CFD Modelling of the Dry Spent Fuel Storage of a Pressurized Heavy Water Reactor

Ahmad Hussain[1*] Hani Sait[2]

Mechanical Engineering Department, King Abdulaziz University-Rabigh,PO box 344, Rabigh 21911, Saudi Arabia

* E-mail of the corresponding author: ahmadutm@gmail.com

Abstract

Currently, concrete canisters are in use for an interim storage of spent fuels of pressurized heavy water reactors. A major factor in spent fuel dry storage design and key thermal safety issue is the dissipation of the residual heat generated by the spent fuel. Also the spent fuel temperature, in a dry storage canister, must be kept below 160 ℃ to avoid fuel oxidation. In this research, CFD modeling has been done to evaluate the temperature distribution in radial and vertical directions of the spent fuel cask. The temperature distribution inside the fuel basket is obtained using ANSYS and FLUENT for CFD analysis. The results suggest that when the effects of conduction and convection are combined and the ambient temperature is taken as 40℃ then the maximum temperature of the fuel bundle is found out to be153℃ which is below the temperature limit of 160℃. Therefore, the proposed indigenous design of the spent fuel storage cask is safe enough to keep the temperatures of the spent fuel well within the limits.

Keywords:, CFD, FLUENT, fuel safety, modeling, spent fuel storage

1. Introduction

The pools that were designed initially for short-term storage have become quasi-permanent storages. Wet storage needs special care to maintain good water chemistry pH-values, chloride and sulphate impurity concentrations and conductivity. Furthermore it has been found that specified pool water temperature control is essential for long term spent fuel integrity and to avoid structural damage to the facility. Several sites have documented degradation/corrosion of the spent fuel assemblies in wet storage resulting mainly from poor water chemistry control. This degradation of the spent nuclear fuel, in turn, produces several problems that ultimately result in an increased cost for continued storage and management. Also, wet storing of spent nuclear fuel was never intended to be permanent.

Therefore, dry storage of spent fuel becomes a viable option after it has cooled to the point where passive heat transfer from encapsulated fuel to its environment is efficient enough to insure that the fuel element temperatures are well below values which would lead to significant degradation over the long-term.

A major factor in spent fuel dry storage design and one of the key thermal safety issues for licensing a spent fuel dry storage system is dissipation of the residual heat generated by the spent fuel. That is, the spent fuel temperature in a dry storage canister must be kept below 160 ℃ to avoid fuel oxidation, Iqbal, M.,et.al, 2006.

The CANDU spent fuel cask is a cylindrical reinforced concrete shell with a capacity to store 9 sealed fuel baskets, each containing 54 CANDU 19-element spent fuel bundles. The orthographic views and isometric views of the spent fuel basket are shown in Figure 1 and Figure 2.

In this design, the atmosphere in the storage basket is air. Fuel bundles are stored in staggered arrangement in the fuel basket. The fuel bundles are to be kept in position by two fixed perforated plates (Figure 3). The basket is perfectly sealed to prevent any releases of radioactive material.

Spent Fuel

- Type: 19-element fuel bundle
- Material: UO_2 (natural U)
- Height: 495 mm
- Diameter: 81.5 mm

Fuel Basket

- Material: Stainless steel (SS-304L)
- Height: 546 mm
- Diameter: 813 mm
- Plate dia: 787 mm
- Hole dia: 82.5 mm.

Also, the minimum cooling time in the spent fuel bay is 10 years. The decay heat power of 10-year cooled spent fuel bundle is 5W at the design burnup of 9,000 MWT/MTU. In order to evaluate the maximum fuel rod temperature in the CANDU dry storage canister under design conditions, one may need to solve a multi-dimensional heat transfer problem with an extremely complicated geometry where three modes of heat transfer are superimposed, Armijo, J.S., et.al,2006

There are three modes of heat transfer that are to be considered in a spent fuel storage cask: conduction, convection and radiation. Conduction and Convection are the dominant modes of heat transfer in a cask. Radiation heat transfer becomes significant when the temperature of the fuel is very high.

