TP 14357E

Computational Fluid Dynamics Analysis of Local Heating of Propane Tanks

Prepared for

Transportation Development Centre
of
Transport Canada

by

Department of Mechanical and Materials Engineering
Queen's University
Kingston, Ontario

December 2004

TP 14357E

Computational Fluid Dynamics Analysis of Local Heating of Propane Tanks

By

K.T. Yoon
A.M. Birk
Department of Mechanical and Materials Engineering
Queen's University
Kingston, Ontario

December 2004

Since some of the accepted measures in the industry are imperial, metric measures are not always used in this report.

Project team

K. T. Yoon
A.M. Birk

Un sommaire français se trouve avant la table des matières.

Transport Canada / Transports Canada

PUBLICATION DATA FORM

1. Transport Canada Publication No. TP 14357E	2. Project No. 5422	3. Recipient's Catalogue No.
4. Title and Subtitle Computational Fluid Dynamics Analysis of Local Heating of Propane Tanks		5. Publication Date December 2004
		6. Performing Organization Document No.
7. Author(s) K.T. Yoon and A.M. Birk		8. Transport Canada File No. 2450-FP654-5
9. Performing Organization Name and Address Department of Mechanical and Materials Engineering McLaughlin Hall Queen's University Kingston, Ontario Canada K7L 3N6		10. PWGSC File No. MTB-3-00363
		11. PWGSC or Transport Canada Contract No. T8200-033505/001/MTB
12. Sponsoring Agency Name and Address Transportation Development Centre (TDC) 800 René Lévesque Blvd. West Suite 600 Montreal, Quebec H3B 1X9		13. Type of Publication and Period Covered Final
		14. Project Officer A. Vincent

15. Supplementary Notes (Funding programs, titles of related publications, etc.)

Co-sponsored by Transport Canada's Transport Dangerous Goods Directorate

16. Abstract

This report describes a three-dimensional, time transient CFD (computational fluid dynamics) analysis of LPG (liquefied petroleum gas) tanks with various types of local heating by fire. The analysis was performed as part of a larger study of the effects of thermal protection defects.

Thermal protection systems are designed to protect tank-cars from accidental fire impingement. If a tank has thermal protection defects, it will heat up more rapidly in a fire and experience higher wall temperatures, lading temperatures and tank pressure, increasing the risk of tank failure.

Testing shows that when a tank is heated in a fire, the liquid near the wall is heated first. This warm liquid then rises to the liquid surface, where it generates vapour to pressurize the tank. This rising of the warm liquid is called temperature stratification. Liquid surface temperature is believed to be closely related to tank pressure.

The CFD analysis was carried out to study the effect of defect size and location on the transient heating of the liquid lading in the tank. Results show that heating near the tank ends or near the liquid level on the tank sides causes the most rapid rise in liquid surface temperature. Heating near the tank bottom results in the slowest rise in liquid surface temperature.

17. Key Words Computational fluid dynamics, CFD, thermal protection defects, local heating by fire, temperature stratification	18. Distribution Statement Limited number of copies available from the Transportation Development Centre

19. Security Classification (of this publication) Unclassified	20. Security Classification (of this page) Unclassified	21. Declassification (date) —	22. No. of Pages x, 20, apps	23. Price Shipping/ Handling

CDT/TDC 79-005
Rev. 96

Canada

Transports Canada | Transport Canada

FORMULE DE DONNÉES POUR PUBLICATION

1. N° de la publication de Transports Canada TP 14357E	2. N° de l'étude 5422	3. N° de catalogue du destinataire
4. Titre et sous-titre Computational Fluid Dynamics Analysis of Local Heating of Propane Tanks		5. Date de la publication Décembre 2004
		6. N° de document de l'organisme exécutant
7. Auteur(s) K.T. Yoon et A.M. Birk		8. N° de dossier - Transports Canada 2450-FP654-5
9. Nom et adresse de l'organisme exécutant Department of Mechanical and Materials Engineering McLaughlin Hall Queen's University Kingston, Ontario Canada K7L 3N6		10. N° de dossier - TPSGC MTB-3-00363
		11. N° de contrat - TPSGC ou Transports Canada T8200-033505/001/MTB
12. Nom et adresse de l'organisme parrain Centre de développement des transports (CDT) 800, boul. René-Lévesque Ouest Bureau 600 Montréal (Québec) H3B 1X9		13. Genre de publication et période visée Final
		14. Agent de projet A. Vincent

15. Remarques additionnelles (programmes de financement, titres de publications connexes, etc.)

Projet coparrainé par la Direction générale du transport des marchandises dangereuses de Transports Canada

16. Résumé

Ce rapport rend compte d'une analyse tridimensionnelle et chronologique, par simulation numérique en mécanique des fluides (CFD, *computational fluid dynamics*), de citernes de GPL (gaz de pétrole liquéfié) soumises à divers types de chauffage localisé causé par des flammes. Cette analyse a été menée dans le cadre d'une vaste étude sur les effets des défauts de la protection thermique des citernes.

Les systèmes de protection thermique sont conçus pour protéger les wagons-citernes contre un feu avec flammes enveloppantes susceptible de survenir lors d'un accident. En présence d'un feu, une citerne dont la protection thermique est défectueuse chauffera plus rapidement, ses parois et son contenu atteindront des températures plus élevées, et sa pression interne atteindra des valeurs plus grandes, ce qui la rendra plus fragile à la rupture.

Les essais révèlent que lorsque des flammes sont appliquées à une citerne, le liquide adjacent à la paroi se réchauffe d'abord. Puis, ce liquide chaud monte à la surface du liquide, où il génère des vapeurs qui mettent la citerne en pression. Cette montée du liquide chaud entraîne ce que l'on appelle la *stratification des températures*. Ainsi, la température à la surface du liquide serait étroitement liée à la pression interne de la citerne.

L'analyse par CFD avait pour objet d'étudier l'effet de l'étendue et de l'emplacement du défaut de protection thermique sur la montée en température du contenu de la citerne. Les résultats indiquent que l'application de chaleur près des extrémités de la citerne ou sur les parois, près du niveau de remplissage, entraîne le réchauffement le plus rapide de la surface du liquide. À l'inverse, c'est lorsque la chaleur est appliquée dans la partie inférieure de la citerne que la température de la surface du liquide monte le plus lentement.

17. Mots clés Simulation numérique en mécanique des fluides, CFD, défauts de protection thermique, chauffage localisé par des flammes, stratification des températures	18. Diffusion Le Centre de développement des transports dispose d'un nombre limité d'exemplaires.

19. Classification de sécurité (de cette publication)	20. Classification de sécurité (de cette page)	21. Déclassification (date)	22. Nombre de pages	23. Prix
Non classifiée	Non classifiée	—	x, 20, ann.	Port et manutention

EXECUTIVE SUMMARY

This report describes a three-dimensional, time transient computational fluid dynamics (CFD) analysis of liquefied petroleum gas tanks with various types of local heating by fire. The analysis, conducted using the commercial CFD code by Fluent Inc., was performed as part of a larger study of the effects of thermal protection defects.

Thermal protection systems are used to protect dangerous goods tank-cars from accidental fire impingement. They are designed so that a tank will not rupture for 100 minutes in a defined engulfing fire, or 30 minutes in a defined torching fire. Recent inspections have shown that some tanks have significant defects in these thermal protection systems. If a tank has thermal protection defects, it will heat up more rapidly in a fire. This means the tank will experience higher wall temperatures, lading temperatures and tank pressure, all of which increase the risk of tank failure.

Testing shows that when a tank is heated in a fire, the liquid near the wall is heated first. This warm liquid then rises to the liquid surface, where it generates vapour to pressurize the tank. This rising of the warm liquid is called temperature stratification. The CFD analysis was conducted to quantify this process for the case of local tank heating due to thermal protection defects.

The CFD analysis was used to study the effect of defect size and location on the transient heating of the liquid lading in the tank. The results show that heating near the tank ends or near the liquid level on the tank sides causes the most rapid rise in liquid temperatures on the liquid surface. Heating near the tank bottom results in the slowest rise in liquid surface temperature.

The following conclusions were made based on the CFD study:

- i) The CFD simulations compare reasonably well with RAX 201 fire test results.
- ii) Simulations show how the heated liquid rises to the liquid surface, where it stays due to buoyancy.
- iii) Heating near the tank top and ends causes the largest temperature rise on the liquid surface.
- iv) Heating near the tank bottom generates the lowest heating of the tank surface.
- v) The larger the heated area, the hotter the liquid surface.

As liquid surface temperature is believed to be closely related to the tank pressure, the above trends should also apply to tank pressurization.

SOMMAIRE

Le rapport rend compte d'une analyse tridimensionnelle et chronologique, par la méthode de simulation numérique en mécanique des fluides (CFD, *computational fluid dynamics*), de citernes de GPL (gaz de pétrole liquéfié) soumises à divers types de chauffage localisé causé par des flammes. L'analyse, réalisée par Fluent Inc. à l'aide du code CFD offert sur le marché, faisait partie d'une vaste étude portant sur les effets des défauts de protection thermique.

Les systèmes de protection thermique sont conçus pour protéger les wagons-citernes transportant des marchandises dangereuses contre un feu avec flammes enveloppantes susceptible de survenir lors d'un accident. Ils sont conçus de façon à empêcher une citerne de se rompre pendant 100 minutes lorsqu'elle est soumise à un feu en nappe, et pendant 30 minutes lorsqu'elle est soumise à une flamme de chalumeau. Des inspections récentes ont révélé que les systèmes de protection thermique de certaines citernes présentent des défauts importants. Or, une citerne dont la protection thermique est défectueuse chauffera plus rapidement si elle est en contact avec des flammes. Cela signifie que les parois de la citerne et son contenu, de même que sa pression interne, atteindront des valeurs plus élevées, ce qui accentuera le risque de rupture de la citerne.

