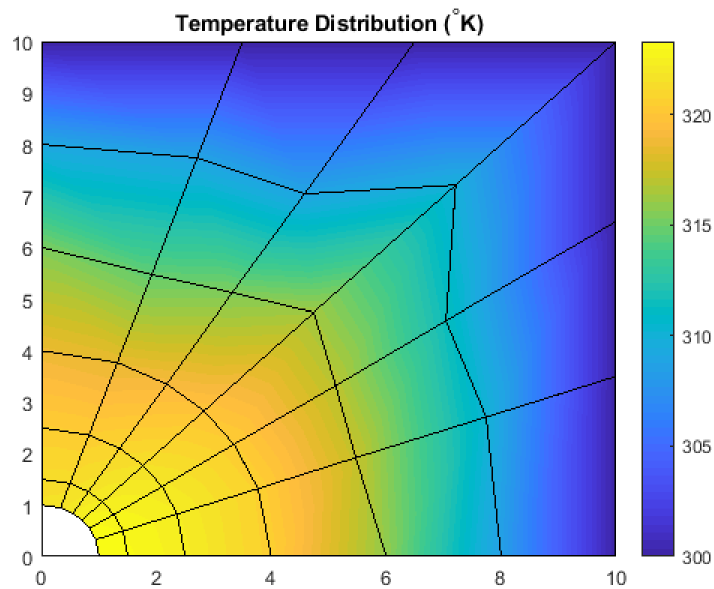

Figure 3: Temperature field in plate with a hole.

in the circumferential direction. Therefore, it is expected that the contours will show circles of constant temperature, concentric with the hole in the plate, and decreasing in temperature in the radial direction until they reach the outer wall temperatures.

The solution to the temperature field is shown in a contour plot of Figure 3. As expected, the hole is the hottest part of the plate, and the top and right walls return the prescribed temperature. In addition, the contour forms distinct rings of constant temperature, as expected. The solution fits with heat transfer principles, and shows good implementation of the element.

4 Reduced Integration

After implementing the four node quadrilateral with full integration successfully, the code was edited to employ a reduced integration rule. For the Q4 element, reduced integration is achieved with one point Guass Quadrature. Reduced integration decreases the compu-

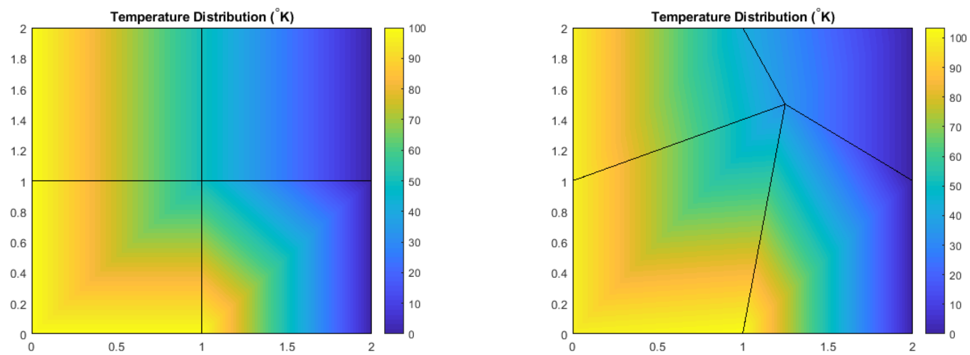

Figure 4: Temperature contours for distorted and undistorted element patch tests using reduced integration.

tational time to form the stiffness matrix and softens the mesh, however at the risk of ill conditioning or a singular stiffness matrix. The effects of reduced integration on the solution are explored below.

4.1 Patch Test

Both patch tests using reduced integration show some nonphysical results. The biggest problem with the solution is that node two in the undistorted patch test is at $100°K$, and in the distorted element it reaches $103°K$. Neither of these results makes physical sense, as the temperature at node two should be interpolated from the side nodes by distance. The temperature of node two is especially concerning in the distorted element patch, because it is greater than the prescribed temperature at the wall, and there is not heat fluxes or heat generation anywhere in the patch. These patch tests show some clear problems with reduced integration for heat transfer, which can be expected in the main model simulated.

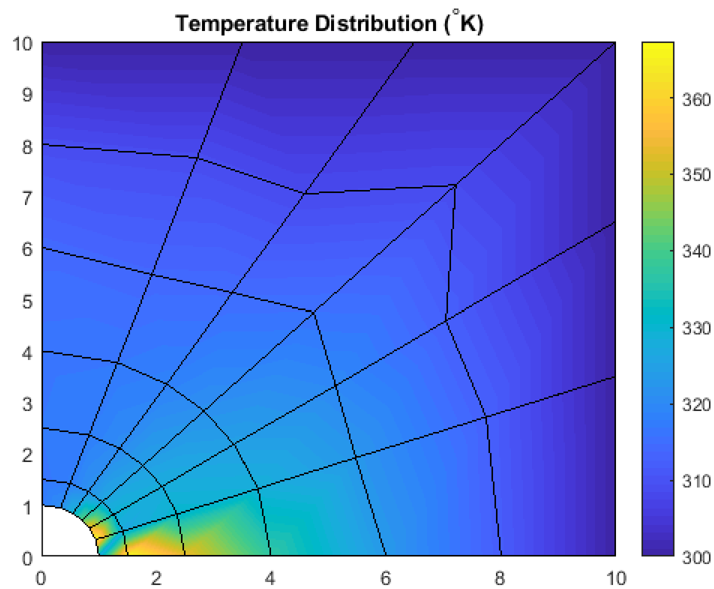

Figure 5: Temperature field in plate with a hole using reduced integration.

4.2 Heat Transfer in Plate with a Hole

Heat transfer in the plate with a hole in the center is modeled again using reduced integration. The same boundary conditions and mesh are used as with the full integration. Based on the two patch tests with reduced integration, non physical results for the temperature field are expected again. The temperature contour can be seen in Figure 5. In this plot, the contours are not uniform as expected, and some nodes on the hole are at the cold temperature, even though heating is applied locally to them. In addition, high temperatures exist in the second and third circles of elements, which is not expected, since max temperatures should occur on the hole where the heating is applied. Once again, it can be seen that reduced integration does not produce physical results for the temperature field.

5 Conclusions

A four node quadrilateral element was implemented in FEMCOD to model heat transfer. The implementation using full integration passed a patch test, and gave good results for the temperature distribution in a heated plate with a hole in the center. Reduced integration was also considered, and as expected, it did not give accurate results, compared to the full integration.

6 Appendices