In this analysis, only convective heat transfer is taken into account. Since the fuel basket is completely sealed therefore there is no entrance of air into the fuel basket and forced convection cannot take place inside the basket, therefore heat is convicted out of the basket only by natural convection.
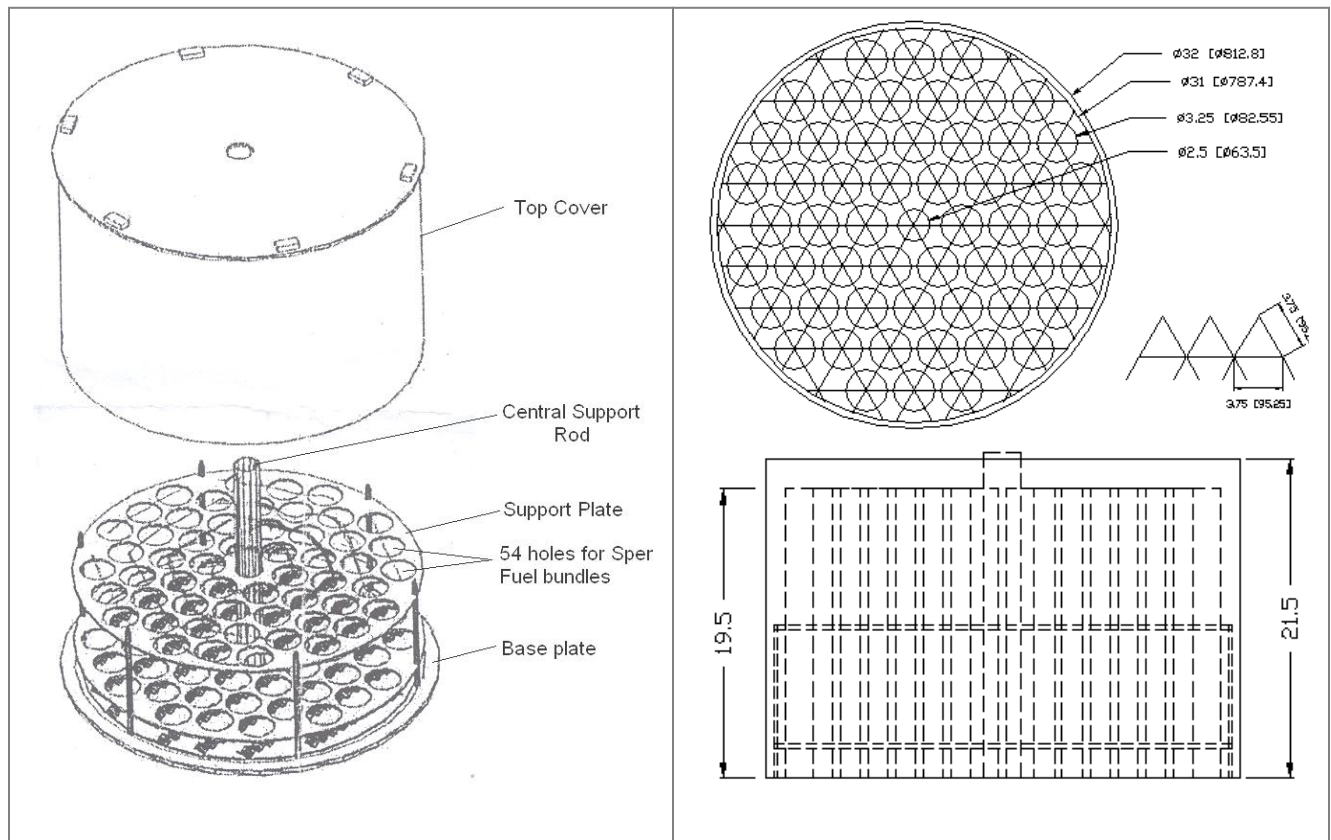
Figure 1:   Isometric view of the CANDU spent fuel
            cask for storing 54 bundles.

Figure 2:   Orthographic views of spent fuel basket

2.   Methodology

2.1   Defining the Modeling Goals

Natural convection heat transfer inside the fuel basket is to be modeled incorporating the effects of buoyancy. The boundary conditions are: constant heat flux on every fuel bundle's surface and constant temperatures on top, side and bottom surfaces. Boussinesq model is used for density variations and temperature dependency of thermal properties is also taken into account.

2.2   Creating the Model Geometry and Grid

This step of the solution process requires a geometry modeler and grid generator. For this purpose, **Pro-Engineer** and **GAMBIT** are used for geometry modeling and grid generation respectively.

