Progress in Nuclear Energy
Manuscript Draft

Manuscript Number: PNUCENE-D-16-00308

Title: Challenge Analysis and Schemes Design for the CFD Simulation of PWR

Article Type: Research Paper

Keywords: PWR; CFD; challenge; scheme

Challenge Analysis and Schemes Design for the CFD Simulation of PWR

Guangliang Chen, Zhijian Zhang*, Zhaofei Tian, Lei Li

Fundamental Science on Nuclear Safety and Simulation Technology Laboratory, Harbin Engineering University, Harbin 150001, China

Abstract: CFD simulation for a PWR is an important part for the development of Numerical Virtual Reactor (NVR) in Harbin Engineering University of China. CFD simulation can provide the detailed information of the flow and heat transfer process in a PWR. However, large number of narrow flow channels with more and complex structures (mixing vanes, dimples, springs, etc.) are located in a typical PWR. To obtain a better understand of the CFD simulation, the challenges created by these structural features were analyzed and some quantitative regularity & estimation were given in this paper. It is found that both the computing resources and time are in great need for the CFD simulation of a whole reactor. These challenges have to be resolved, so two schemes were designed to assist/realize the reduction of the simulation burden on resources and time. One scheme is used to predict the combined efficiency of the simulation conditions (configuration of computing resources & application of simulation schemes), so it can assist the better choice/decision of the combination of the simulation conditions. The other scheme is based on the suitable simplification and modification, and it can directly reduce great computing burden. Therefore, both these two schemes will be useful to overcome the challenges in the CFD simulation of a whole reactor.

1. Introduction

Advanced simulation tools based on the latest physical models, computing condition, and numerical technology have been the research focus in the international scope, such as VERA which is being developed by CASL in USA (DOE, 2011), and NERISIM/NERISP/NURESAFE (Chauliac et al. (2011) and Chanaron et al. (2012)) which were developed by the cooperative countries in Europe. The development of these advanced products rely on the fine-scale multi-physics computational models (Saha et al., 2013). With the same objective, Numerical Virtual Reactor (NVR) is another advanced simulation tool which ~~is~~ was being developed by Harbin Engineering University of China. NVR is based on: 1) The physical models using the first principle; 2) The fine 3D modeling of the simulation objects; 3) The strong coupling of multi-physics; 4) Advanced numerical solution; 5) High performance computing, etc. The objectives of NVR are: 1) The accurate prediction of the physical process of reactor; 2) 3D monitor of the physical status; 3) Best understanding of the interaction in multi-physics; 4) Analysis with less safety margin for nuclear reactor; 5) Improvement of the system analysis code. So that NVR can be used for the design, safety analysis, operation research, and education & training

CFD simulation of reactor is an important part for the development of NVR. Because CFD simulation can reflect the details of the flow and heat transfer which can be used in the further application or research. For instance, with the application of CFD simulation, In et al. (2008) analyzed the principle of heat transfer enhanced by spacer grid with mixing vane; Kim and Seo (2004 and 2005) gave the optimal design of the shape of mixing vane; Lee and Choi (2007) study the influence of the region size of the research object, etc. Though CFD simulation is so important, some challenges still exist in the application.

Firstly, for a reactor, the successful CFD simulation depends on the mesh quality and the size of research object, which always bring great computing burden. In a typical pressurized water reactor, there are thousands of narrow flow channels and complex structures around the spacer grid and mixing vane, so that great cell number is needed. Thus the computing resource is also great, or the research object also can be cut into a local region with several rods. Then the computing resource can be satisfied easily, however the region size can affect the accuracy of the simulation results (Lee and Choi, 2007). To obtain the high fidelity results, simulation for the whole or the large region of a reactor has to be done. So some optimal methods and schemes should be designed to reduce the computing burden.

Secondly, simulation time is also too long when the

* Corresponding author. Tel.: +86 18686836080
E-mail addresses: chenguangliang.heu@gmail.com

cell number is large because of the iterative numerical technology in CFD computation. Though some optimal methods or schemes can reduce a certain cell number, the total cell number is still very large for a whole reactor or a large region of a reactor. The research of physical process and the engineering application using CFD simulation are also impossible with low simulation efficiency, so the optimal methods and schemes are also needed to improve the efficiency.

Thirdly, the accuracy can be tested by the experimental data, however it is confusing to have a high efficient CFD simulation without a quantitative guidance on the design and choice of simulation conditions which contains the configuration of computing resources and the design of simulation schemes (meshing methods, solution methods, physical models, etc.). Usually, a researcher has to try and compare many times to find a suitable combination. Thus lots of time is lost and much computing resources are occupied during this process. Furthermore, the combination set for the simulation is always just suitable, but not optimal. So the judgment standards are also needed for the efficiency improvement.

For the reasons above, this paper tries to obtain: 1) the clear understanding of the challenges for the CFD simulation of a PWR reactor; 2) the quantitative estimation to choose the high efficient simulation conditions; 3) the advanced methods and schemes to reduce the burden in both computing resources & time.

2. Research objects and approaches

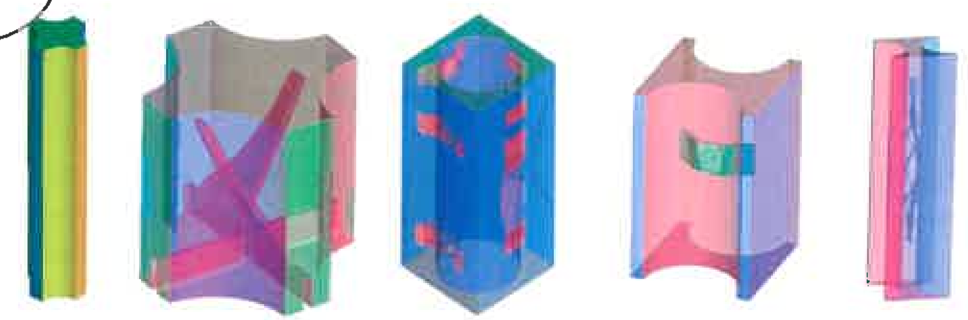
2.1. Research objects

In application of CFD simulation, the first and important step is the mesh analysis which is closely related to the accuracy. So the mesh analysis is carried out in this part. In most previous research, the mesh analyis is only done for the whole research object. However, the last chosen mesh may be still not good because poor quality mesh can exist in the local feature region. Many feature regions are in a typical PWR, so this research was carried out for each feature region respectively.

The typical PWR has hundreds of fuel assemblies, and each assembly contains hundreds of fuel rods. Fortunately, similar feature structures exist there, though great number of narrow flow channels are created by these fuel rods. These feature strucutres are the simple sub-channel (SC), spacer grid (SG), and mixing vanes (MV), as shown in Fig. 1 (a~c). The geometry of SG is more complex. To suit the small local region, SG is divided into two regions (regions around dimple (DI) and spring (SP) as shown in Fig. 1 (d~e)).

In addition, the number of the feature structures (SC, MV, DI, and SP) is dozens of thousands. Therefore, the cell namber saved in one local region will be enlarged thousands of times for the whole reactor. So this research compared different mesh types and cell numbers for each feature region to find the better meshing method.

The geometrical parameters of the SC and MV come from the work of Karoutas et al. (1995), as shown in Fig. 2. The parameters of this object can be found in the reference from Navarro et al. (2011). The object of SG is a part of AFA2G grid as shown in Fig. 3, and the parameters can be obtained from Ma (2012).



