PAPER • OPEN ACCESS

Applicability of one-body model to predict thermal distribution in spent nuclear fuel pool

To cite this article: Husaini Roslan *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **555** 012004

View the article online for updates and enhancements.

Applicability of one-body model to predict thermal distribution in spent nuclear fuel pool

Husaini Roslan[1,a)], Nahrul Khair Alang Md Rashid[1], Muhammad Arif Sazali[1]

Department of Energy Engineering, Faculty of Chemical and Energy Engineering, Universiti Teknologi Malaysia, 81300 Johor Bahru, Johor, Malaysia.

[a]husainiroslan93@gmail.com

Abstract. An assumption system with a one-body model of spent nuclear fuel (SNF) assemblies was developed to predict the thermal distribution in the spent fuel pool (SFP) in the absence of external cooling system. It was based on the three-dimensional (3D) thermal hydraulic behaviour computed using the computational fluid dynamic (CFD) software, Fluent 18.0. This study aims to determine the applicability of the one-body model (average decay heat) by comparing it with the individuals-body model (variation of decay heat) computation results. The thermal distribution for both models were compared and evaluated, neglecting other effects such as position of the SNF assemblies, decay heat, and axial heat flux profile. It was found that the one-body model was applicable to predict the thermal distribution of the SFP during loss of active cooling system with error less than 1˚C.

1. Introduction

Spent fuel pool (SFP) is a type of storage to cool spent nuclear fuel (SNF) unloaded from nuclear reactors. The SNF is loaded into the pool immediately to reduce the radioactive decay heat and radiation level [6]. SFP usually equipped with storage rack assemblies to hold the SNF rods. To cool the SNF and to ensure enough decay heat removal, SFP is generally designed with external active cooling system. However, this cooling system will stop working if there is a loss of electrical supply. As the cooling capability of the SFP is unavailable, the temperature of SFP water and SNF will increase continuously [8]. Therefore, it is important to evaluate the thermal behavior of the SFP during a loss of cooling accident.

The accident in Fukushima Daiichi Nuclear Power Plant in March 2011 had proven the importance of consistent monitoring on the thermal behavior of SFP. There is also an increase in awareness to improve the safety level of SFP when there is a loss of active cooling system [10]. Many researches have been done to investigate the thermal behavior of the SFP and to improve the safety level of the SFP during loss of cooling accident. Yanagi *et al*. [1-4] have developed a system to predict the thermal distribution in the SFP during loss of cooling accident. They carried out a series of studies comprising of prediction on decay heat, calculation of heat flux to the air, and three-dimensional (3D) hydraulic model of the SFP using computational fluid dynamic (CFD) software Fluent 18.0.

Since the CFD solving the conservation equation has become more efficient and robust due to the tremendous advancements of both software and hardware technologies in the past decade, it has becoming a useful tool for engineering design and analysis [6]. CFD method also has become one of the most popular methods to predict the thermal behavior of the SFP. However, one of its disadvantages is that 3D computation requires a long computing time. Many researches had been
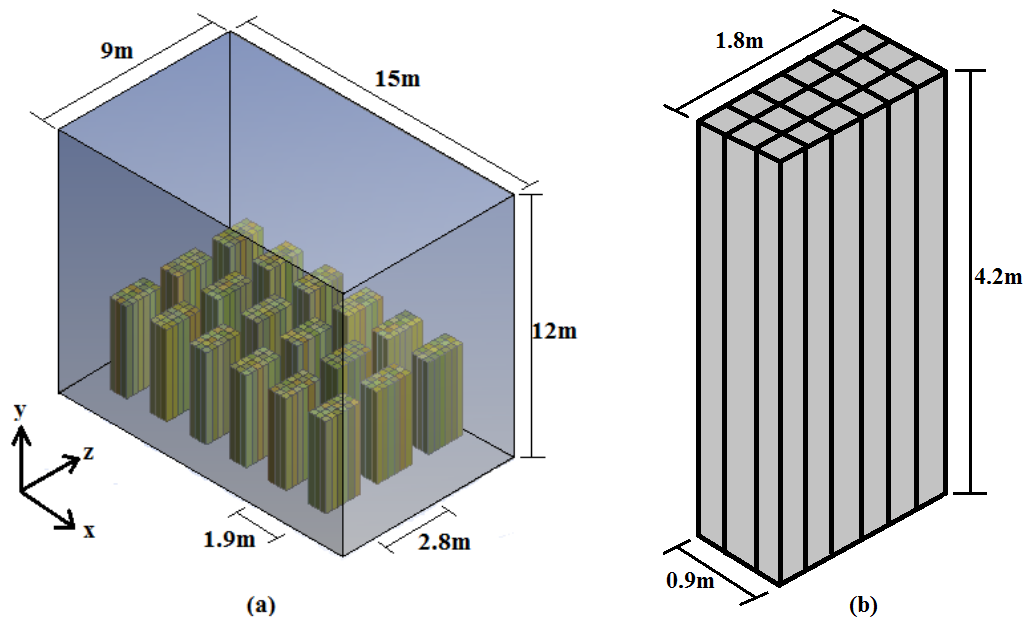
conducted to investigate the applicable model to reduce the computing time and modelling complexity. Yanagi *et al.* [1] had developed one-region model to predict thermal behavior of the SFP. They also investigated the effect of computing parameters such as the mesh size and the timestep size. Based on the previous works, the authors had confirmed the applicability of the one region model to predict thermal behavior of the SFP. They also found that the mesh size and the timestep size had low effect on the result of the computation. Therefore, other computation parameters could be optimized to reduce the computing time.