Using Pro-Engineer, model geometry of the fuel basket is developed, which is then imported to GAMBIT as IGES file where it's meshing is done.

In GAMBIT, the top and bottom surfaces are meshed using Quadrilateral type element and the whole volume is then meshed using Hexahedral elements of Cooper type.

Finally, following five zones are defined for applying boundary conditions:

1. Fuel Bundles side surfaces named "const heat flux".

2. Fuel Bundles top surfaces named "fuel rods top".

3. Basket side wall named "const temp side".

4. Basket top surface named "const temp top".

5. Basket bottom surface named "const temp

These meshed drawings are saved as mesh (*.msh) files. These mesh files are then imported into FLUENT where the CFD analysis is done.

3.   Solid Modeling CFD details

3.1   *Importing and Scaling the grid*

In FLUENT, the mesh file generated by GAMBIT is imported and its grid is scaled.

3.2   *Setting the numerical solver*

First, the numerical solver is set to the following specifications:
- Solver type: Pressure based
- Formulation: Implicit
- Space: 3D
- Time: steady
- Velocity formulation: Absolute
- Gradient option: Green-Gauss cell based

3.3 Selecting the physical model
The Energy model is selected as the physical model.

3.4 Defining material properties
The working fluid inside the fuel basket is air. Its thermal properties like viscosity, thermal conductivity and specific heat are taken temperature dependent. The variations of these properties with temperature are modeled by using appropriate polynomials. The coefficients of those polynomials are then entered into FLUENT. For density variations, Boussinesq model is used, Heng, X.,et, al. 2006.

3.5 Prescribing operating conditions

The operating pressure is set to normal atmospheric pressure (i.e.101325 Pa). The effect of gravity is incorporated in the model by setting the gravity value equal to -9.8 along y-axis. The Boussinesq operating temperature is set to 393 K.

3.6 Applying boundary conditions

Following boundary conditions are applied to the above-defined five zones:
- Set "const heat flux":   Heat Flux = 37.853 W/m^2
- Set "const heat flux":   Heat Flux = 37.853 W/m^2
- Set "fuel rods top":   Heat Flux = 37.853 W/m^2
- Set   "const temp side": Temperature = 393 K

- Set   "const temp top":   Temperature = 400 K
- Set   "const temp bottom": Temperature = 393 K

3.7 Setting up solver controls

The solver controls are set to the following settings:
- Equations: Flow and Energy
- Pressure-velocity coupling: SIMPLE
- Under-Relaxation factors:
    – Pressure = 0.3
    – Density = 1
    – Body Forces = 1
    – Momentum = 0.7
- Discretization:
    – Pressure = PRESTO!
    – Momentum = Second Order Upwind
    – Energy = Second Order Upwind

3.8 Setting up convergence monitors

The convergence monitors are set appropriately and the plotting of residuals (continuity, x-velocity, y-velocity, z-velocity, energy) during the calculations has been enabled.
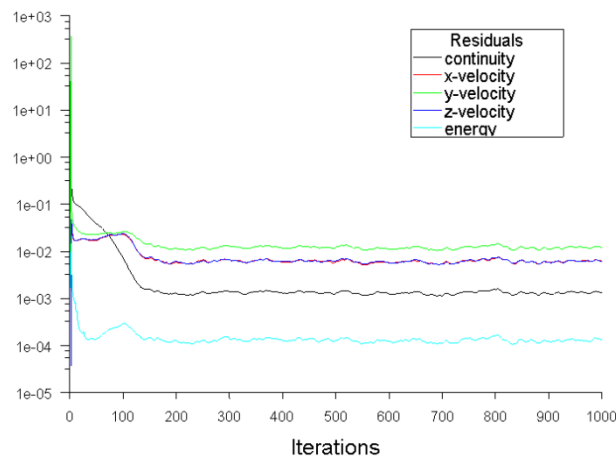

Figure 3: Convergence pattern of the residuals

3.9 Initializing the flow field

The calculations are stared by giving the number of iterations with all velocities = 0 and temperature = 395K.

3.10 Computing and Monitoring the Solution

The discretized conservation equations are solved iteratively. A number of iterations are usually required to reach a converged solution.

Convergence is reached when:

1. Changes in solution variables from one iteration to the next are negligible. Residuals provide a mechanism to help monitor this trend.
2. Overall property conservation is achieved.