Les essais révèlent que lorsque des flammes sont appliquées à une citerne, le liquide adjacent à la paroi se réchauffe d'abord. Puis, ce liquide chaud monte à la surface du liquide, où il génère des vapeurs qui mettent la citerne en pression. Cette montée du liquide chaud entraîne ce que l'on appelle la *stratification des températures*. L'analyse par CFD avait pour but d'établir les paramètres de ce processus dans le cas où le chauffage localisé d'une citerne est attribuable à des défauts de la protection thermique.

Cette analyse a permis d'étudier l'effet de l'étendue et de l'emplacement du défaut sur la montée en température du contenu de la citerne. Les résultats indiquent que l'application de chaleur près des extrémités de la citerne ou sur les parois, près du niveau de remplissage, entraîne le réchauffement le plus rapide de la surface du liquide. À l'inverse, c'est lorsque la chaleur est appliquée dans la partie inférieure de la citerne que la température de la surface du liquide monte le plus lentement.

Les travaux ont mené aux conclusions suivantes :

- i) Les résultats des simulations CFD se comparent raisonnablement bien aux résultats des essais RAX 201.
- ii) Les simulations montrent comment le liquide chauffé monte à la surface, où la poussée hydrostatique le maintient.
- iii) L'application de la chaleur près du sommet et des extrémités de la citerne entraîne le plus fort réchauffement de la surface du liquide.
- iv) L'application de la chaleur près de la partie inférieure de la citerne entraîne le plus faible réchauffement de la surface du liquide.
- v) Plus la zone chauffée est étendue, plus la température de la surface du liquide est élevée.

Comme on croit que la température à la surface du liquide est étroitement reliée à la pression interne de la citerne, les tendances constatées ci-dessus devraient aussi s'appliquer à la mise en pression de la citerne.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

This report describes a three-dimensional, time transient computational fluid dynamics (CFD) analysis of a liquefied petroleum gas (LPG) tank-car with various types of local heating by fire. This report is intended for specialists interested in modeling thermally protected tank-cars in fires.

This work has been done as part of the overall study of defects in thermal protection systems of rail tank-cars conducted for the Transportation Development Centre of Transport Canada.

This work was done to study the effects of thermal protection defects, more specifically, the effect of defect size and location on the transient heating of the liquid lading. This CFD analysis was carried out to provide some much needed data on how the liquid in a tank-car heats up when the tank is only heated in local areas, as opposed to being 100% engulfed in fire.

1.1 Background

This CFD analysis was part of an overall program to study thermal protection system defects in rail tank-cars.

Certain dangerous goods tank-cars must be thermally protected so they can survive accidental fire impingement. The requirement for thermal protection systems for pressure tank-cars is specified in CAN/CGSB-43.147-2002, section 15.8, which states:

If a thermal protection system is specified by this standard, the system must be capable of preventing the release of any dangerous goods from the tank car, except release through the pressure relief device, when subjected to the following conditions:

> *(1) A pool fire for 100 min, and*
> *(2) A torch fire for 30 min.*

It is known from field surveys that some operating dangerous goods tank-cars have defective thermal protection systems. With the size of the North American fleet of tank-cars, it is not feasible to fix all of these defects immediately, due to both cost and logistical reasons. This research program was intended to help identify which tanks need immediate attention.

Several published reports have been prepared by A.M. Birk and his team in connection with this issue [1-5].

The work has led us to the point where a computer model has been developed to predict critical thermal protection defect sizes on tank-cars. This thermal model requires some additional data and final validation before it can be used to assess defects in the field.

The following data is needed to further support the theoretical work.

i) Obtain high temperature stress-rupture data (by test) of tank steels, including both old steels and new (to cover the true condition of the tank-car fleet).
ii) Conduct a CFD study to obtain predictions for the pressurization of tanks with fire heating of localized thermal protection defects.
iii) Obtain medium-scale fire test data of tanks with defective thermal protection for final validation of failure times.

This report presents the results of the CFD study only. Reports on the stress-rupture testing and the fire testing of medium-scale propane tanks with thermal protection defects will be published separately.

1.2 Objectives and Scope

The objective of this work was to model local heating of a propane tank to study the internal convection currents in the liquid space of the tank. This was done to estimate the liquid temperature on the top of the liquid. From this temperature the tank pressure can be estimated.

The scope was limited to computer modeling only using a commercial CFD code. The results will be used to formulate and validate a computer model of the complete tank and thermal protection system.

The reader is cautioned that this was a limited scope study, and the results are approximate only. The results should be considered preliminary and for discussion purposes only. A full CFD study of this problem would require considerable resources.

1.3 Summary

It is known from experiments and analytical studies that tanks heated in fires have complex internal flows, including:

i) Free convection in vapour space
ii) Free convection and boiling in liquid space
iii) Liquid and vapour temperature stratification

The liquid temperature in the boundary layer and at the liquid/vapour interface dictates the tank pressure.

This study involved a 3D transient CFD study of the free convection currents in the liquid space of a locally heated railway tank-car. Several different local heating conditions were considered, including top heating, side heating, bottom heating and end heating. From this data a summary graph was produced that showed, in relative terms, the liquid surface temperature rise rate as a function of heating location and the size of heated area as a fraction of the liquid wetted wall area.

The results show that the liquid surface heats up faster if the tank is heated on its ends or near the liquid/vapour interface. This is in general agreement with experiments involving propane tanks heated by fire.

2. METHODOLOGY

The CFD study was conducted using the commercial CFD code by Fluent Inc. This is a finite volume type solver and is well known around the world for being a general purpose CFD analysis package.

The CFD code solves the pressure, velocity and temperature fields in the 3D solution domain by integrating the conservation equations (mass, momentum and energy). In this study we are interested in the free convection flow and temperature patterns that are generated by local heating at the tank wall. The heating rates are similar to what we would expect from a thermal protection defect engulfed in a hydrocarbon pool fire.

The CFD solver works by dividing the domain into finite volumes. The properties in the volumes are then solved simultaneously for all the volumes defined. This is a transient analysis that marches through time. This required that we define a starting point in time or initial condition. At time = 0, the heat flux is applied and the solution is determined at each time step.

2.1 Problem Definition

We want to estimate how quickly a thermally protected tank-car, with thermal protection defects, will pressurize when it is engulfed in fire. We will model this case as a tank-car that has locally heated areas.

The problem was defined as follows:

- i) Horizontal cylinder (D = 3 m, L = 18 m)
- ii) 2:1 elliptical ends
- iii) Filled partially (about 95%) with liquid propane
- iv) Remainder of tank filled with propane vapour
- v) Tank is heated in local areas below the liquid level
- vi) PRV (pressure relief valve) is closed
- vii) In defect areas the wall heat flux is assumed to be 54,000 W/m^2
- viii) In non defect areas the heat flux is assumed to be 0 W/m^2

This is the extreme case of a perfectly thermally protected tank with defects.

We want to determine how the liquid is heated. We know the liquid near the heated wall will rise to the liquid surface where it will remain due to buoyancy. This warm liquid will then spread over the liquid surface. It is believed that this warm layer drives the pressure in the tank. We want to determine the average liquid surface temperature over time so we can estimate the pressure in the tank as a function of time from when the heating began.

2.2 Analysis Approach

The thermal protection defects were modeled as local heated areas on the tank-car primary shell surface. No attempt has been made here to model the details of the steel jacket, thermal insulation and steel wall. A heat flux was applied on the tank surface where a defect was assumed to be. Where there was no defect, the surface was assumed to be adiabatic (perfectly insulated). The applied heat flux is approximately what we would expect in the area of a thermal protection defect.

We were only interested in the brief period of time before the PRV is activated. This is about 2-3 minutes for an unprotected tank and is expected to be about 20-40 minutes for a thermally protected tank.

For a 95% full tank, the mass transfer from the liquid to the vapour space before the PRV is opened is negligible compared to the total liquid mass. Therefore we have ignored this mass transfer. We have also ignored the energy transfer from the liquid to the vapour space since it too is negligible compared to the energy transferred to the liquid.

In this study we have only considered the liquid volume. We ignored the effect of the vapour space on the liquid flow patterns. The liquid surface was assumed to be insulated from the vapour space.

2.3 Solution Steps

The following steps were followed in conducting the analysis:

i) Define domain geometry (i.e. the tank geometry).
ii) Divide the domain into finite volumes (i.e. generate grid)
iii) Assign boundary conditions (heat transfer).
iv) Assign initial conditions (temperature).
v) Define material properties (equation of state, viscosity, thermal conductivity, etc.)
vi) Select fluid model type (viscous).
vii) Define solution time step.
viii) Define convergence criteria (for mass, momentum and energy).
ix) Run solution.
x) Post process solution (plot results).

All of the above were done using the commercial codes by Fluent Inc.

2.4 CFD Model

The geometry of the modeled LPG tank car is shown in Figure 1. A symmetric part of the tank domain (half tank) was used for most of the CFD calculations and the mesh generated in the symmetric part was 220,000 volumes as shown in Figure 2. The full model contains 592,225 volumes as shown in Figure 2B. The positions of experimentally measured transient temperatures measured in the RAX 201 fire test are shown in Figure 3. These are compared with CFD results in section 3.