The current study aims to investigate the applicability of the one-body model of the SNF assemblies on predicting the thermal distribution in the SFP during loss of cooling accident using computational fluid dynamic software, Fluent 18.0. This study compares the measured point temperature from both one-body assemblies model (average decay heat) and individual-body (variation of decay heat) assemblies model.

2. Materials and Method

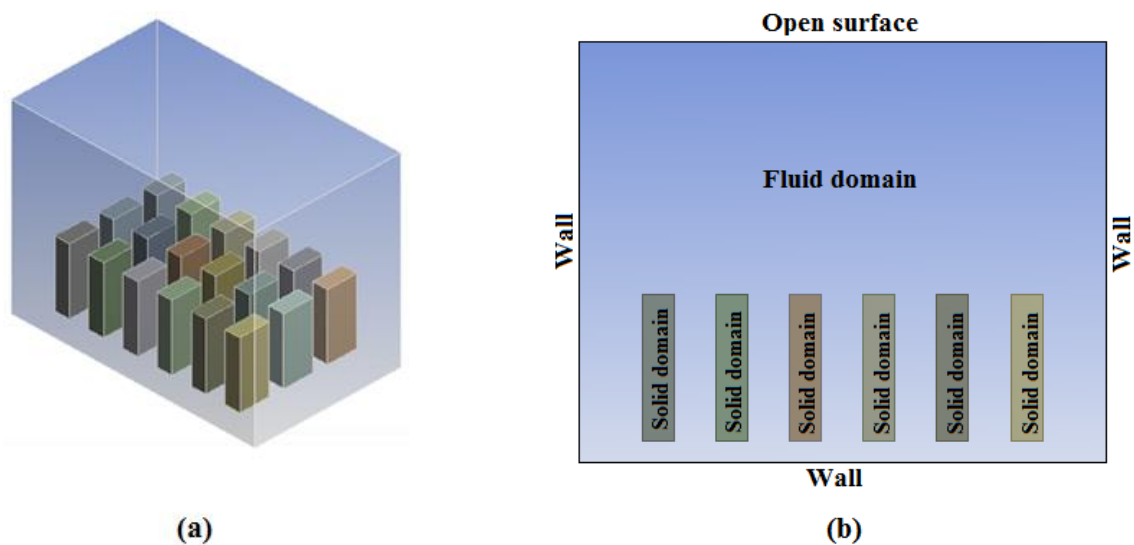### 2.1. Computational Model Development

The conceptual design of SFP established by Yanagi *et al.* [4] is used to develop the computational model of SFP which is shown in Figure 1(a). The dimensions of the SFP are 15 m (W) × 9 m (L) × 12 m (D). The elevation of the SNF is 0.5 m from the bottom of the SFP. The distance between each SNF rack (center to center) are 1.9 m and 2.8 m for x and z direction respectively. The water surface level is 7 m above the top of the SNF assemblies which functions as gamma radiation shielding [3]. No external active cooling system is involved in this study.