The convergence pattern of the residuals for the given problem is shown in Figure 3.

4.   Results and Discussion

After the convergence of monitored variables, the data can be displayed in the form of contours and vectors of temperature and velocity to view the temperature distribution and flow patterns.

For the given problem, three additional planes are defined for the display of data: xy-plane, yz-plane and xz-plane. These planes pass through the point of origin.

Figure 4 is a temperature contour diagram showing that the temperature is increasing from the bottom to the top of the fuel bundles. This is due to the fact that, as the air is heated, it moves upward and cold air comes down, thus the temperature at the bottom (120°C) is lower than that at the top (180°C).
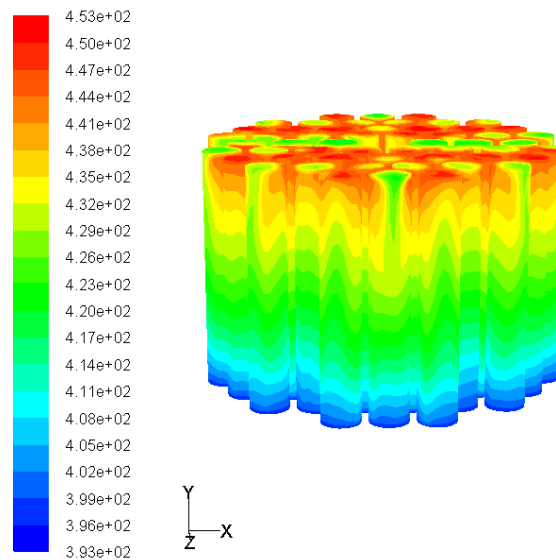

Figure 4: Side view of temperature contour diagram of fuel bundles

Figure 5 and 6 shows the top view of the temperature contour diagram which reveals that temperature is very high in those regions where the fuel bundles are more clustered. This is due to the fact that heat is highly accumulated in those regions, thus the temperature goes very high in those regions (i.e. 180°C).

Figure 7 shows the velocity vectors at the top surface of fuel bundles while Figure 8 shows contours of y-velocity at xy, yz and xz-planes. These figures clearly demonstrate the movement of air inside the fuel basket. Air is coming upward with maximum velocity (i.e. 0.238m/s) from within the annular spaces of fuel bundles while it is going down along the basket wall as well as along the central rod.
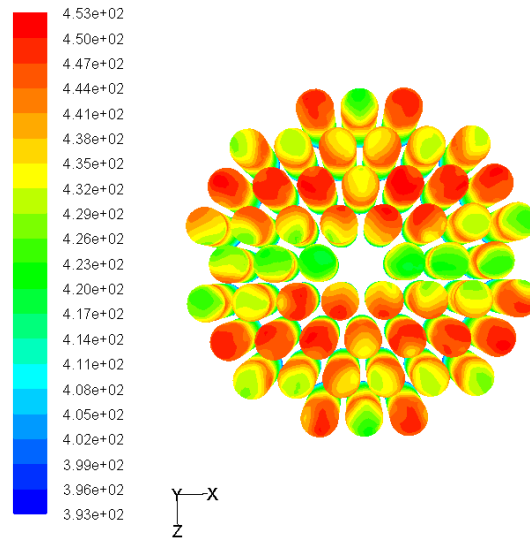
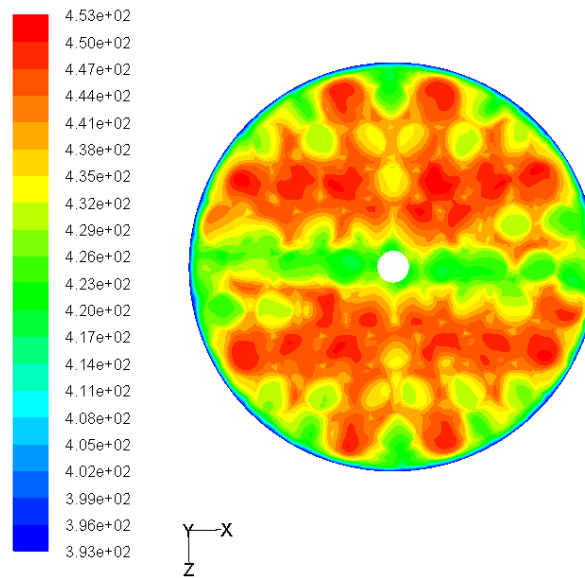*Figure 5: Top view of the temperature contour diagram of fuel bundles*