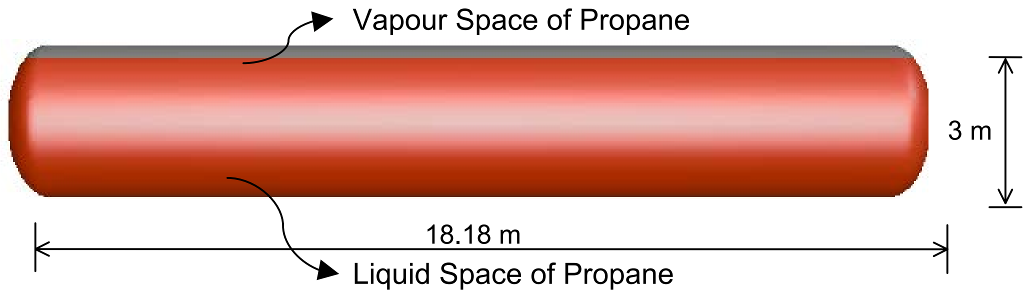


Figure 1: LPG Tank Configuration

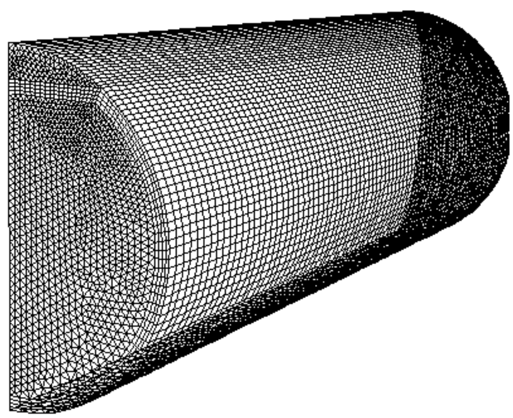


Figure 2: Mesh Configuration (half model)

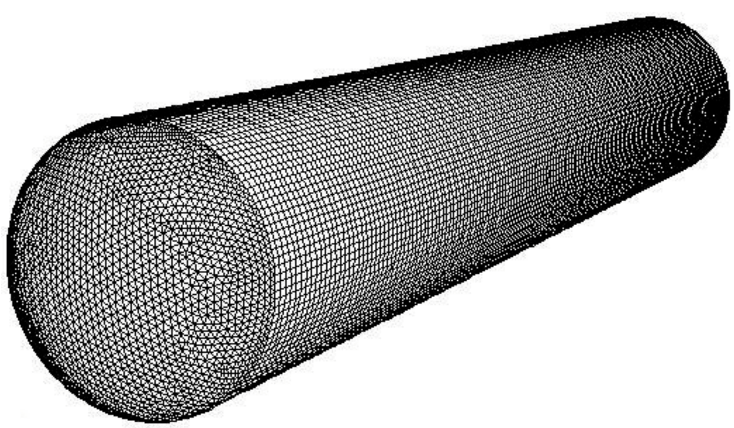


Figure 2B: Mesh Configuration (full model)

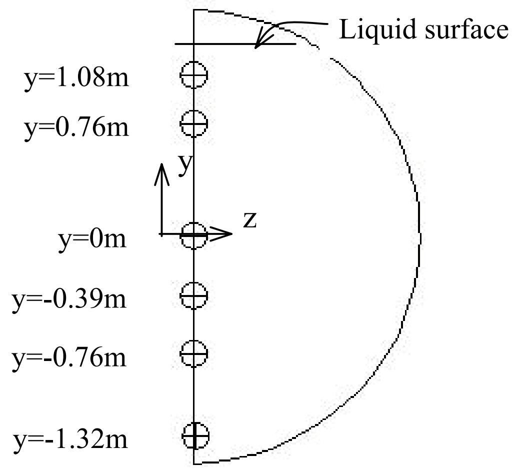


Figure 3: Measurement Points

Both laminar and turbulent flow simulations were investigated. Turbulence was modeled using the standard k-ε model with non-equilibrium wall functions.

The standard k-ε turbulence model is widely used in industry for engineering calculations of turbulent fluid flows. It is not an exact model and there is uncertainty as to how applicable it is for the present analysis. The scope of this study did not allow for an in-depth evaluation of turbulence models for this topic area. The results of this analysis should be considered approximate only. For more information on the details of turbulence models and wall functions the reader is directed to Fluent Inc.'s user manuals.

The temperature-dependent fluid properties used in this work are summarized in Appendix A. The volume of fluid (VOF) model was employed for calculating two-phase flow (i.e. free surface).

Comparisons of the calculations of the laminar and turbulent flows with the experimental data [6] are shown in Section 3 at the points in labeled in Figure 3 on the cross section of LPG tank at 4.64 m from symmetric axis.

We applied a heat flux of 54 kW/m^2 over the local heating points. This is about half of a credible engulfing fire, which is what we would expect in an area of the tank covered with just steel jacket and wetted by liquid on the tank inside wall. The steel jacket acts as a radiation shield and this reduces the heat flux by about half.

3. CFD VALIDATION

The following is a partial and approximate validation of the method. We did not have detailed fire and temperature data to do a full validation. The validation case for the CFD was the RAX 201 full-scale fire test of a 33,000 gal. tank-car filled with propane [7].

The features of the RAX 201 tank are as follows:

i) 125,000 L propane tank, filled to 95% of liquid propane
ii) ID=3 m, ends 2:1 elliptical
iii) Tank L/D = 6
iv) Initial temperature, T = 294°K
v) Steel wall thickness 16.5 mm

For this simulation the heat flux to the liquid was assumed to be 54 kW/m^2. This is about half the expected heat flux for a fully engulfing fire. We are modeling only the first 180 seconds of the heating and it is known that it took a few minutes to get the fire going in the test. If we assume the fire went from 0% to 100% in three minutes, then the average flux would be on the order of 54 kW/m^2. This is obviously an estimate of the fire condition and is for comparison purposes only.

The CFD predictions for the liquid temperatures at the tank section 4.64 m from the mid plain at various distances *y* from the tank centre are shown in Figure 4. The CFD appears to do a good job for the liquid below the tank centreline. The predictions are not as good above the centreline and near the liquid surface before about 100 seconds. After that the predictions are reasonable. Once again the reader is reminded that we did not have any accurate data on the actual heat flux for the first 180 seconds so we are presenting this data for discussion only.

Three-dimensional temperature distributions on tank wall and on symmetric planes are shown in Appendix C.

The time transient CFD calculations have simulated the free convection and temperature stratification in liquid and vapour space. The CFD results with turbulence appear to give the best predictions.

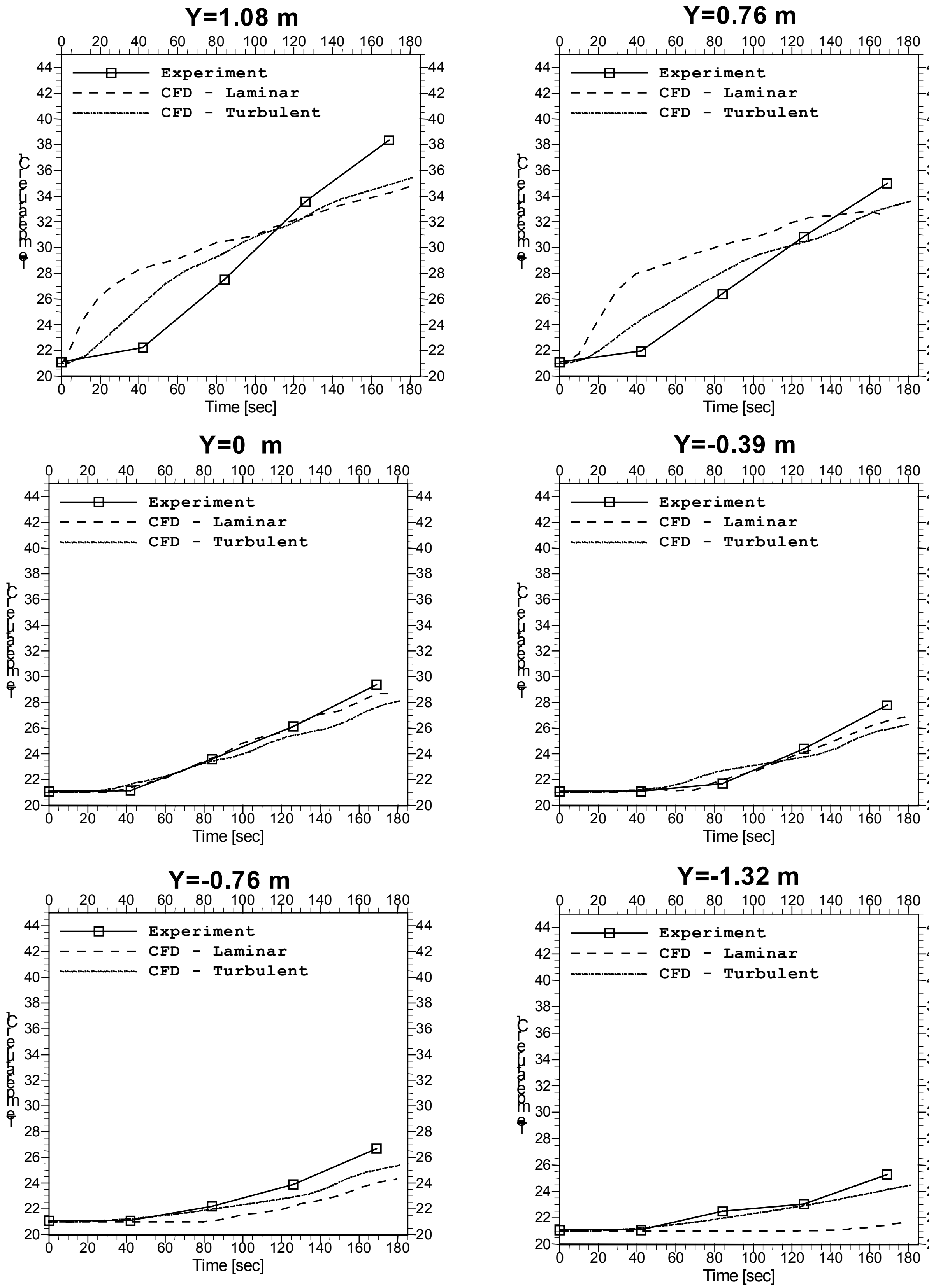


Figure 4: Comparisons of Temperatures Measured and Calculated at Various Liquid Points on the Cross Section of Tank at 4.64 m from Symmetric Axis

4. CFD RESULTS

In this section we model a range of local heating conditions. The following has been assumed:

i) Same tank geometry
ii) 97% fill
iii) Turbulence is modeled
iv) 54 kW/m^2 used for the local heating
v) All other areas assumed insulated

4.1 CFD Study Cases

It total 18 heating cases were considered. They involve heated cells of different size and location. The various cases are summarized in Figures 5 through 8 and Tables 1 and 2.