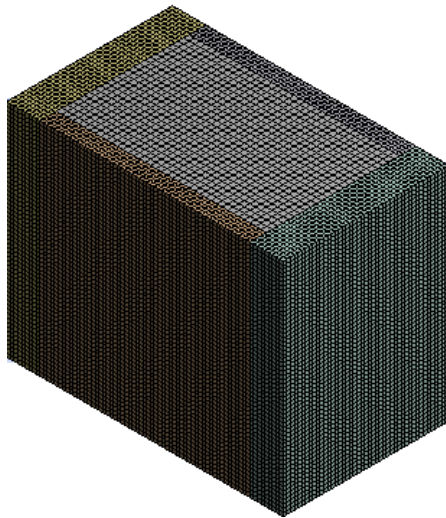**Figure 1.** Conceptual design of SFP [4].

There are 18 SNF racks in the SFP in which each rack contains 18 cells and each cell stores 17x17 SNF rod assemblies. The dimension of a single SNF rack are 1.8 m (W) × 0.9 m (L) × 4.2 m (D) as shown in Figure 1(b). To reduce the computation complexity and errors during modelling of the real SFP, some geometry simplification usually was made. Based on previous studies, all SFP models were described as individual-body model in which each SNF cells was treated as an individual body source with respective decay heat value, regardless the number of the SNF rod. In this study, the effect of decay heat distribution in the SFP was evaluated by using the one-body model, in which all the SNF

cells in each rack were treated as one body source with average decay heat value. Figure 1 shows the computational design of the SFP loaded with individuals-body assemblies, while Figure 2 (a) consists of one-body SNF assemblies. Figure 2(b) shows the side view of the computational design of SFP loaded with one-body model. For the individuals-body model, there were 18 assemblies of SNF rack loaded with 18 SNF cells. The dimensions of each SNF cells are 0.3 m (W) × 0.3 m (L) × 4.2 m (D) (refer Figure 1(b)). The modeled SFP were also assumed to have SNF assembly in the form of SNF cell box placed in the rack where there are no holes or gap in the cell rack. The value of decay heat generated from each SNF cells box was varied from 5MW to 15MW. For the one-body model, there was also 18 assemblies of SNF racks, but each SNF racks was treated as one body source with averaged decay heat value which is 10MW.
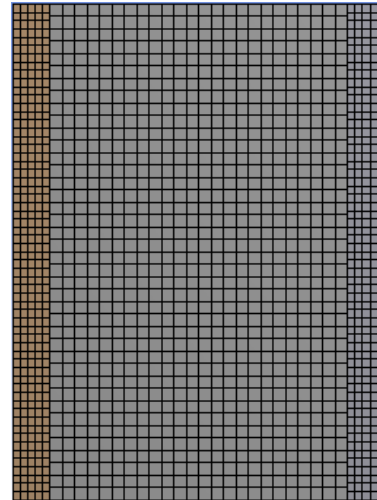

Figure 2. Computational design of SFP loaded with one-body assemblies.

Figure 3 and Figure 4 show the computational grid model and cross-sectional view of the grid model respectively. They were applied to investigate the thermal hydraulic behavior of the SFP. Uniform structured mesh was used for all body involved in this simulation. The maximum size of the mesh cells was about 0.3 m. Different name selections such as fluid domain, solid domain, wall domain, and open domain were assigned to each body and surface as shown in Figure 2(b). The total number of computation cells was set to about 72,000 cells with 0.2 m mesh size near the wall to captured relevant turbulence flow and 0.7 orthogonal quality.

Figure 3. Computation grid model.

Figure 4. Cross sectional view of computational grid model.

2.2. Computation condition and solution

This research deals with thermal and fluid flow problems. Therefore, the boundary condition had been set up based on the appropriate parameters. The computation model approach and boundary condition used in this simulation were summarized in Table 1. To model the fluid flow field in the SFP, standard k-epsilon turbulence model was used with scalable wall function as the model walls treatment. This turbulence model was used because there were regions in the SFP where Rayleigh number could be high [5]. The wall of the pools was set with non-slip condition and standard wall function in which zero heat flux passes through. The heat transfer between the water and concrete wall was neglected since the heat loss to concrete is small because of its low conductivity and only 10% of the heat is loss to the air [2]. For conjugate heat transfer part between solid (SNF) and fluid (water), both the interfaces were set up as coupled wall. To calculate the natural convection force, and others temperature dependent parameters, the polynomial function based on standard thermodynamic properties of saturated water was used. The transient simulations were done for both models. Time step size was set up at 10 seconds with more than five hours of simulation real time.