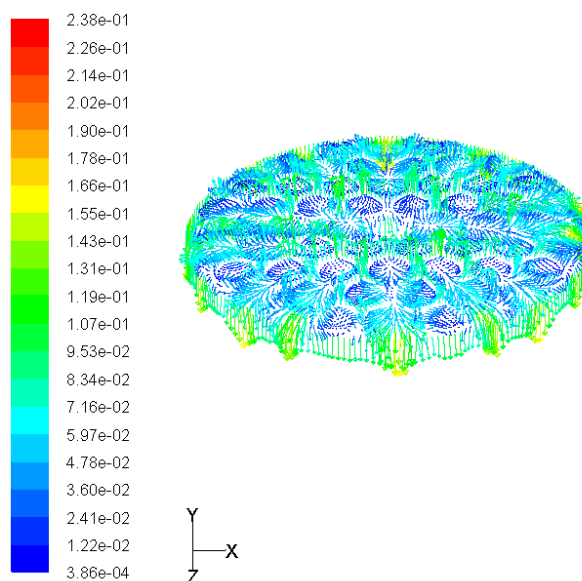Figure 6: Temperature contours at xz-plane

17

Figure 7:   Velocity vector diagram at the top of fuel bundles

Also, it can be seen that the air above the fuel bundles is mostly static. The boundary layer formation can be visualized from these diagrams. The xy-plane view shows the boundary layer formation at the basket wall while the yz-plane view shows the boundary layer formation at the central portion of the fuel basket. The xz-plane view shows that the air above the fuel bundles is static but it is coming up from the annular spaces of the fuel bundles (as shown by red-colored portions) and going down along the basket wall (as shown by blue-colored portions).

Table 1 gives the summary of CFD analysis, as reported by FLUENT. Some quantities of major concern like heat transfer rate, heat transfer area, flow area and fluid volume are given in the table. The maximum, minimum and mass-weighted average values of certain major parameters are also incorporated in the same table.

5. Conclusion

The maximum fuel rod temperature at some points is about 180oC which is beyond the limit (160oC). However, in this CFD analysis the conduction and radiation heat transfer were not given due consideration in some sections of the dry fuel storage geometry which could have resulted in lower fuel rod temperatures. It is expected that if conduction and radiation effects are incorporated in the analysis of all the sections of dry fuel, then heat transfer will improve and temperature will certainly reduce because these modes also play a major role in the dissipation of heat from the fuel bundles. However, the current CFD analysis has given an insight into the heat transfer that has given confidence in designing such cask.

References

Armijo, J.S., Kar, P. and Misra, M. (2006) "Second generation waste package design and storage concept for the Yucca Mountain Repository", Nuclear Engineering & Design, 236 : 2589-2598.

Heng, X., Zuying, G. and Zhiwei, Z. (2006) "A numerical investigation of natural convection heat transfer in horizontal spent-fuel storage cask", Nuclear Engineering and Design,. 213: 59-65.