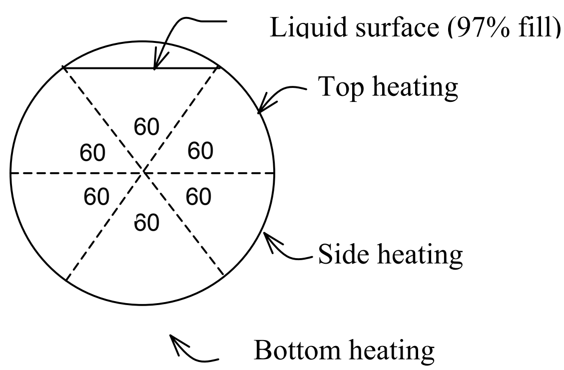


Figure 5: Cross Section of Tank Showing Heated Zones Relative to Liquid Level

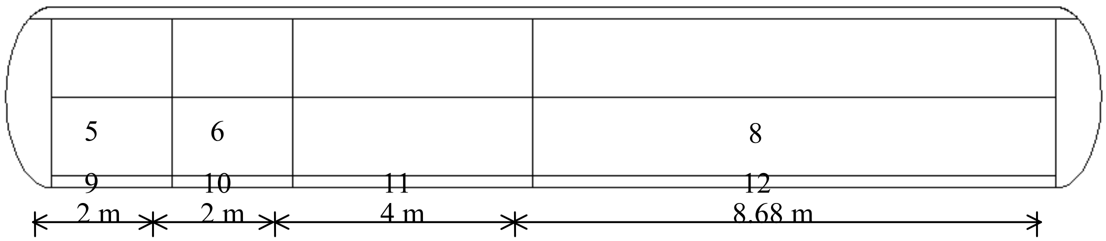


Figure 6: Local Heating Cells (for top, side, and bottom heating cases)

Table 1: Heating cells on CFD study cases of top, side, and bottom heating

case \ cell	1	2	3	4	5	6	7	8	9	10	11	12
1	x											
2	x	x										
3			x									
4	x	x	x									
5	x	x	x	X								
6					x							
7					x	x						
8							x					
9					x	x	x					
10					x	x	x	x				
11									x			
12									x	x		
13											X	
14									x	x	X	
15									x	x	X	x

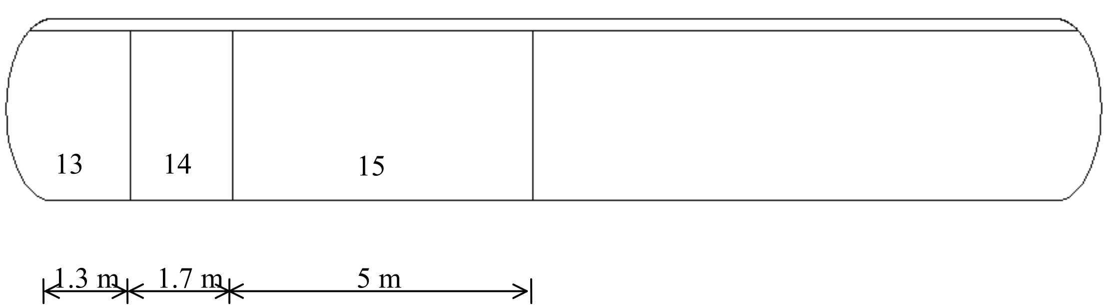


Figure 7: Local Heating Cells (for 10%, 20% and 50% end heating cases)

Table 2: Heating cells on CFD study cases of end heating

cell / case	13	14	15
16	x		
17	x	x	
18	x	x	x

Figure 6 shows the local heating cells whose combinations were used in the CFD study. The cases include top heating, side heating and bottom heating of the liquid space wall as shown in Table 1. There are also end heating cases of 10%, 20%, and 50%, which are shown in Figure 7 and their cell combinations are in Table 2.

Configuration of each CFD study case is displayed in Figures 8 through 11.