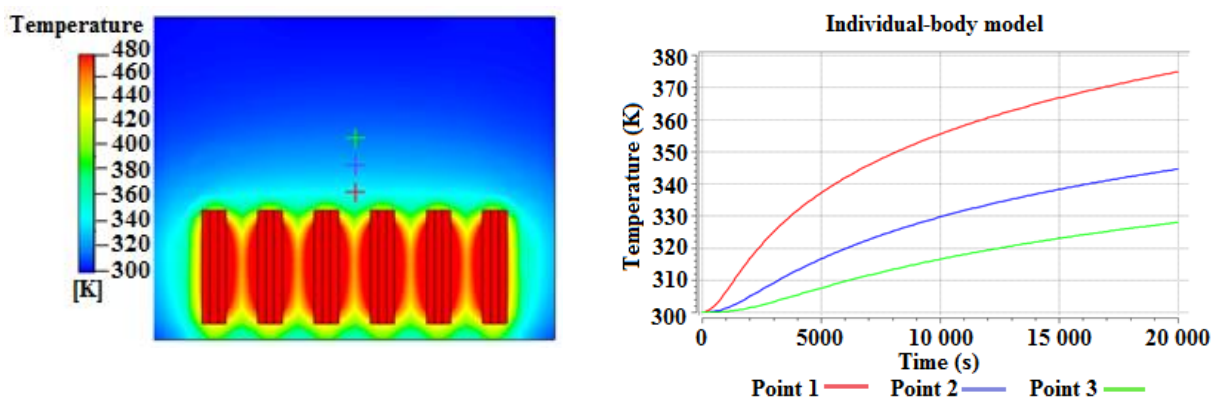**Table 1.** Computation parameters and description

Computation Parameters	Descriptions
1. Turbulent model	Standard k-epsilon model
2. Convective term	First order upwind
3. Bodies condition	Pool: fluid medium,
	SNF: Source term in solid state
4. Boundary condition	Top surface: Ambient pressure (1atm) and temperature (300K)
	Pool bottom and side wall: Stationary with no slip condition
	Pool and SNF interface: Coupled wall
5. Solution Discretization Method	Energy: Second order upwind
	Momentum: Second order upwind
	Turbulent kinetic energy: First order upwind
	Turbulent Dissipation rate: First order upwind
6. Calculation	Time step size: 10s
	Number of timesteps: 10 000
	Recursion Termination: Temperature of water reach 373K

In terms of decay heat generation, all rack assemblies and individual cells assembly were set up as volumetric heat source. The standard solid zircaloy properties were used to define the properties of the heat source. According to Yanagi *et al*. [2], decay heat has no significant effect on the relationship between the average temperature and the surface temperature, but it affects the point temperature and the computation time. The smaller the decay heat, the higher the computing time. In addition, the maximum difference of water temperature for different SNF arrangement was about 4˚C when there is no external active cooling system [9]. Therefore, simulations for one-body model were conducted in which average of 10 MW decay heat were applied to all SNF racks regardless the arrangement of the SNF cells. For individual-body model, the decay heat varies from 5MW to 15MW randomly. Each rack contains nine cells which have decay heat value below 5MW, and the other cells have decay heat values below 15MW. The heat source was set at the center of each SNF cells and racks for individual-body and one-body model respectively. This was to determine the temperature and flow field in the SFP caused by natural convection process due to decay heat transfer. All SFP conditions used in this computation were based on pressurized water reactor SFP, having maximum burnup value of 55GWd/t. The value of 10MW corresponds to the decay heat value of an SNF that is unloaded one week after the reactor shutdown. All SNF were assumed to have the same SNF decay heat profiles and arrangements. The simulation was terminated after five hours of real time just before the temperature at one of the points reached a maximum value of about 373K.
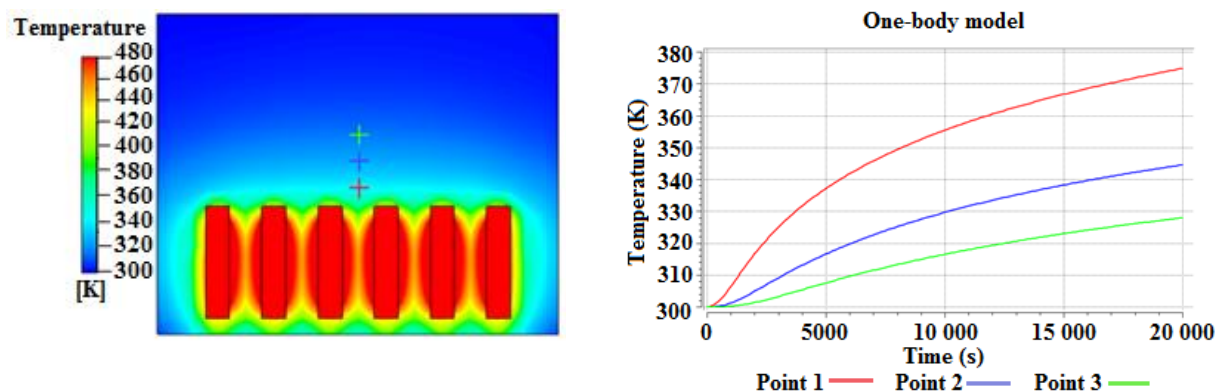
3. Results and Discussion