(a) SC (b) MV (c) SG (d) DI (e) SP

Fig. 1. Regions for research of meshing methods

Fig. 2. Experimental spacer grid of Westinghouse

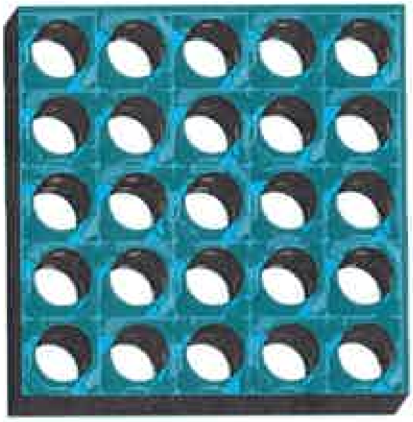


Fig. 3. AFA2G grid

2.2. Numerical approches

In this paper, the research was performed by ANSYS Fluent 15.0. Three RANS models (SKE, RNG, and RSM) were used. The standard wall function method was used and the y^+ in the first layer of the mesh near the wall is kept between 30 and 60 for these RANS models. The spatial discretization methods in the simulations are second order for pressure term, and second order upwind for other terms. The convergence criteria of each term is set as 1e-6. The boundary condition is shown in **Table 1**. The physical properties of the coolant is calculated by IAPWS - IF97.

Table 1
Boundary conditions

Parameter	Value
Inlet pressure	0.483 MPa
Inlet temperature	26.67 ℃
Inlet velocity	6.79 m/s
Outlet boundary	Outflow ?

Initial conditions ?
BC? P

2.3. Analysis of meshing methods

To avoid the low accurate simulation, the mesh independent analysis was done for each meshing method (mesh type) for each feature region. Then the minimum cell number can be obtained for the meshing method for each feature region. So more suitable meshing method can be found by the comparison of the minimum cell number for each feature region.

In this research, the pressure distribution, lateral flow distribution, the total simulation time (TST), the mean time per step (MTPS), the iteration step number ? (ISN), the cell number ? (CN), etc. ~~were recorded~~, compared, and analyzed for each meshing method to obtain some regularity. The following part will introduce the details of the analysis.

2.3.1. Mesh analysis for simple sub-channels

The simple sub-channel region has the same crosswise structure along the vertical direction, so prismatic mesh types were compared since their high mesh quality and low need of cells. The triangular prism (TP), polyhedral prism (PP), and hexahedral prism (HP) were chosen for the sub-channel region, as shown in Fig. 4 (a-c). In the mesh establishment, the same axial mesh size was used firstly to find the suitable crosswise cell size, then various axial sizes were used to find the suitable axial size.

The simulation results created by different mesh types and cell numbers are shown in Fig. 5. **Table 2** gives the simulation time for each mesh type with the least cell number. In the comparison, it can be seen that each mesh type can get the same result when the cell number is over a certain value, and both the cell number and the simulation time needed by hexahedral prism mesh are the least. Hexahedral prism also can be established by many meshing methods as shown in Fig. 4 (c-e). Hexa-prisim2 (HP2) and hexa-prisim3 (HP3) are two structured mesh types which are always used in the previous simulations as the work of Horváth and Dressel (2013) and Nematollahi and Nazifi (2008). Hexa-prism (HP) is an unstructured mesh type. It's always known that structured types are better for the data storage and transfer, so the simulation speed of structured mesh is faster and the iteration step number is smaller. However, HP2 and HP3 need more cell number from the comparison in Fig. 6 and

arguable?

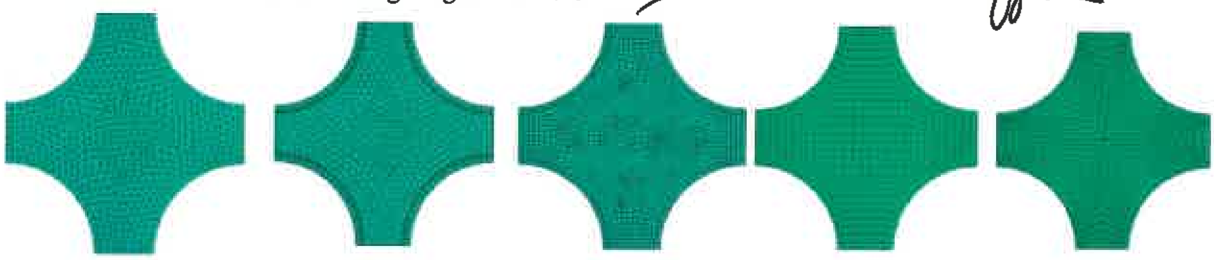
And what is HP4?

Table 2. It can be seen that the unstructured meshing method is more efficient with less cell number and simulation time. The reason may be that the unstructured mesh has strong geometrical adaption for the structural feature of the simple sub-channel region, however the quality of the structured mesh will decline obviously when the cell number is low for this structural feature. Meanwhile, the unstructured mesh is also simple and convenient for the establishment, but the structured mesh types need more experience and time to design and modify. Furthermore, the establishment of the unstructured mesh is easier for a large region like the whole reactor. So it's better to choose HP for the simple sub-channel region.

Then the mesh independent analysis was done on the axial mesh size for HP mesh. As shown in Fig. 7, HP4 is the mesh type with the optimal axial and cross cell size.

From the comparison in **Table 2**, the suitable cell number is from 133k to 9k, and the simulation time is from 198s to 2s. This is also one evidence for the significant of the design and choice of the optimal meshing method to reduce the simulation resource and time.

diff?



(a) Tri-prism (b) Poly-prism (c) Hexa-prism (d) Hexa-prism2 (e) Hexa-prism3

Fig. 4. Prim mesh for the simple sub-channel region

Table 2
Computational cost of each mesh type for SC

Region	Mesh type	Least cell number (k)	Least time (s)
SC	TP	133	198
	PP	106	117
	HP	29	19
	HP2	104	94
	HP3	131	134
	HP4	9.2	2

? ?

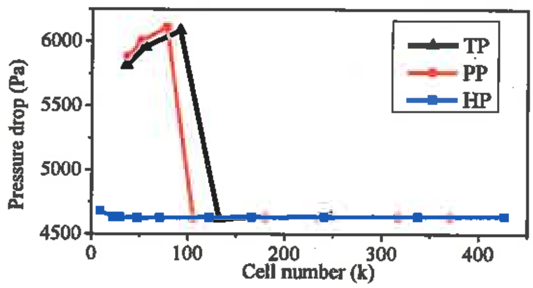


Fig. 5. Analysis on prism meshes

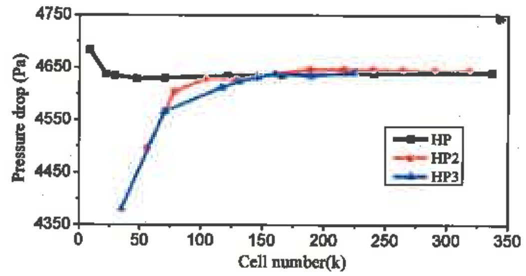


Fig. 6. Analysis on hexahedral meshes

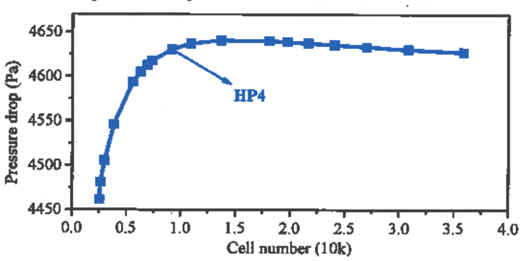


Fig. 7. Analysis on axial mesh size

2.3.2. Mesh analysis for complex structure regions

The regions surrounding mixing vane, dimple, and spring have complex geometrical feature. The polyhedral (PH) mesh was used because of its high adaptive ability, and PH has been proved better for these regions in PWR by Li and Gao (2014). Each mesh is shown in Fig. 8.