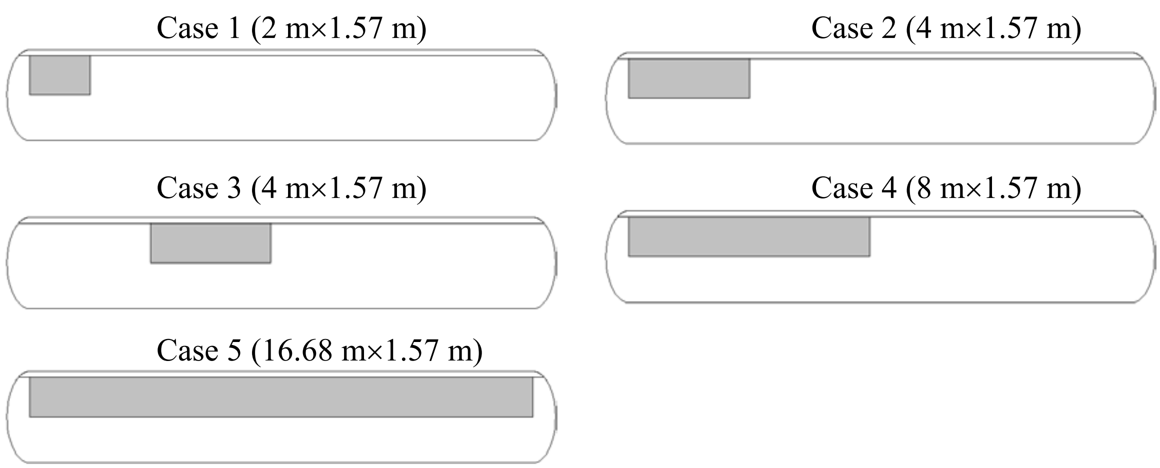


Figure 8: Top Heating Cases

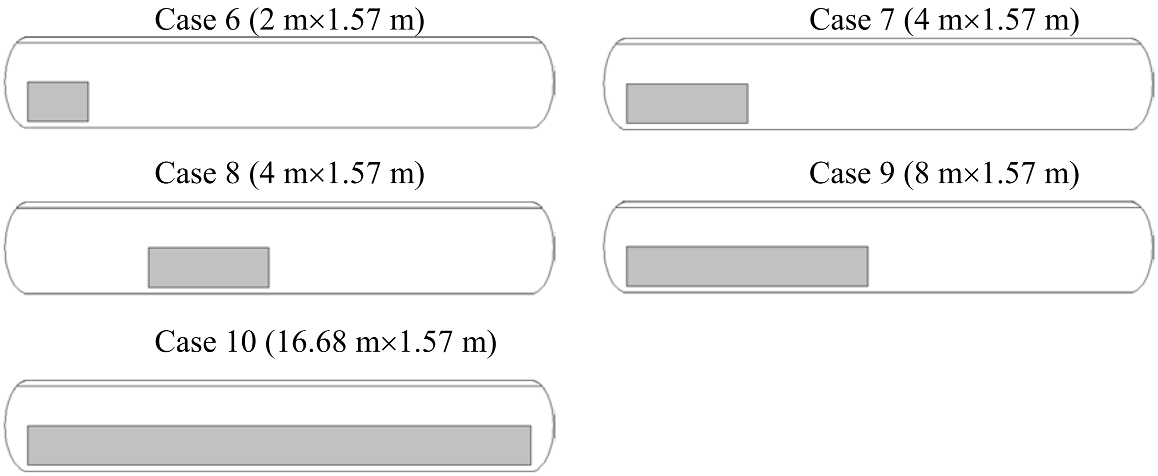


Figure 9: Side Heating Cases

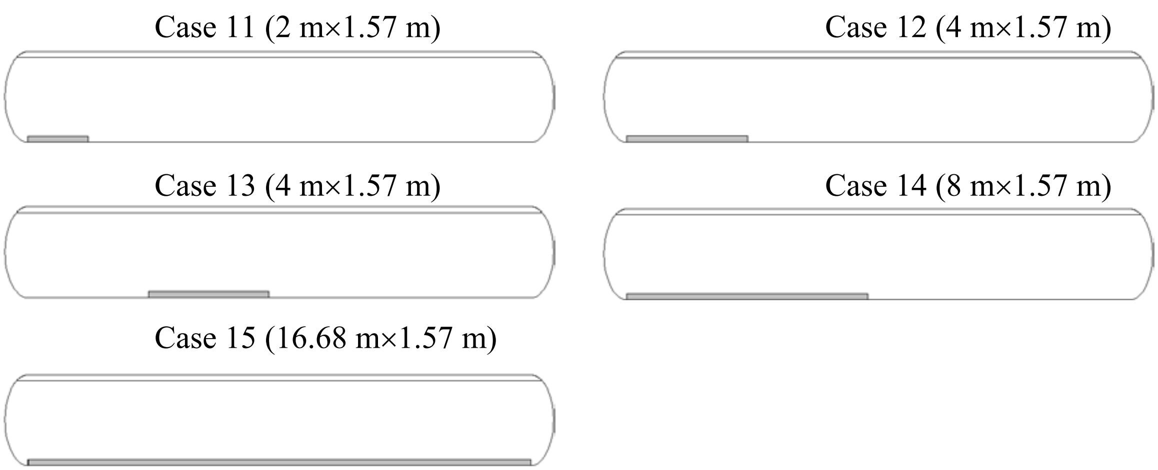


Figure 10: Bottom Heating Cases

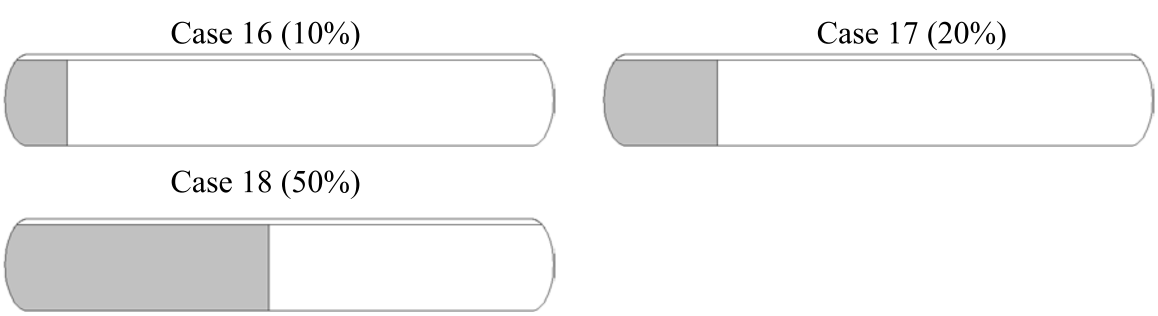


Figure 11: End Heating Cases

4.2 CFD Temperature Results

Sample points on the liquid surface are identified in Figure 12. Sample temperatures vs time are presented in Figure 13 for various run cases. Temperature distributions on the tank liquid wetted wall and liquid surface and on a tank cross section are displayed graphically in Appendix D.

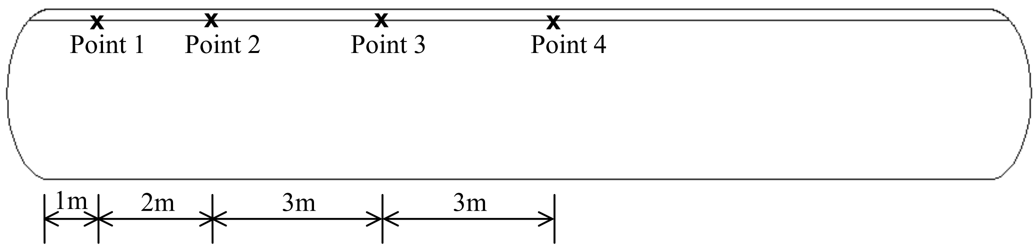


Figure 12: Measurement Points of CFD Data

As expected, the following is observed:

i) Highest temperatures are seen for largest heated areas
ii) Warm liquid rises to liquid surface and disperses over the surface
iii) Highest temperatures for top heated and end heated
iv) Lowest temperatures for bottom heated

Temporal variations of temperatures at the specified points on the liquid surface with the different size of end heating, 10%, 20%, and 50% of tank surface area, are shown in Figure 10. The end heating also reveals the same trends of the top, side, and bottom heating such that the larger the heating area, the higher the temperature.

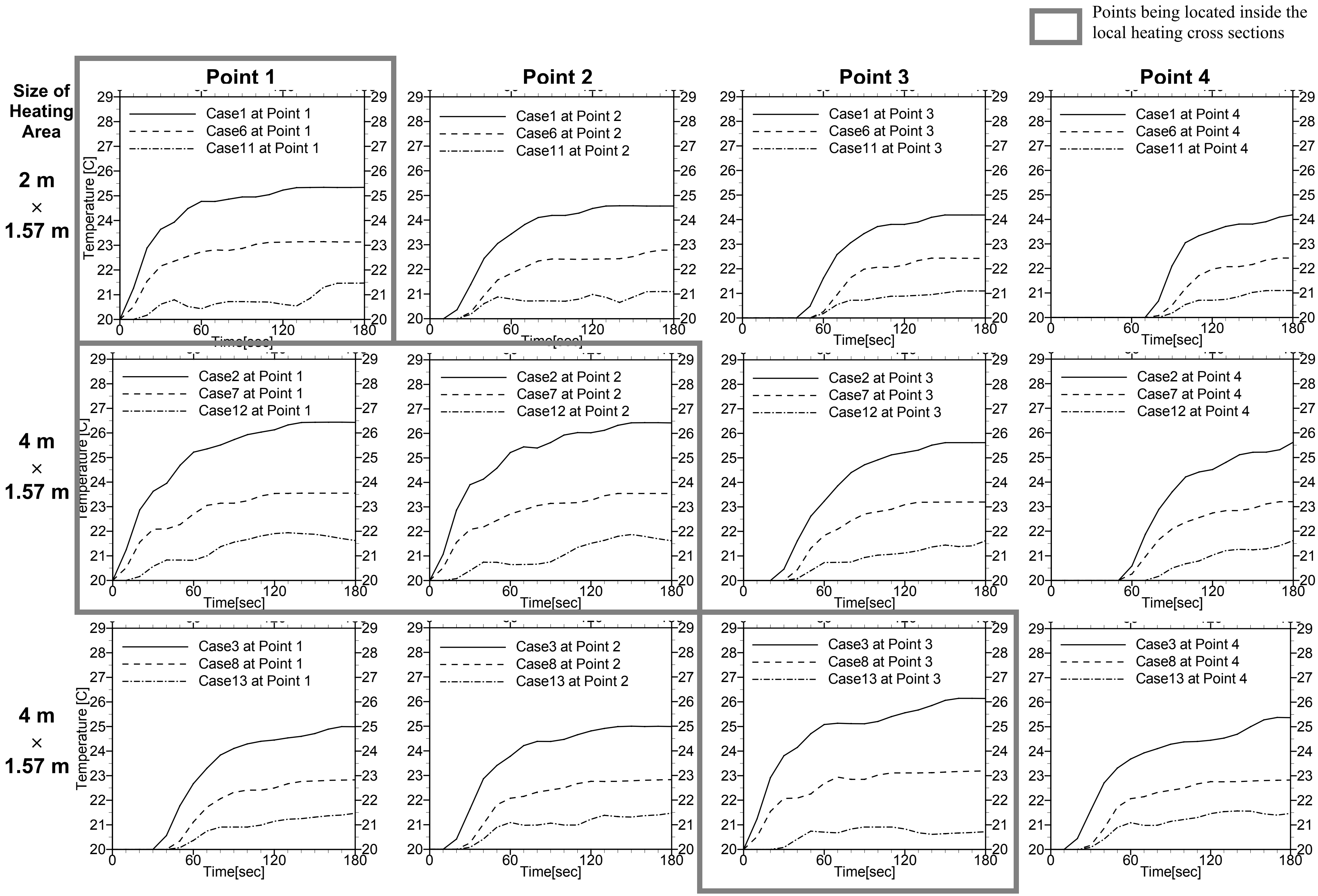


Figure 13: Temporal Variation of Temperatures at Specified Points on Liquid Surface of Top, Side, and Bottom Heating Cases

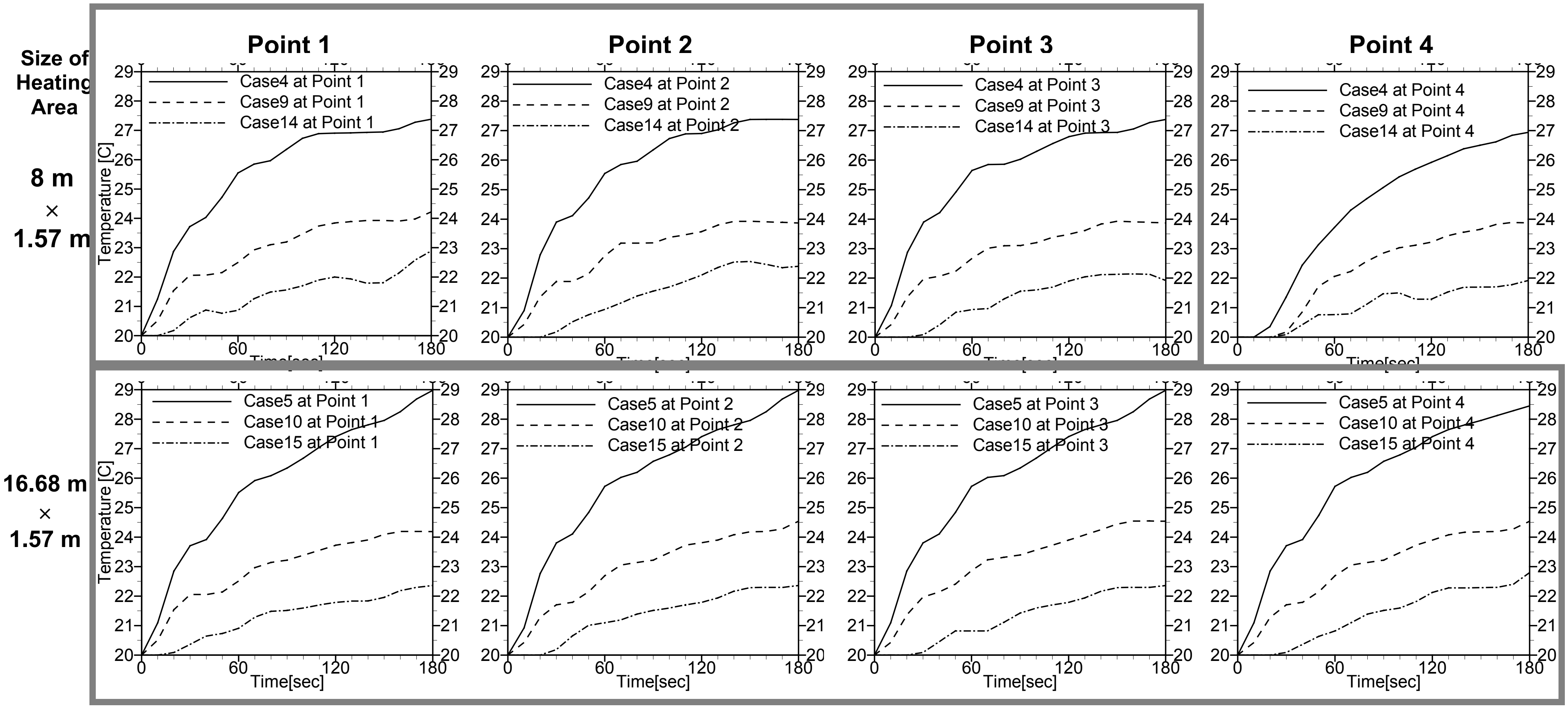


Figure 13: Temporal Variation of Temperatures at Specified Points on Liquid Surface of Top, Side, and Bottom Heating Cases (continued)