The temperature distribution and three different point temperatures for individual-body model and one-body model are shown in Figure 5 and Figure 6 respectively. These three points were selected based on the axial distance from the SNF (Point 1: 0.5 m, Point 2: 1.5 m, and Point 3: 2.5 m). Both Figures show the temperature at the points of the SFP after five hours simulation time. The time taken for individual-body and one-body models to achieve the result was about six days and three days respectively.

Based on the results, the temperature at the bottom part of the SFP was higher than that of the upper part since the SNF was located at the bottom of the SFP. Although the temperature at the nearby SNF in the SFP had increased to almost 100 ºC, the surface of the water inside the SFP was still at its initial temperature. The results were consistent with the analysis result from Chan et. al [7], which indicates that the upper part temperature of SFP was found to increase more slowly than the bottom part of the SFP. This suggests that the rate of heat transfer due to natural convection process is low. Thus, it is unable to transfer the decay heat from the SNF to the surface of the water efficiently. It was also observed that the temperature between the SNF assemblies was higher than in other locations in the SFP.

Figure 5. Temperature distribution and three different point temperature of individual-body model.
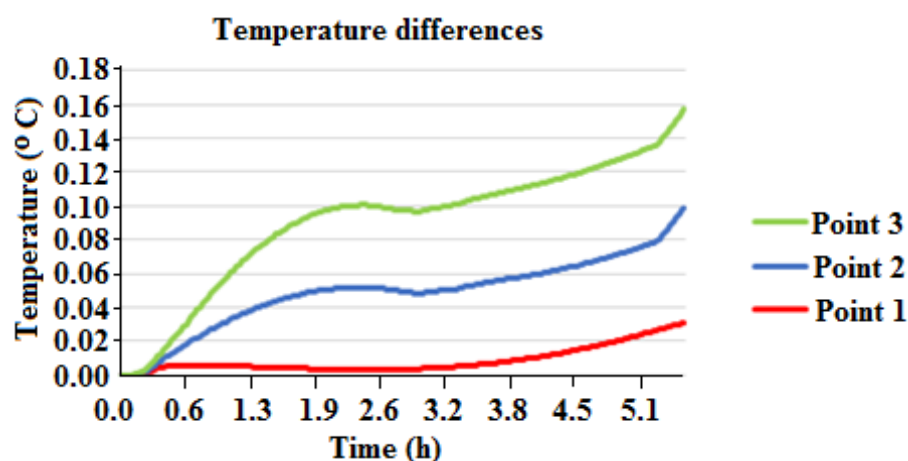
Figure 6. Temperature distribution and three different point temperature of one-body model.