Holman, J.P.   (2002) Heat Transfer , 9th   Edition, MCGraw-Hill, USA

Iqbal, M., Khan, J. and Mirza, S.M. (2006) "Design study of a modular dry storage facility for typical PWR spent fuel", Progress in Nuclear Energy. 48: 487-494.
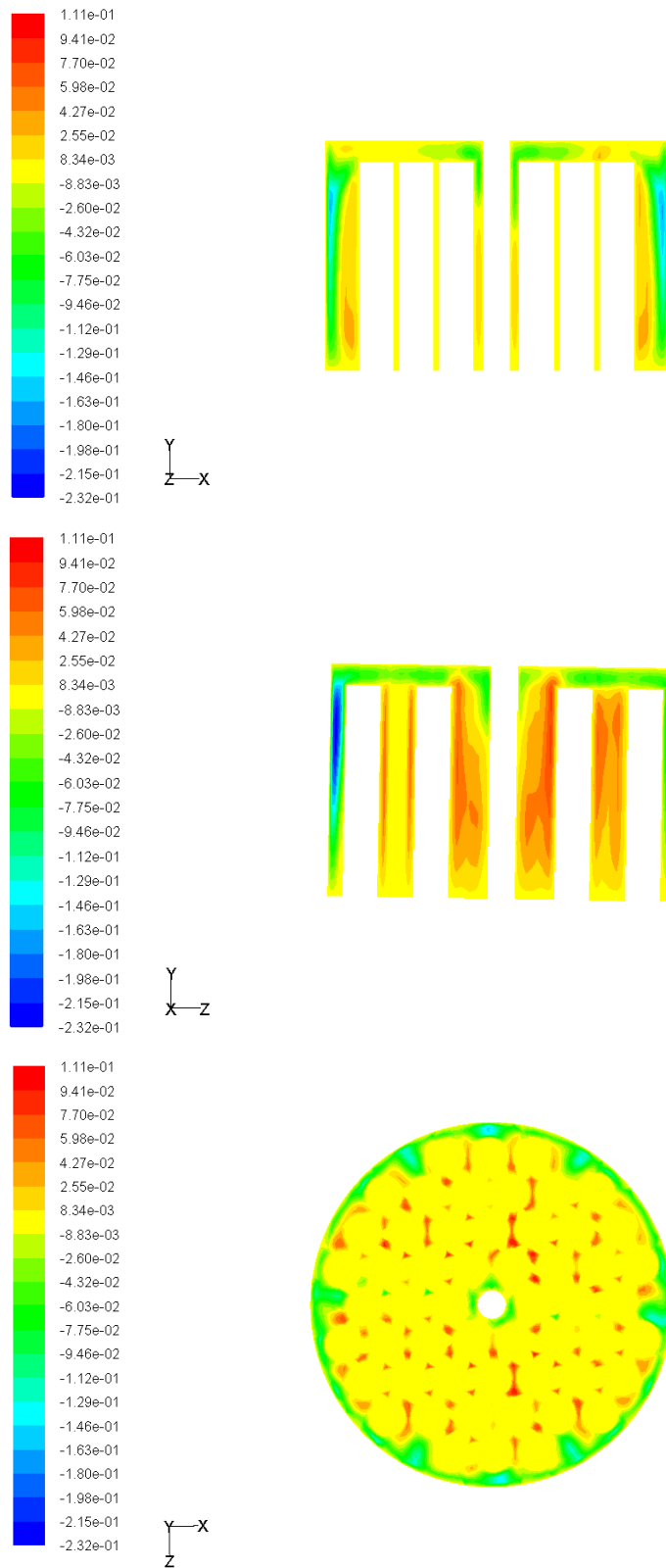
Figure 8: Contours of Y-velocity at xy, yz and xz-planes

Table 1:   Summary of CFD analysis

FLUENT Reported Data

Heat Transfer Rate (W)			269.24
Heat Transfer Area (m^2)			7.113
Total Flow Area (m^2)			0.2368
Total Fluid Volume (m^3)			0.1434
	Maximum	**Minimum**	**Mass-weighted average**
Static Temperature (K)	453	393	420
X – velocity (m/s)	0.120	0.0000434	-
Y- velocity (m/s)	0.116	0.00746	-
Z – velocity (m/s)	0.119	0.00363	-
Total Velocity (m/s)	0.2376	0.000386	0.056
Prandtl Number	2.926	2.265	2.566

This academic article was published by The International Institute for Science, Technology and Education (IISTE). The IISTE is a pioneer in the Open Access Publishing service based in the U.S. and Europe. The aim of the institute is Accelerating Global Knowledge Sharing.

More information about the publisher can be found in the IISTE's homepage:
http://www.iiste.org

CALL FOR PAPERS

The IISTE is currently hosting more than 30 peer-reviewed academic journals and collaborating with academic institutions around the world. There's no deadline for submission. **Prospective authors of IISTE journals can find the submission instruction on the following page:** http://www.iiste.org/Journals/

The IISTE editorial team promises to the review and publish all the qualified submissions in a **fast** manner. All the journals articles are available online to the readers all over the world without financial, legal, or technical barriers other than those inseparable from gaining access to the internet itself. Printed version of the journals is also available upon request of readers and authors.

IISTE Knowledge Sharing Partners

EBSCO, Index Copernicus, Ulrich's Periodicals Directory, JournalTOCS, PKP Open Archives Harvester, Bielefeld Academic Search Engine, Elektronische Zeitschriftenbibliothek EZB, Open J-Gate, OCLC WorldCat, Universe Digtial Library , NewJour, Google Scholar