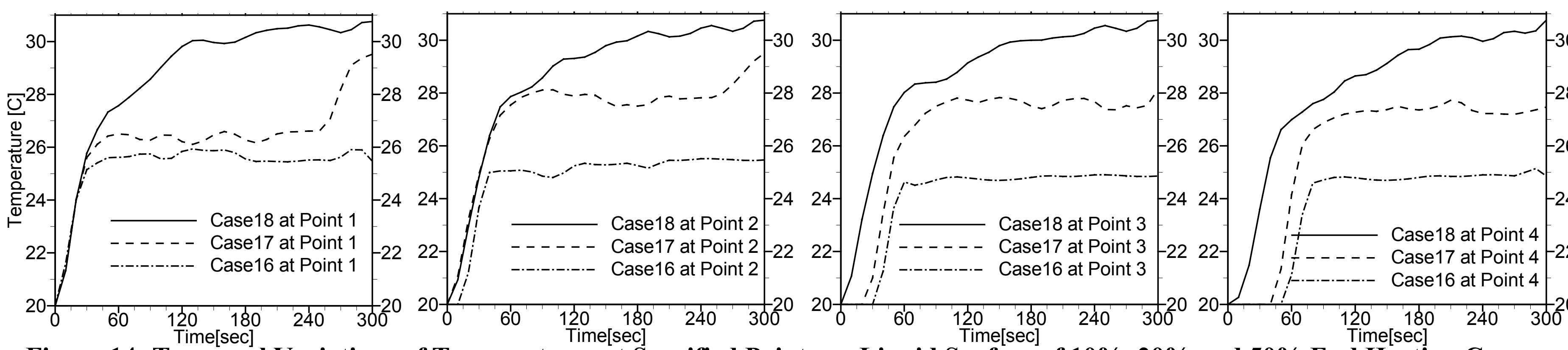


Figure 14: Temporal Variations of Temperatures at Specified Points on Liquid Surface of 10%, 20% and 50% End Heating Cases

4.3 CFD Result Summary

In this work we are interested in how a tank will pressurize when local areas on the tank surface are heated. We will estimate the tank pressurization rate from the average liquid surface temperature. The CFD analysis allows us to compare the different heating cases in terms of the liquid surface temperature. The temperature ratio β is defined as follows:

$$\beta = \left[\frac{T_{defect-local} - T_{init}}{T_{full.defect} - T_{init}} \right]_t$$

where

T = average liquid surface temperature at time t
defect local = local defect case (i.e. 54 kW/m^2)
full defect = 100% defect = entire tank liquid wetted surface has defect (i.e. 54 kW/m^2)
init = initial condition

The CFD results of the 18 local heating cases shown in Figure 13 and 14 are shown in Figure 15. The ratio β relates the predicted surface temperature for the various heating cases relative to 100% heating case. A β = 0.5 means the temperature on the surface rises half as fast as the 100% defect case. From this we believe that the tank pressure also rises half as fast as the 100% defect case.

We see the following in Figure 15.

i) The larger the area of tank insulation defects, the higher the average liquid surface temperature at 180 s.
ii) The closer the heating is to the free surface, the higher the surface temperature. This makes sense because the liquid boundary layer has the shortest distance to rise before it reaches the liquid surface.
iii) End heating is similar to heating near the liquid surface.
iv) The lowest liquid surface temperature is achieved with heating at the tank bottom. This makes sense because the liquid plume rises though the liquid core and the heat is mixed more into the liquid core.

These trends agree with observations from fire tests of 500 gal. tanks by Birk et al. [8].

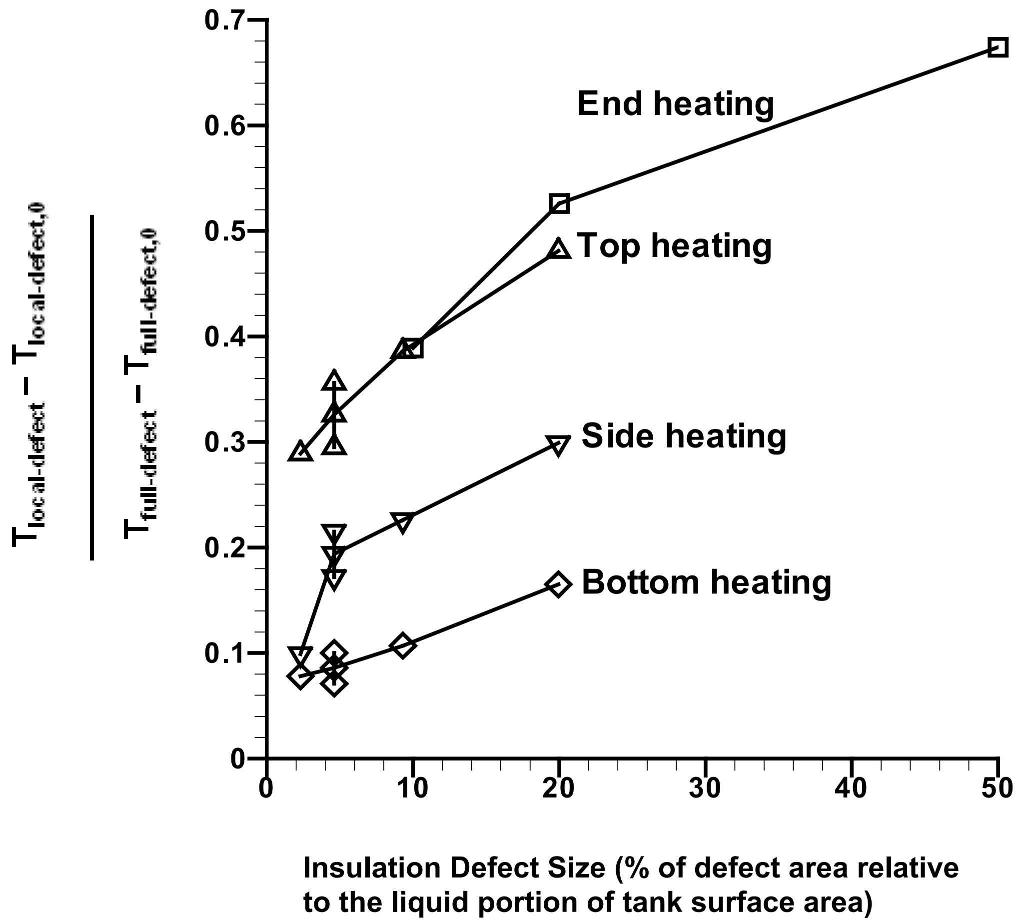


Figure 15: Liquid Surface Temperature Ratio as a Function of Locally Heated Area

4.4 Use of Results

These results are approximate only and have not been validated. We have used them to make crude estimates of tank pressurization with thermal protection defects.

As an example, let us consider the case of a tank with 15% thermal protection defect that covers the tank side from bottom to top. From the graph we see $\beta = 0.26$ for 15% side heating. This means the liquid temperature at the surface rises about 26% as fast as the 100% defect case.

We now assume this also applies to pressure. We know an unprotected tank filled to 95% capacity with propane will pop its PRV in about 2-3 minutes. The same tank covered with jacket only (100% defect) should pop its PRV in about 4-9 minutes because the jacket reduces the heat flux by about half. With $\beta = 0.26$ we would expect the PRV to pop in about (4 to 9)/0.26 = 15 to 35 minutes.

This is obviously an extrapolation and is an estimate only. We do not have data to confirm this for a tank-car.

5. CONCLUSIONS

The following conclusions were made based on the CFD study:

- i) The CFD simulations compare reasonably well with RAX 201 fire test results.
- ii) Simulations show how the heated liquid rises to the liquid surface where it stays due to buoyancy.
- iii) Heating near the tank top and ends causes the largest and fastest temperature rise on the liquid surface.
- iv) Heating near the tank bottom generates the lowest heating of the tank surface.
- v) The larger the heated area the hotter the liquid surface.

It is believed that the liquid surface temperature drives the tank pressure. Therefore the above trends should also apply to tank pressurization.

These results appear to agree with the results from fire tests of 500 gal. tanks by Birk et al. [8]. In those tests, tanks heated by fire near the liquid surface pressurized faster than tanks heated near the tank bottom.

Further work is needed to verify that these results are generally applicable to tank-cars.