The mesh independent analysis is shown in Fig. 9, and Table 3 gives the least cell number and the lease simulation time for the simulations to these complex structure regions. In the analysis, only one single feature region was considered, however the cell number is still not small. So the simulation burden will be enlarged obviously when the simulation object is a large region with a large number of complex regions.

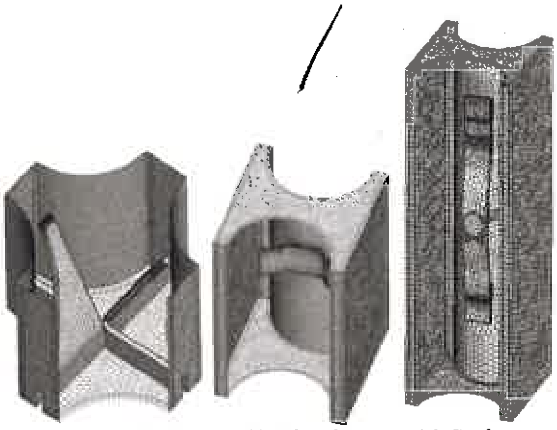
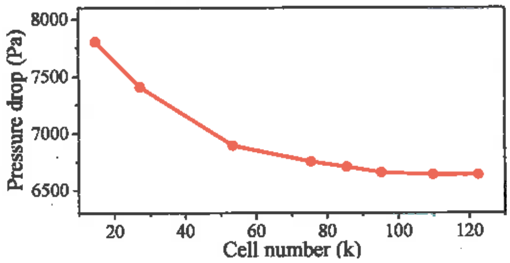
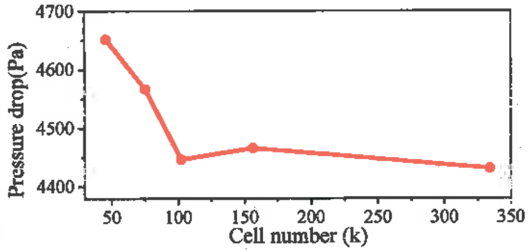


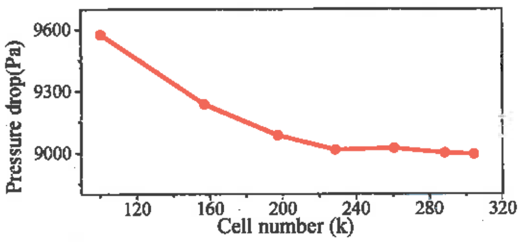
Fig. 8. Polyhedron for the complex structure



(a) Region surrounding the mixing vane



(b) Region surrounding the dimple



(c) Region surrounding the spring

Fig. 9. Mesh analysis for complex regions

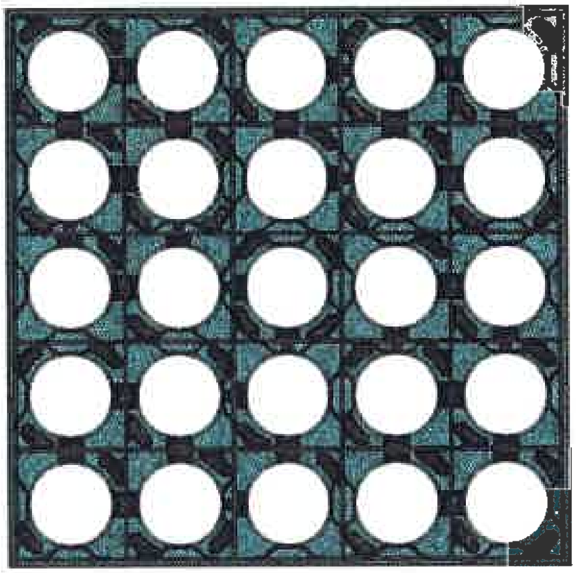
Table 3
Cost for the simulation of each region

Region	Mesh type	Least cell number (k)	Least time (s)
MV	PH	85	78
DI	PH	103	146
SP	PH	229	385

2.3.3. Mesh analysis for 5×5 AFA 2G grid

Based on the research of meshing method for each feature region, the meshes were built and checked for a 5×5 AFA 2G grid. As the previous research, the polyhedron is used for the spacer grid as shown in Fig. 10.

Meanwhile, as the previous estimation, the cell number for the inner region of 5×5 AFA 2G grid is nearly $((103\times2+229)/2+85)\times5\times5$ k (≈756 million). The mesh analysis was done with both pressure drop and lateral flow status as shown in Fig. 11 and Fig. 12. It can be found that the difference on resistance distribution is not obvious, however 7.98 M cell number can obtain better lateral flow rate. So the suitable cell number is very near the previous estimation.


(a) Mesh for a cross section

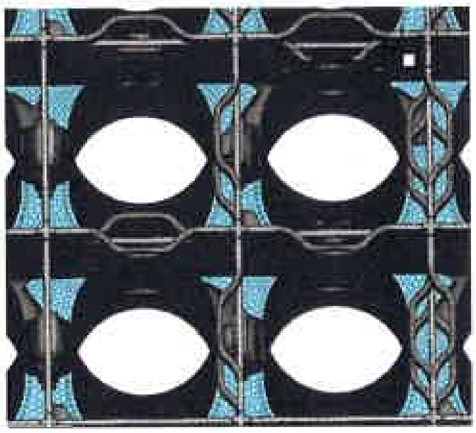

(b) Mesh for a local region

Fig. 10. Mesh for 5×5 AFA 2G grid

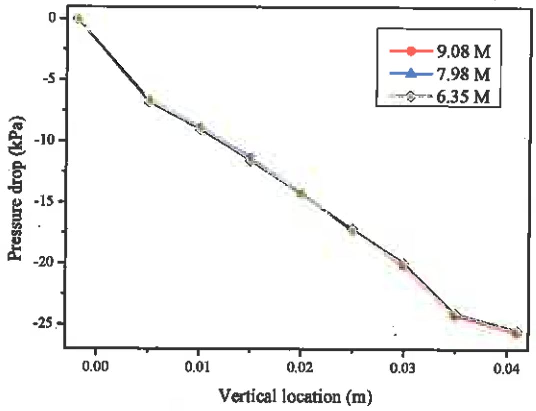


Fig. 11. Analysis on pressure drop for 5×5 grid

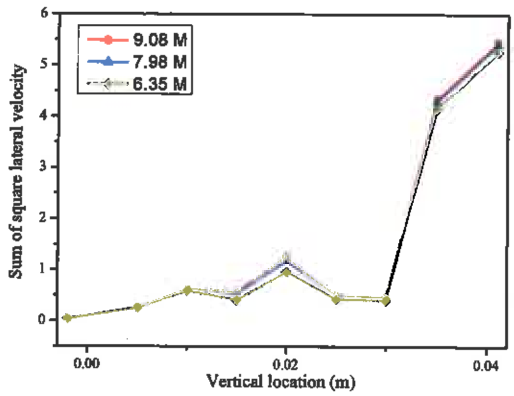


Fig. 12. Analysis on square lateral velocity for 5×5 grid

2.4. *Validation on the simulation*

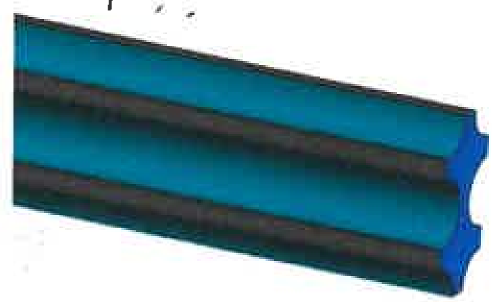
Not only the mesh independent analysis is important for a CFD simulation, the validation is also an important step for the application. In a validation, the mesh type, cell density, turbulence model, design of boundary layer, operation of geometry, numerical discrete scheme, solution scheme, etc. can be tested to see whether they are suitable or optimal for the research object.