3.1. Comparison of individuals-body and one-body models simulation results

Figure 7 shows the temperature differences between the individual-body and the one-body models at three different points in five hours of simulation time. The temperature differences between these two models were very low at less than 1ºC and they increased at a very low rate as simulation time passed. The temperature differences also increased at a very low rate when the distance of the measuring point from the SNF increased. These results show that using the one-body model only had a very small effect on the computation of SFP temperature distribution. This is due to the conduction process that occurs between each SNF cells which transfer the heat to achieve equilibrium condition and standardize the temperature of all the SNF assembly. This results also might be because of heat transfer due to natural convection process in the SFP which has certain limit regardless the amount of the decay heat introduced. Besides that, the small difference in temperature occurs between these two models is due to the effect of the nearest SNF cell which has generated different amount of heat. The real simulation time taken for individuals-body and one-body model was six days and three days respectively. The latter only took three days less to obtain a comparable SFP temperature distribution with 1ºC difference than the individuals-body model.

Figure 7. Temperature difference between individual-body and one-body models at three different point location.

4. Conclusions

In this study, a 3D thermal behavior of SFP during the loss of external cooling system was computed using Ansys Fluent 18.0 CFD software to investigate the applicability of the one-body model to predict the thermal distribution in the SFP. The investigation was done by comparing the one-body model thermal distribution with the individual-body model thermal distribution. The following conclusions were obtained:

- There was a high temperature difference (about 15 ºC) between the bottom part (near the SNF) and the upper part of the SFP. The temperature at the surface of the water was still at its initial temperature even when certain parts in the SFP had reached 100 ºC. This shows that natural convection process has a low rate of heat transfer and therefore, it is unable to transfer the decay heat from the SNF to the surface of the water efficiently.
- Compared to the thermal distribution of the individual-body model, the thermal distribution obtained by using one-body model had no significant difference. Based on the three-point measuring temperature, the temperature difference was low at less than 1 ºC. The one-body model simulation was completed in three days, while the other took six days. Therefore, using one-body model seems to be favorable in predicting thermal distribution in nuclear safety analysis.

5. References

[1]   Yanagi C, Murase M, Yoshida Y, Utanohara Y 2016 Effect of decay heat distribution on water temperature in a spent fuel pit and prediction errors with a one-region model *J. Nuclear Eng. and Rad. Sci* **2(3)** p 031001(1)-031001(10).

[2]   Yanagi C, Murase M, Yoshida Y, Utanohara Y, Iwaki T, Nagae T 2012 Numerical simulation of water in a spent fuel pit during the shutdown of its cooling system *J. Power Energy Sys.* **6(3)** p 423-430.

[3]   Yanagi C, Murase M, Yoshida Y, Kusunoki T 2015 Prediction of temperature and water level in a spent fuel pit during loss of all AC power supplies *J. Nuclear Sci. and Tech.* **52(2)** p 193-203.

[4]   Yanagi C, Murase M, Yoshida Y, Iwaki T, Nagae T 2012 Evaluation of heat loss and water temperature inn a spent fuel pit *J. Power Energy Sy.,* **6(2),** p51-62

[5]   Hasan R, Tudor J, Ramadan A 2015 Modelling of flow and heat transfer in spe.nt fuel cooling ponds *Int. Congr. on Adv. in Nuc. Power Plant* p 15239

[6]   Tzu C H, Vijay K D, Bau S P, Yen S C, Fengjee P T 2013 The development of a three-dimensional transient CFD model for predicting cooling ability of spent fuel pools *Appl. Thermal Eng.* p 496504.

[7]   Chan H P, Arim L, Jung M L, Joo H M 2013 Estimation of the spent fuel pool water temperature at a loss of pool cooling accident *J. Math.l, Comp., Phys., Eelec. and Comp. eng.* **7(7)** p 1183-1186.

[8]   Chen S R, Lin W C, Ferng Y M, Chieng C C, Pei B S 2014CFD simulating the transient thermal-hydraulic characteristics in a 17x17 bundle for a spent fuel pool under the loss of external cooling system accident *Ann.s of Nuc. Energy* **73** p 241-249.

[9]   Hung T C, Dhir V K, Pei B S, Chen Y S, Tsai E P 2013The development of a three dimensional transient CFD model for predicting cooling ability of spent fuel pools *Appl. Thermal Eng.* **50(1)** p 496-504

[10]   Xiaoli W, Wei L, Yapei Z, Wenxi T, Guanghui S, Suizheng Q 2014Analysis of the loss of pool cooling accident in a PWR spent fuel pool with MAAP5 *Ann. of Nuc. Energy* p 198-213