REFERENCES

1. Birk, A.M., VanderSteen, J.D.J., Burner Tests on Defective Thermal Protection Systems, Transport Canada Report TP 14066E, Transportation Development Centre, March 2003.

2. Birk, A.M., Review of AFFTAC Thermal Model, Transport Canada Report TP 13539E, Transportation Development Centre, January 2000.

3. Birk, A.M., Cunningham, M.H., Tank-Car Insulation Defect Assessment Criteria: Thermal Analysis of Defects, Transport Canada Report TP 13518E, Transportation Development Centre, October 1999.

4. Birk, A.M., Thermographic Inspection of Tank-Car Insulation: Field Test Manual, Transport Canada Report TP 13517E, Transportation Development Centre, October 1999.

5. Birk, A.M., Cunningham, M., Thermographic Inspection of Tank-Car Thermal Insulation, Transport Canada Report TP 13203E, Transportation Development Centre, March 1998.

6. Anderson, C., Townsend, W., Zook, J., Cowgill, G., The Effects of a Fire Environment on a Rail Tank Car Filled with LPG, Report No. FRA-OR&D 75-31, Federal Railroad Administration, U.S. Department of Transportation, 1974.

7. Townsend W., Anderson C.E., Zook, J., Cowgill, G., Comparison of Thermally Coated and Uninsulated Rail Tank-Cars Filled with LPG Subjected to a Fire Environment, Report No. FRA-OR&D 75-32, Federal Railroad Administration, U.S. Department of Transportation, 1974.

8. Birk, A.M., Vandersteen, J.D.J., Davison, C.R., Cunningham, M.H., Mirzazadeh, I., PRV Field Trials: The Effects of PRV Fire Condition and PRV Blowdown on Propane Tank Survivability in a Fire, Transport Canada Report TP 14045E, Transportation Development Centre, January 2003.

Appendix A: Temperature Dependent Physical Properties of Liquid and Vapour Propane

$$\text{Physical Property} = C_0 + C_1T + C_2T^2 + C_3T^3$$

1. Liquid Propane

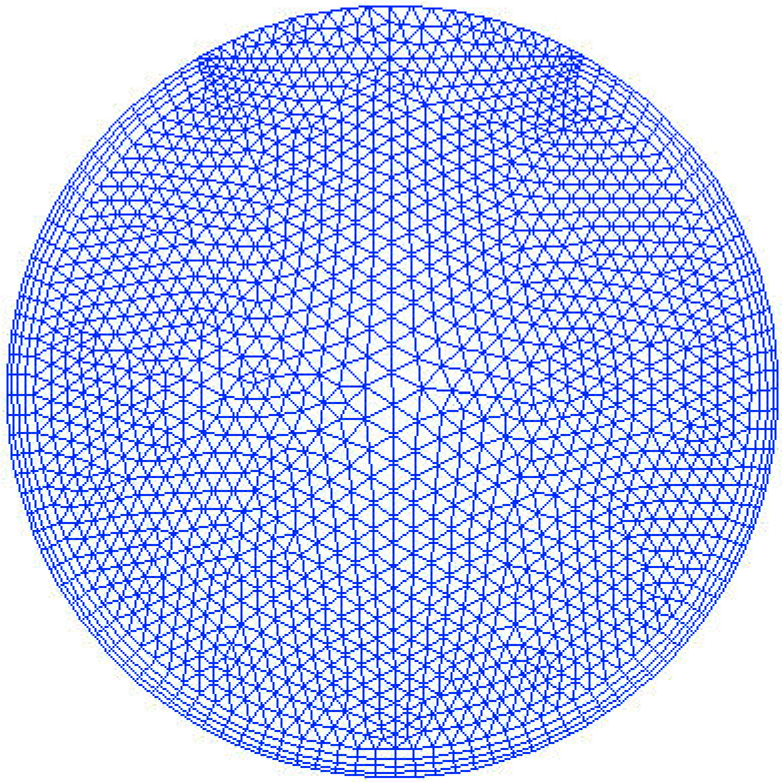
	C_0	C_1	C_2	C_3
Density, ρ [kg/m3]	5648.3	-49.8	0.165018	-0.00018782
Specific heat, Cp [J/kg-K]	-565457	5610.275	-18.4481	0.0202061
Thermal conductivity, k [W/m·K]	0.960851	-0.007497	2.26E-5	-2.41336E-8
Viscosity, μ [kg/m·s]	0.001828	-1.4E-5	3.91711E-8	-3.88228E-11

2. Vapour Propane

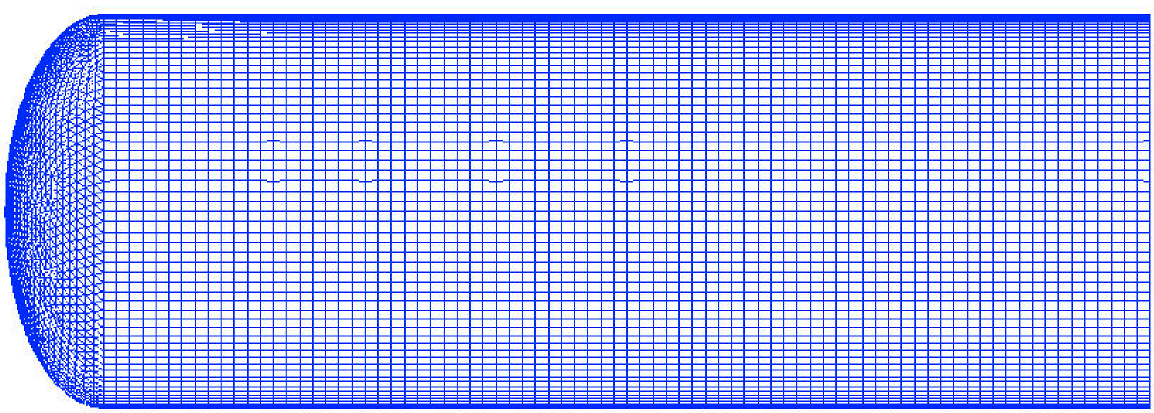
	C_0	C_1	C_2	C_3
Density, ρ [kg/m3]	-5278.7	53.275	-0.179943	0.000204196
Specific heat, Cp [J/kg-K]	-1018629	10062.27	-33.01038	0.036043065
Thermal conductivity, k [W/m·K]	-2.078094	0.020729	-6.86135E-5	7.60771E-8
Viscosity, μ [kg/m·s]	-3.83308E-4	3.87667E-6	-1.28864E-8	1.43879E-11

* Temperatures in above formula are used in Kelvin.

Appendix B: Mesh Configuration

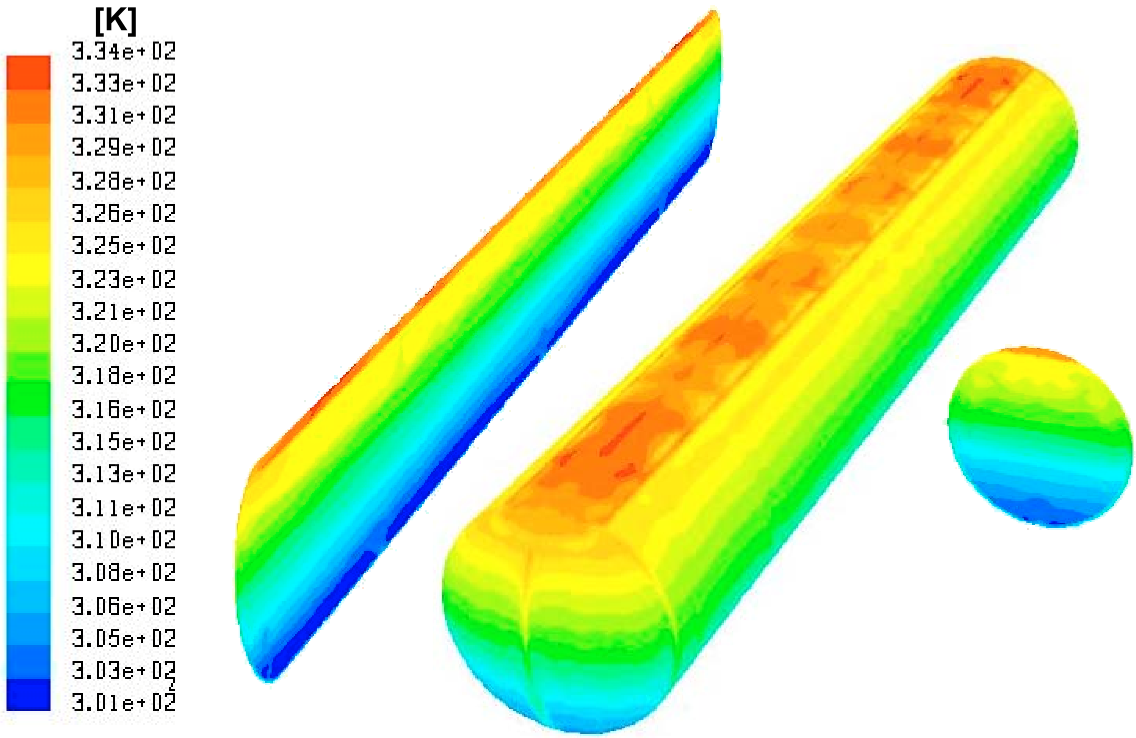


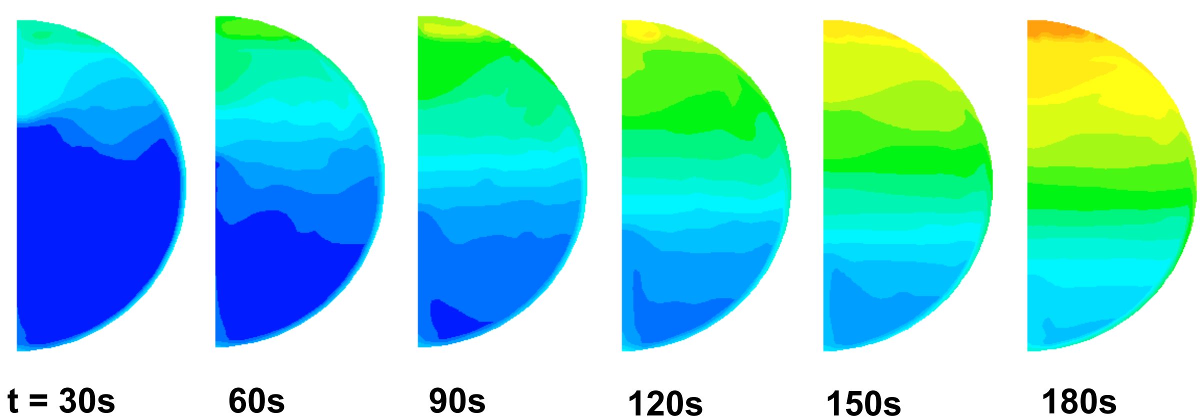
Mesh configuration on tank cross section