The tested object and boundary condition come from the work of Karoutas et al. (1995). The mesh type of each feature region for the validation is shown in Fig. 13. The hexahedral prism is used for the simple sub-cahnnel. The polyhedron is used for the spacer grid. The mesh sizes are based on the previous mesh independent analysis for each feature region.

The spatial discretization methods are introduced in section 2.2. The main boundary conditions are listed in **Table 1**. As shown in Fig. 14, other side boundaries are designed with periodic boundaries. PB1 & PB2, PB3 & PB4, and PB5 & PB6 are three pairs of periodic boundaries.

The transverse flow status is important for the mass, momentum, and energy exchange between adjacent coolant channels. Therefore, this status is chosen for the comparison and the parameter is defined as the rate of ~~the~~ lateral velocity and ~~the~~ bulk velocity. The lateral velocity is normal to the red path line in Fig. 14. The cross sections with vertical position of 12.7 mm and 101.6 mm were chosen for the comparison. The red path line of each section for the comparison of velocity profile was designed as shown in Fig. 14. Its length is 2 pitch.

The simulation results are shown in Fig. 15. It can be seen that it is very near between the simulation results and the experimental data (Karoutas et al., 1995) when we use the mesh and other simulation conditions which were introduced in the previous sections. Therefore, our research approach can be used for the following research in this paper. ??



(a) Mesh type for simple sub-channels

(b) Mesh type for spacer grid
Fig. 13. Mesh scheme for each region

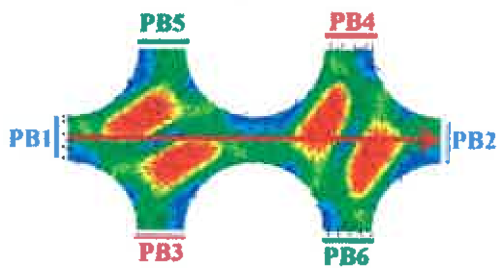


Fig. 14. Boundary design and pathline for analysis

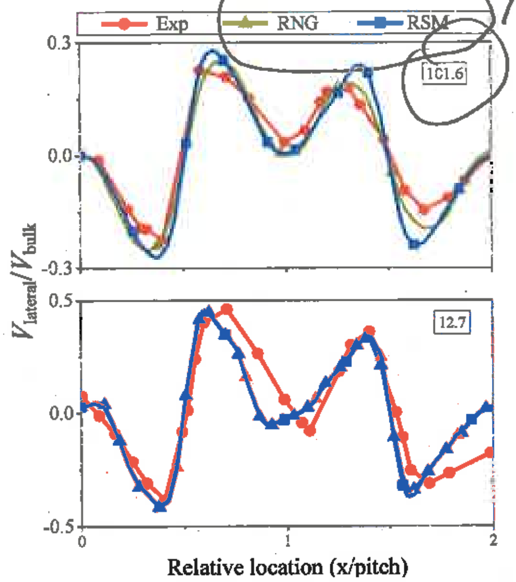


Fig. 15. Validation for simulation approach

3. Challenge analysis

3.1. Great need of computing resources

Based on the mesh independent analysis above for the feature regions (MV, DI, SP, and SC), the cell number can be estimated for a whole PWR. **Table 4** gives the parameters of the nuclear power plant Qinshan I of China. Based on these parameters, the estimation of the cell number for this reactor is listed in **Table 5.** The total cell number is around 70 billion, which is too large. Furthermore, the estimation is just based on the regions of SC, MV, and SG. The cell number for the regions between adjacent assemblies and the upper/downer plenum is not included. So the cell number for a whole reactor must be larger than 70 billion.

In the estimation of other studies, such as the work of Conner et al. (2010), the cell number for a 5×5 rod bundle with one layer of grid spacer and mixing vane is around 20 million. Then the number for a whole reactor must be more than 20×15×15/(5×5)×8×121 million (1700 billion). So the computing quantity for a whole PWR is really extraordinary. Accordingly, high performance computing (HPC) is in great need. However, the computing time is also very long even if the advanced HPC equipment and technology are used. This is really a challenge for the CFD simulation of PWR.

Table 4
Parameters of nuclear power plant of Qinshan I

Height of fuel assembly	3500 mm
Height of fuel rod	3200 mm
Height of one layer of spacer grid	36 mm
Number of fuel assemblies	121
Arrangement of fuel rods	15×15
Layer number of spacer grid	8
Layer number of mixing vane	6
Height of the simple sub-channel	3.2-0.036×8≈2.9m

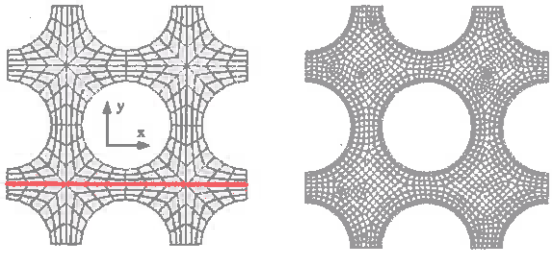
Table 5
Estimation of the cell number for the CFD simulation of Qinshan I

Region	Cell number per unit region	Region size per channel	Fuel rods per assembly	Assembly number	Total cell number of each region
Simple Chanel	102 k/m	2.9 m	15×15	121	~ 8.05 billion
Mixing Vane	85 k/layer	6 layers	15×15	121	~ 13.88 billion
Spacer Grid	218 k/layer	8 layers	15×15	121	~ 47.48 billion
Total cell number > 69.41 billion					

3.2. High demand on fine mesh

Some research has been focused on the coupling technology between CFD and MOC for a whole reactor (Weber et al., 2004 and 2007). However, the mesh density used in these CFD simulations is coarse. This kind of mesh may affect the acuracy, although it can save great computing resouces.

To discuss the influence from the coarse mesh, some research was done. As shwon in Fig.16, the left one is the mesh type used in reference (Weber et al., 2007), and the right one is the mesh type tested by previous mesh independent analysis. Simulations using these two tpyes on a 2×2 sub-channels (a part of the experimental object (Karoutas et al., 1995) and the experimental conditions were also used in the simulations) were done and compared. As shown in Fig.17, two cross sections with vertical locations, 12.7mm and 101.6mm, were used for the comparison. The red line in Fig.16 is chosen for the data analysis. The velocity at the y direction is used as the parameter for the comparison. In Fig.17, two simulation results and experimental data are shown, and it can be seen that the coarse mesh is not suitable to reflect the accurate flow status. Though mesh type 1 is also very fine, it still create large error. So the mesh must be kept fine enough for the CFD simulaiton of a PWR, which is also a challenge for the CFD simulation of a whole PWR.



a) Mesh type 1 (b) Mesh type 2

Fig.16. Mesh types for comparison

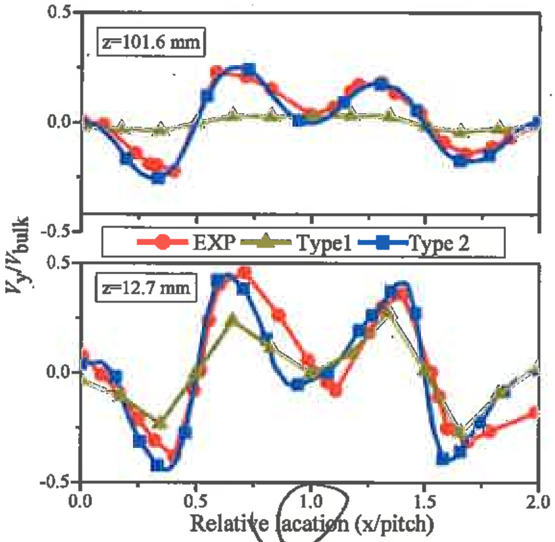


Fig.17. Analysis of the influence of mesh density

This depends on: mesh, disc Schemes, solvers used, ... and none of them were listed/described

3.3. Large cost of computing time

As shown in Fig. 18, similar regularity is suitable for each feature object (SC, MV, DI, and SP). This regularity is that the mean time per iteration step (*MTPS*) and the cell number (*CN*) have a good linear relationship which can be written as Eq.1. In Eq.1, k_1 and a are two constant parameters for the linear relationship.

$$MTPS = k_1 CN + a \tag{1}$$

Regularity also exists between the cell number (*CN*) and the iteration step number (*ISN*). As shown in Fig. 19, it is also a good linear relationship for each research region. Eq.(2) can be used to reflect this regularity, and k_2 and b are two constant parameters for the linear relationship.

$$ISN = k_2 CN + b \tag{2}$$

The total simulation time is decided by the iteration step number and the mean time per iteration step. The product of two linear relationship makes a parabolic relationship. So the relationship between the total simulation time (*TST*) and cell number (*CN*) is a parabolic relationship which can be seen from Fig. 20. This relationship can be expressed as Eq.3, and C_1, C_2, and C_3 are the constant parameters affected by the parabolic relationship.

$$TST = ISN \times MTPS = C_1 CN^2 + C_2 CN + C_3 \tag{3}$$

From the previous research, it was known that the fine mesh is in great need for the CFD simulation of a PWR. Then the high mesh density and the parabolic relationship decide the total simulation time will be very long for the CFD simulation of a whole PWR. This also is a challenge for the CFD simulation of a whole reactor.

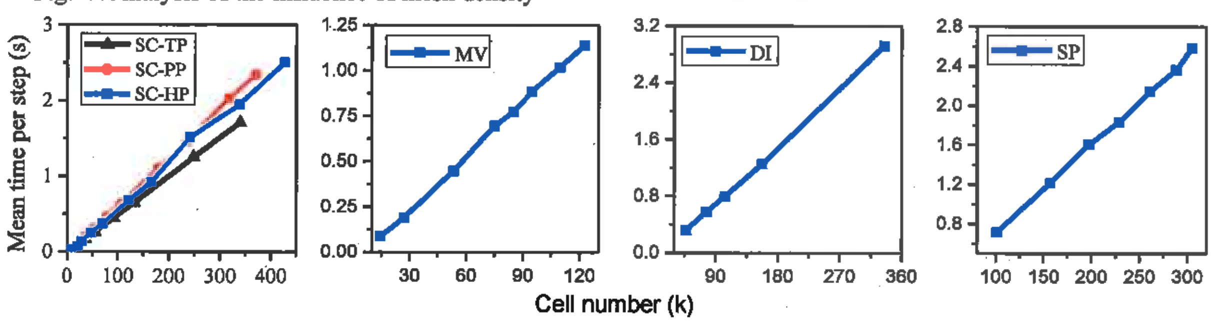


Fig. 18. Regularity for the mean time per iteration step

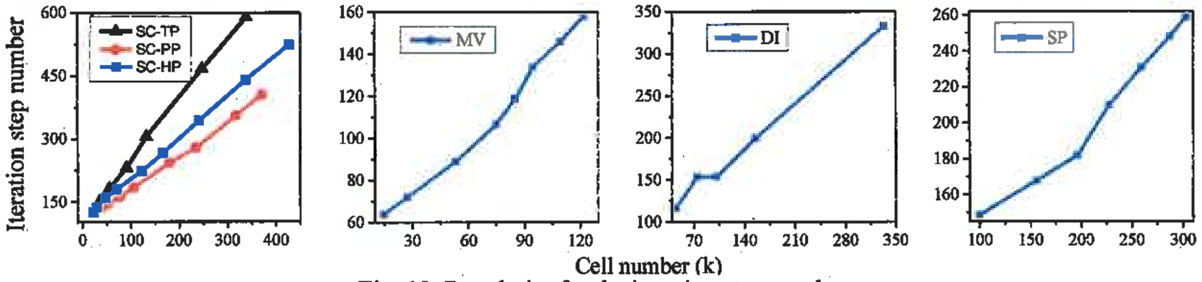


Fig. 19. Regularity for the iteration step number

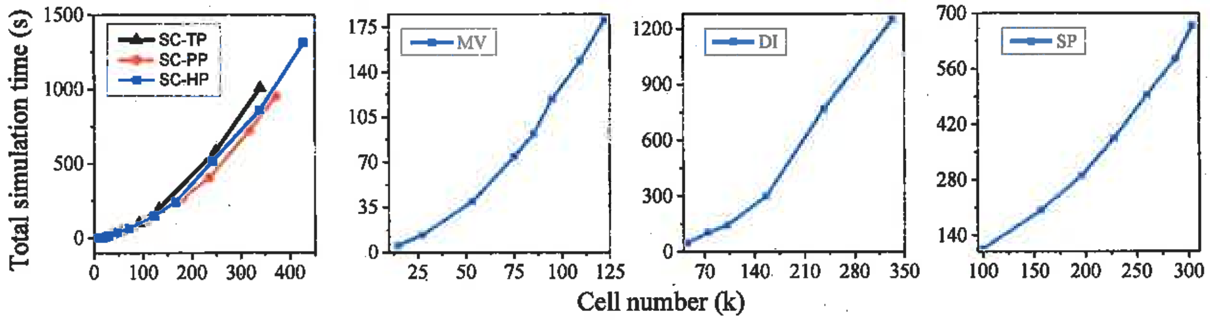


Fig. 20. Regularity for the total simulation time

4. Design of the simulation scheme

From the analysis above, it can be known that the computing resources and time are in great need for a CFD simulation of a whole PWR. Then the improvement of the simulation efficiency is very important. Because it is meaningless if the simulation is without an end, or the simulation needs too much computing resources.

To improve the efficiency, one usual approach is to design the optimal mesh scheme with the minimum cell number (related to computing resources) and the least simulation time, as the mesh analysis in section 2.3. However, both the resources and time are still great for the whole PWR after the application of optimal mesh scheme. So some research was also done to develop other schemes to assist/obtain the further reduction of simulation resource and time in this paper. The details are as follow.

4.1. Quantitative estimation scheme for efficiency

To improve the efficiency, the estimation of the efficiency has to be done firstly. However, few research has been focused on this field. Though the knowledge is lack, one thing can be sure is that the efficiency can be affected by the simulation conditions (the configuration of computing resources & the simulation schemes), as shown in Fig. 21. So some research was done in this direction.

In one aspect, the computing resources contains many factors, such as the number of computing nodes, the type and the clock speed of CPU, the memory size, etc. In the other aspect, there are also many simulation schemes which can affect the efficiency. For example, the schemes can focus on the establishment of cell meshing, design of boundary layer, choice of turbulence model, discretization of numerical model, design of solution methods, etc. (Conner et al., 2015; Wells et al., 2015).

All these factors above can affect the efficiency. So the efficiency should be considered by the overall performance of these factors. Furthermore, the research and development of the configuration of computing resources and design of simulation schemes are carried out in the international scope. The optimal one is also difficult to choose, because the standard is lack for the comparison.