Longitudinal mesh configuration

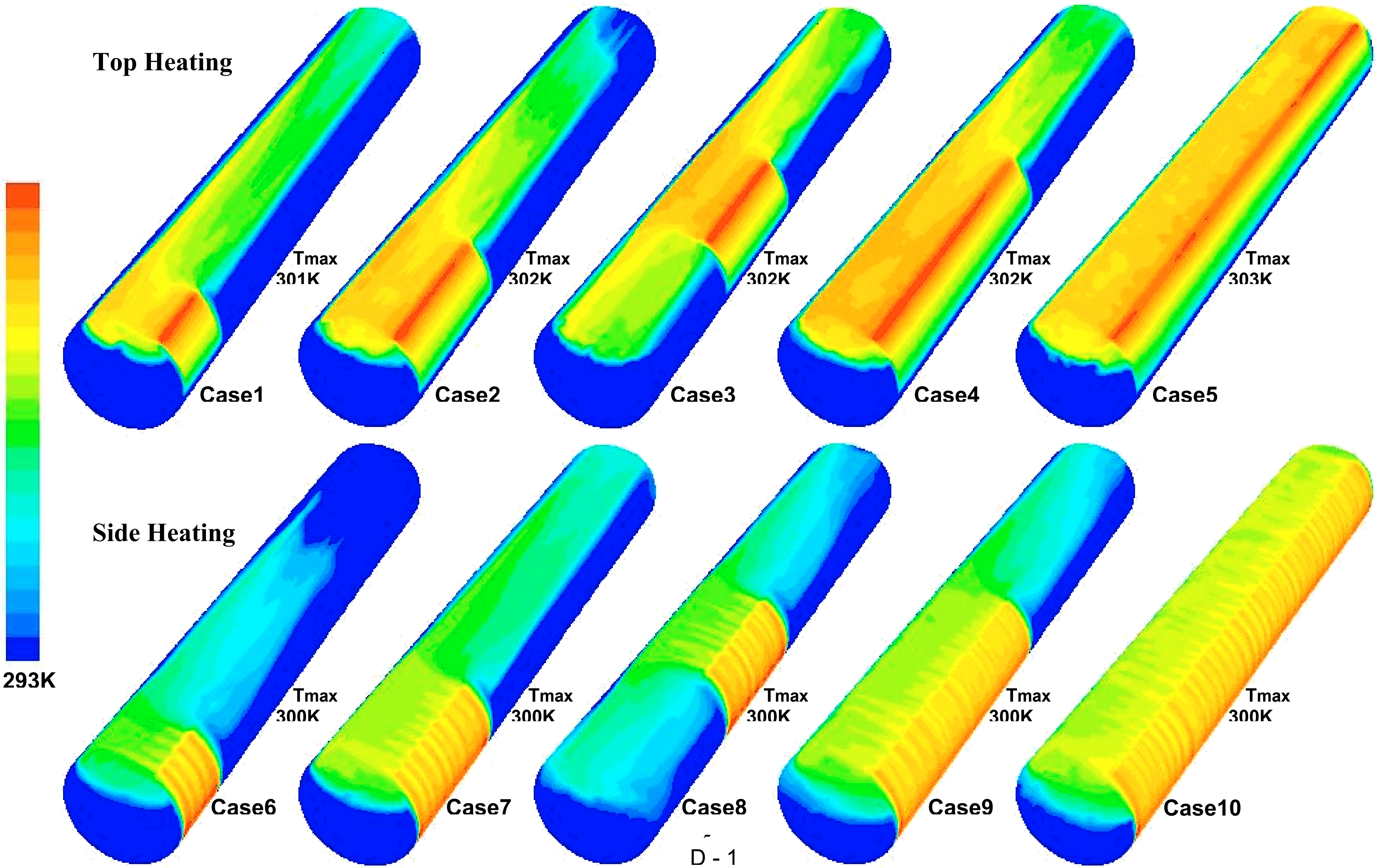
Appendix C: Temperature Distribution in an Engulfing Fire

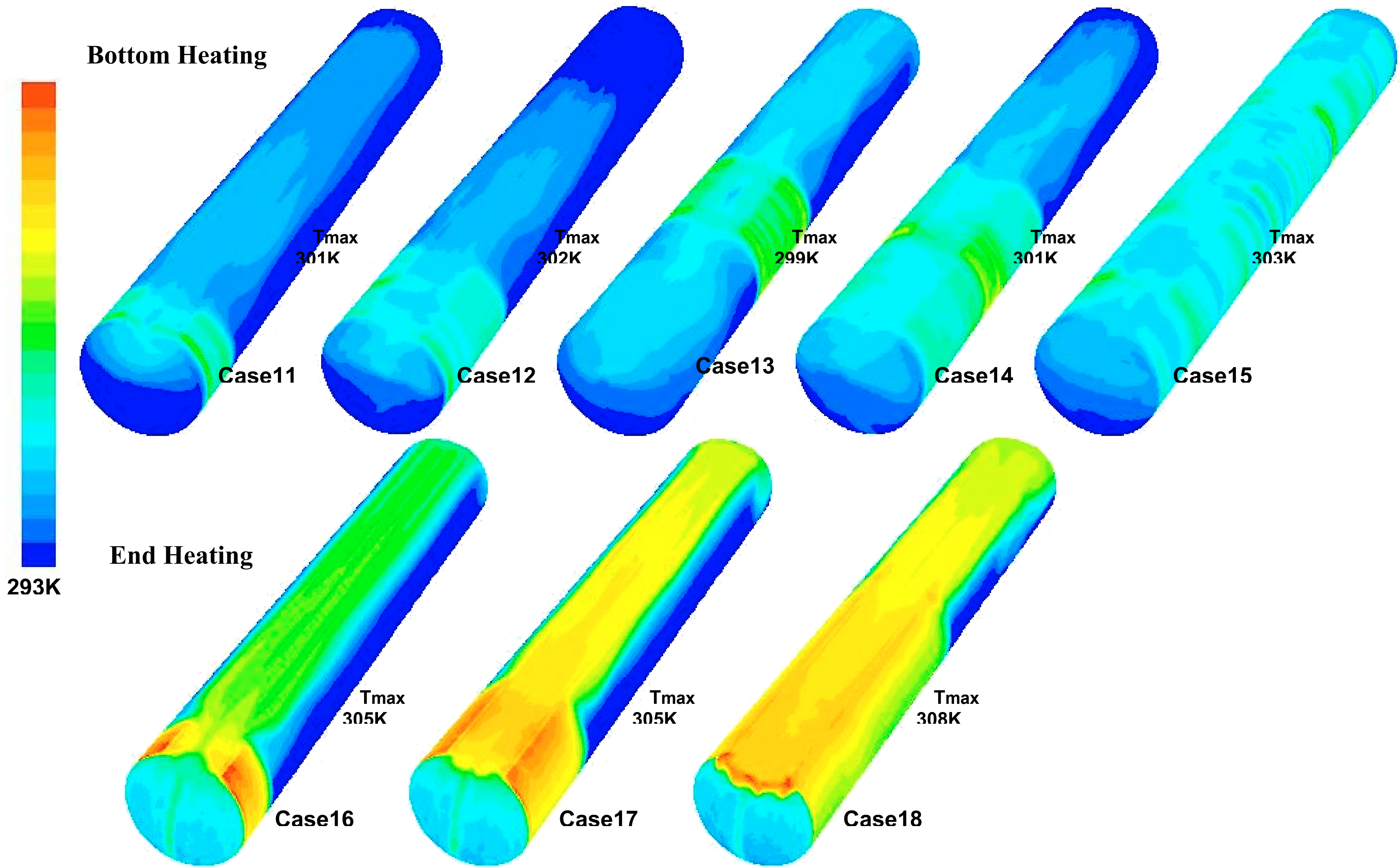




Static (at t=210s) and transient temperature distributions are simulated with heat fluxes of 108 kW/m^2 on liquid space wall and 50 kW/m^2 on vapour space wall. The radiation heat transfer in the vapour space is modeled by Discrete Ordinate radiation model.

Appendix D: Temperature Distributions on Liquid Walls and Free Surfaces of Various Insulation Defect Tanks in 180 seconds




Bottom Heating
End Heating
293K
Tmax
301K
Case11
Tmax
302K
Case12
Tmax
299K
Case13
Tmax
301K
Case14
Tmax
303K
Case15
Tmax
305K
Case16
Tmax
305K
Case17
Tmax
308K
Case18