To obtain an optimal simulation with high efficiency, ~~A~~ quantitative standard will be useful if it can give a combination assessment for the efficiency status of the computing resources and simulation schemes. To obtain a successful CFD simulation of a whole PWR, some work has been done.

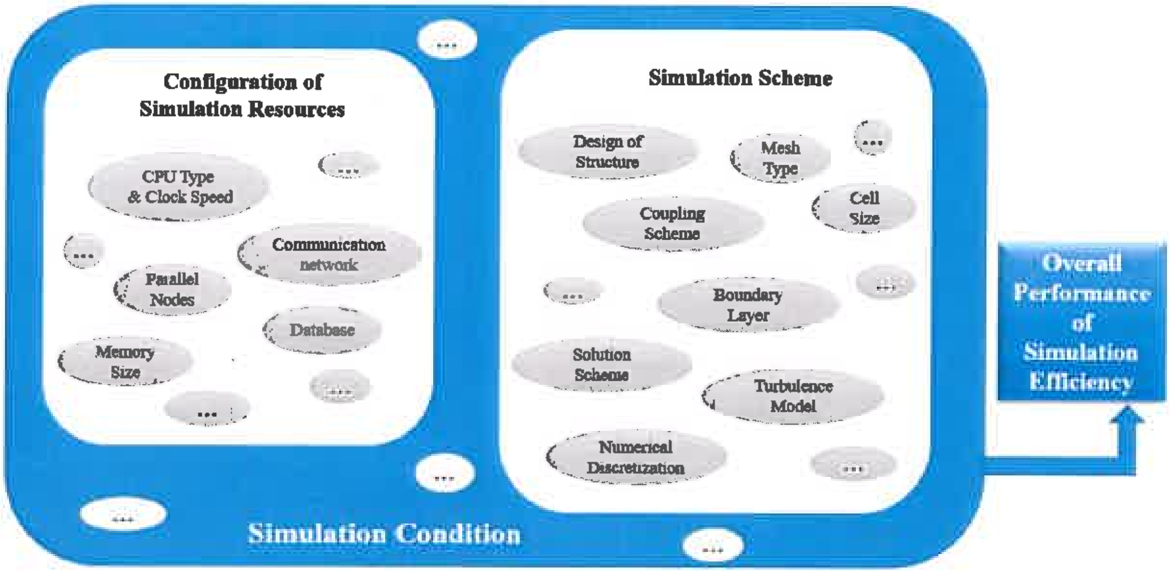


Fig. 21. Influence factors on the overall performance of the simulation efficiency

4.1.1. Pre-estimation method

As the regularity discussed in section 3.3, the total simulation time can be estimated by some equations. Meanwhile, the main coefficients are all constant parameters because of the liner or ~~parabolic~~ relationship, and these constant parameters should be ~~decided~~ by the combined effect of the simulation conditions. The constant parameters k1, a, k2, b can be obtained by two test simulations because of the linear relationship. Then from Eq.3, C1, C2, and C3 can be obtained by two test simulations, because C1 is equal to k1k2, C2 is equal to ak2+ bk1, C3 is equal to ab. In other words the total simulation time can be estimated by two simulation tests for one kind simulation using any cell number.

This regularity will be useful to give the quantitative reference. We can predict the nearly time before the simulation with great cell number is applied. So this method is called pre-estimation. The details of the application of this method are listed as follows.

Firstly, based on the linear fitting, we can obtain the slops of linear relationship for *ISN&CN* and *MTPS&CN*. The slop between *IN* and *CN* is set as K1, and the slop between *MTPS* and *CN* is set as K2. Then the product of K1 and K2 should be important for the efficiency when *CN* is great because K1K2 is the coefficient of quadratic term. The need of cell number is great for a whole PWR, therefore K1K2 is meaningful to decide the nearly total simulation time for the CFD simulation of a PWR. For example, one overall efficiency will be better than the other one if the ratio of K1K2 between the first one and the second one is more than one. This estimation will be more suitable if the mesh cells are in great number.

Furthermore, this quantitative method just needs nearly two simulation tests. Meanwhile, just small cell number is enough ~~for~~ the simulation tests. Because K1 and K2 are the slops for linear relationship, and a line can be obtained by two points. So the cost of test will be very small for both the resources and time, because we can control the cell number in the tests. Certainly, ~~more~~ tests will be better, because the solution of the slop is by the linear fitting of simulation tests. However, too many tests will increase the simulation burden, and too many tests will not create obvious superiority because of linear relationship.

The steps of pre-estimation method can be generalized as: 1) two or several test simulations need to be done with the target configuration of computing resources and simulation schemes, and the cell number should be small in the test simulations; 2) linear fitting needs to be done on the simulation results to obtain K1K2; 3) with the comparison of K1K2 for the simulations using different simulation conditions, the judgment can be done.

4.1.2. Analysis using Pre-estimation

To give the further explanation of pre-estimation method, the example is given as follows.

Each computer is a combination of a set of computing resources, such as CPU type, clock speed, memory, nodes quality for parallel computation, etc. So two computers were used to stand for different configurations. The details of the configurations are shown in **Table 6**. In addition, many simulation schemes also should be decided for a high efficient CFD simulation. So this example also considered some schemes as shown in **Table 7**. Then several combinations of these simulation conditions can be set as shown in **Table 8**. These combinations respect different configurations, parallel nodes, turbulence models, and discrete methods.

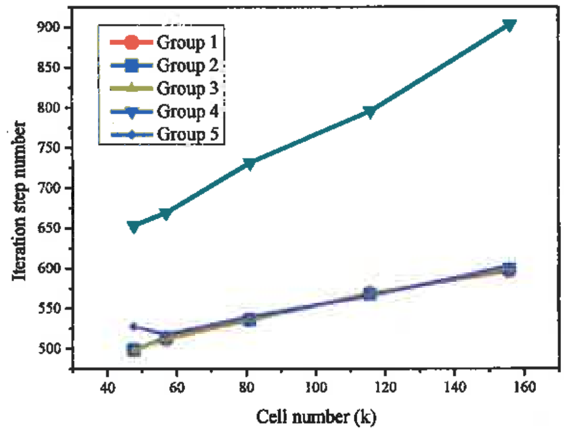
Table 6
Configurations of computing resources

	PC1	PC2
CPU type	Intel Xeon(R) E3-1280 v3	Intel Core(TM) i5-3340M
Clock speed /GHz	3.6	2.7
Memory /GB	32.0	4.0
Max parallel nodes	8	8

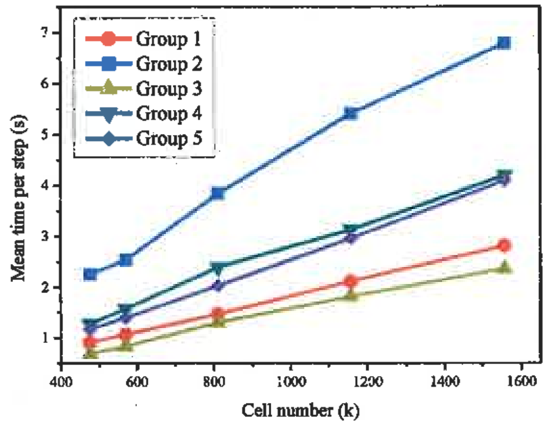
Table 7
Some schemes for CFD simulation

Parallel nodes	4 nodes	8 nodes
Turbulence models	RNG	RSM
Numerical orders	2 order	3 order

With the application of pre-estimation approach, the iterative step number and the mean time per step were recorded, and the results are shown in Fig. 22. Based on the good liner relationship between these two parameters and the cell number, the slops (K1 and K2) were computed respectively. Then the product K1K2 can be got as shown in **Table 8**. The smaller K1K2 means the better efficiency when the cell number is great. So the quantitative standard is obtained, and the judgment can be easier for various combinations of simulation conditions. It can be used to clarify a more suitable one when the difference is small between two conditions or when the experience is lack to judge the conditions in the use.



(a) Number of iterations



(b) Mean time per step

Fig. 22. Efficiency analysis for various schemes

Table 8
Various schemes under pre-estimation

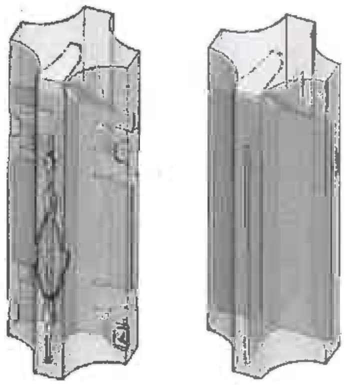
Group	Schemes	K1K2
1	PC1+4Nodes+RNG+2O	0.0160
2	PC2+4Nodes+RNG+2O	0.0390
3	PC1+8Nodes+RNG+2O	0.0143
4	PC1+4Nodes+RSM+2O	0.0604
5	PC1+4Nodes+RNG+3O	0.0202

4.2. SSMS scheme to improve efficiency

The mixing vane is one typical structure of PWR. It has great effect on the lateral flow. The spacer grid is another important structure. Its structure is complex for the existence of springs and dimples. Complex structure makes it difficult to establish the mesh. The cell number will be reduced obviously if the simplification can be done on the structure of the spacer grid. However, the influence of the simplification has to be kept little enough. So some research was also done to discuss the feasibility of the structure simplification.

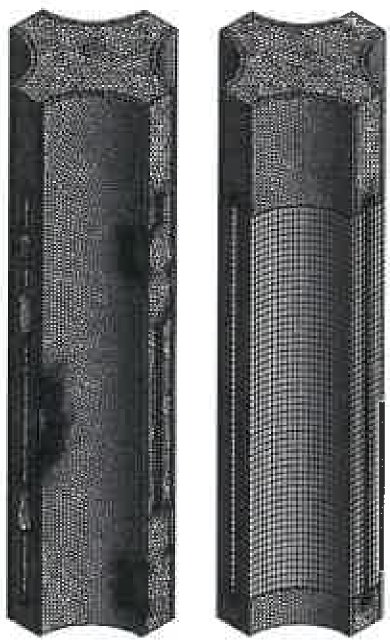
4.2.1. Flow characteristics due to feature structures

As shown in Fig. 23, the research object is a part of AFA 2G spacer grid. The left one is with real structure, and the right one is simplified without springs and dimples. The second region is much simple, because the springs and dimples are gone, and prismatic mesh can be used for meshing. Prismatic mesh is better at the computing resources, speed, and simulation stability. On the contrary, tetrahedron and polyhedron can be easily used for the real structure of a spacer grid. However, the mesh cells will be great more, and the simulation time will also be longer. Moreover, simulations may fail due to the poor stability.



(a) Real grid (b) Simple grid

Fig. 23. Simulation objects for structure research



(a) Real grid (b) Simple grid

Fig. 24. Mesh for structure research

The lateral flow status and the pressure profile were chosen for the research, and the comparison between the two structure types was done. In the simulations, there are two flow channels with 300 mm at the upstream and downstream of the spacer grid as shown in Fig. 25. In the comparison, six cross sections downstream of spacer grid has been chosen, and the vertical locations are 50mm, 100mm, 150mm, 200mm, 250mm, and 300mm respectively. The zero vertical location is at the bottom of the spacer grid. The pathline for the data collection of each cross section is designed as the red line in Fig. 26, and the length of this red line is 1 pitch.

As shown in Fig. 27, the difference of the lateral flow status is not obvious, especially for the region downstream of 100mm. However, as shown in Fig. 28, different pressure profiles exist at the inner region of spacer grid. Fortunately, the pressure profiles at other locations upstream and downstream of spacer grid are not affected by the inner structure of spacer grid, which can be seen from the gradient of the pressure distributions. This regularity make it possible to modify the simulation results at the outlet of the grid region, when the simplification scheme is used to the structure of spacer grid.

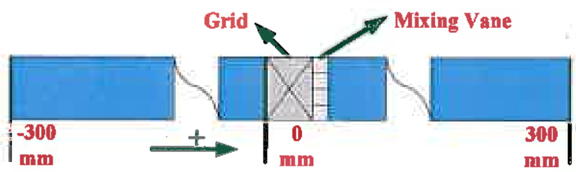


Fig. 25. Simulation range

Fig. 26. Pathline for the comparision of lateral flow

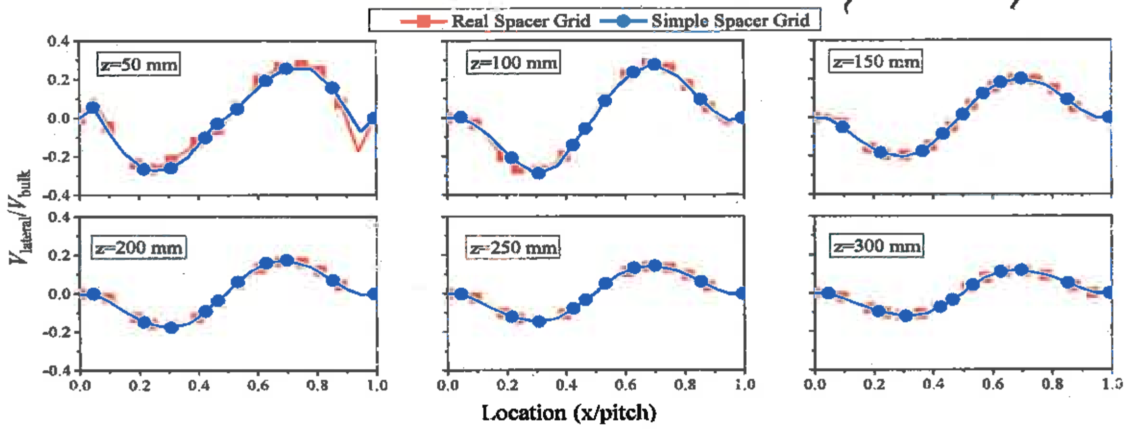


Fig. 27. Comparison of the lateral flows between two kinds of grids

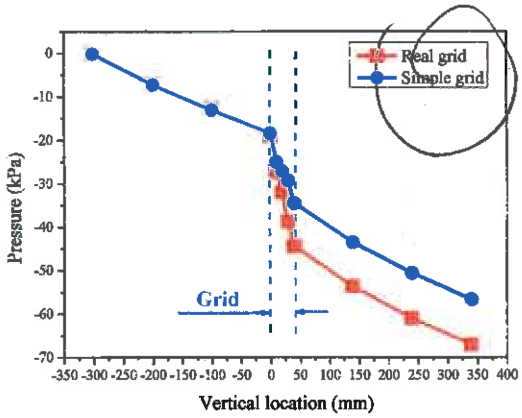


Fig. 28. Pressure profile along vertical direction

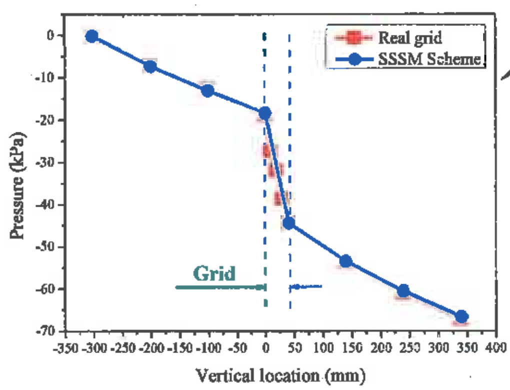


Fig. 29. Pressure profile using SSSM scheme

4.2.2. Simulation using SSSM scheme

From the analysis above, we can know the simple grid without spring and dimple also can be applied to give the nearly real flow status, however the modification has to be done for the pressure distribution in the inner region of a spacer grid. This scheme is named as structure simplification and simulation modification (SSSM). As shown in Fig. 29, the pressure profile using SSSM is nearly the same as the profile using the real grid. In this comparison, the pressure modification was only done on the outlet of the grid region using Eq.4, then the simulation result was improved greatly. However, before the modification, further research is still needed to obtain the resistance characteristic of the grid region (ΔP_{grid}) using CFD simulation or experiment.

$$P_{gridout} = P_{gridout\_simple} + \Delta P_{grid} \quad (4)$$

As shown in **Table 9**, the comparisons of the cell number and simulation time were done for the grid region with the real structure and simple structure. There is nearly half reduction of cell number, and the time is near the fifth times. Therefore, the computing resources and time can be reduced greatly when SSSM scheme is used.

Table 9
Comparison of the simulation cost

Object	Cell number (k)	Time (s)
Real grid	352	302
Simple grid using SSMS scheme	192	66

5. Conclusions

CFD simulation is needed urgently in the development of NVR in Harbin Engineering University of China. For the successful application of the CFD simulation for a PWR in NVR, some work has been done and the conclusions were obtained as follows:

Firstly, based on the analysis on meshing methods and regularities of simulation process for the local feature regions and 5×5 rod bundle, suitable research methods (mesh type, cell size, numerical sets, etc.) were validated and quantitative challenges on computing resources and time were estimated. Large number of narrow coolant channels with complex structures makes the cell size must be small enough. Then near seven billions of mesh cells should be used in the CFD simulation of a whole reactor. Both the relationship for the simulation time & the cell number and the demand on the fine mesh indicate the great need of computing time for the CFD simulation of a whole reactor. To realize the successful CFD simulation of a whole PWR, computing burden on resources and time must be resolved.

Secondarily, the quantitative regularities were used in the design of pre-estimation method which can be applied in the design and choice of high efficient simulation conditions (high efficient configuration of computing resources and simulation schemes). Furthermore, a scheme with structure simplification and simulation modification (SSSM) was designed, and the feasibility of this scheme was validated. Both these two approaches can assist/realize the reduction of computing burden. They will be useful and important to overcome the great challenges in the CFD simulation of a whole reactor.

REFERENCES

Chauliac, C., Aragonés, J. Maria, Bestion, D., Cacuci, D.G., Crouzet, N., Weiss, F.P., Zimmermann, M.A., 2011. NURESIM - A European simulation platform for nuclear reactor safety: Multi-scale and multi-physics calculations, sensitivity and uncertainty analysis. Nucl. Eng. Des. 241, 3416-3426.

Conner, M.E., Baglietto, E., Elmahdi, A.M., 2010. CFD methodology and validation for single-phase flow in PWR fuel assemblies. Nucl. Eng. Des. 240, 2088-2095.

Conner, M.E., Karoutas, Z. E., Xu, Y., 2015. Westinghouse CFD modeling and results for EPRI NESTOR CFD round robin exercise of PWR rod bundle testing. NURETH-16, Chicago, IL, 667-678.

Chanaron, B., Weiss, F.P., 2012. NURISP OVERVIEW. http://www.nurisp.eu/www/nurisp/final_seminar/NURISP_overview.pdf

DOE, U.S., 2011. CASL Project Summary. http://www.casl.gov/docs/CASL-U-2011-0025-000.pdf.

Horváth, Ákos, Dressel, B., 2013. On numerical simulation of fuel assembly bow in pressurized water reactors. Nucl. Eng. Des. 265, 814-825.

In, W.K., Chun, T.H., Shin, C.H., Oh, D.S., 2008. Numerical computation of heat transfer enhancement of a PWR rod bundle with mixing vane spacers. Nucl. Technol. 161, 69-79.

Karoutas, Z., Gu, C., Sholin, B., 1995. 3-D flow analyses for design of nuclear fuel spacer. In: Proceedings of the 7th International Meeting on Nuclear Reactor Thermal-hydraulics (NURETH-7), New York, USA, 3153-3174.

Kim, K.Y., Seo, J.W., 2004. Shape optimization of a mixing vane in a sub-channel of nuclear reactor. J. Nucl. Sci. Technol. 41, 641-644.

Kim, K.Y., Seo, J.W., 2005. Numerical optimization for the design of a spacer grid with mixing vanes in a pressurized water reactor fuel assembly. Nucl. Technol. 149, 62-70.

Lee, C.M., Choi, Y.D., 2007. Comparison of thermo hydraulic performances of large scale vortex flow (LSVF) and small scale vortex flow (SSVF) mixing vanes in 17×17 nuclear rod bundle. Nucl. Eng. Des. 237, 2322-2331.

Li, X., Gao, Y., 2014. Methods of simulating large-scale rod bundle and application to a 17×17 fuel assembly with mixing vane spacer grid. Nucl. Eng. Des. 267, 10-22

Ma, T., 2012. Numerical Analysis of Thermal Hydraulic Characteristics of Spent Fuel Rod Assembly. Master dissertation of Harbin Engineering University.

Navarro, Moysés A., Santos, André A.C., 2011. Evaluation of a numeric procedure for flow simulation of a 5×5 PWR rod bundle with a mixing vane spacer. Prog. Nucl. Energ. 53, 1190-1196.

Nematollahi, M.R., Nazifi, M., 2008. Enhancement of heat transfer in a typical pressurized water reactor by different mixing vanes on spacer grids. Energ. Convers. Manage. 49, 1981-1988.

Saha, P., Aksan, N., Andersen, J., Yan, J., Simoneau, J.P., Leung, L., Bertrand, F., Aoto, K., Kamide, H., 2013. Issues and future direction of thermal-hydraulics research and development in nuclear power reactors. Nucl. Eng. Des. 264, 3-23.

Wells, D.M., Peturaud, P., Yagnik, S.K., 2015. Overview of

CFD Round Robin Benchmark of the High Fidelity Fuel Rod Bundle NESTOR Experimental Data. NURETH-16, Chicago, IL, 627-639.

Weber, D.P., Sofu, T., Yang, W.S., 2007. High-Fidelity Light Water Reactor Analysis with the Numerical Nuclear Reactor. Nucl. Sci. Eng. 155, 395-408.

Abstract

Abstract: CFD simulation for a PWR is an important part for the development of Numerical Virtual Reactor (NVR) in Harbin Engineering University of China. CFD simulation can provide the detailed information of the flow and heat transfer process in a PWR. However, large number of narrow flow channels with more and complex structures (mixing vanes, dimples, springs, etc.) are located in a typical PWR. To obtain a better understand of the CFD simulation, the challenges created by these structural features were analyzed and some quantitative regularity & estimation were given in this paper. It is found that both the computing resources and time are in great need for the CFD simulation of a whole reactor. These challenges have to be resolved, so two schemes were designed to assist/realize the reduction of the simulation burden on resources and time. One scheme is used to predict the combined efficiency of the simulation conditions (configuration of computing resources & application of simulation schemes), so it can assist the better choice/decision of the combination of the simulation conditions. The other scheme is based on the suitable simplification and modification, and it can directly reduce great computing burden. Therefore, both these two schemes will be useful to overcome the challenges in the CFD simulation of a whole reactor.