राष्ट्रीय प्रौद्योगिकी संस्थान दुर्गापुर
(मानव संसाधन विकास मंत्रालय, भारत सरकार के अधीन राष्ट्रीय महत्व का संस्थान)
महात्मा गांधी एभेन्यू, दुर्गापुर - 713209,(पश्चिम बंगाल), भारत

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR

(Institute of National Importance Under MHRD, Govt. of India)
MAHATMA GANDHI AVENUE, DURGAPUR-713209, (WEST BENGAL), INDIA

Ref.No. NITD/RTI/CPIO/OFF/168

Date: 14.10.2019

Subject:- Information under RTI Act, 2005.

Ref. :- Your letter dated 20/09/2019 seeking information under RTI Act 2005.

Sir,

In connection with above mentioned subject and reference, the available information's are as under:-

Sl. No.	Query	Reply
01.	Your queries	Available Information is enclosed as Annexure – "A" (page-01-75) & Annexure-"B"(page-01-02)

14.10.19

Assistant Registrar & CPIO

Enclo: as above.

NATIONAL INSTITUTE OF TECHNOLOGY,DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR-713209

No. NITD/ RTI/Estt./3216/ 219 Dated:- 11th October , 2019

Inter Departmental Memo.

From:-
Dy.Reg. (Estt.)

To
The Assistant Registrar &
CPIO

Sub:-Information sought by , under RTI Act-2005.

Ref. NITD/RTI/ CPIO/ OFF/ 168 dated 24.09.2019 .

The following information is hereby provided in respect of RTI application of . Under RTI Act-2005.

Reply of information sought No 01:- The photocopy of application form as duly filled in by , Department of Mechanical Engineering of NIT Durgapur for mapping from the post of Professor, AGP of Rs.10,500/- is enclosed as Annexure-A, page 01 to 75.
Reply of information sought No 02:- The photocopy of Office Order No.NITD/Estt/HAG-faculty/2018 dated April 24th , 2018 issued to , Department of Mechanical Engineering of NIT Durgapur for mapping from AGP Rs. 10,000/- to AGP Rs. 10,500/- is enclosed as Annexure-B, page01 to02.

11/10/19
Dy.Reg. (Estt.)

JKR

[Annexure - A] [page - 01]

November 30, 2017

From: Dr. Amar Nath Mullick
Professor
Department of Mechanical Engineering
National Institute of Technology
Durgapur – 713 209

To The Registrar
National Institute of Technology
Durgapur – 713 209

Through Dean, Faculty Welfare

Sub.: Application for Mapping of Existing Professor with AGP of Rs. 10,000/-

Reference: Advertisement No. NITD/Estt./02/09/2017 dated 25/09/2017

Dear Sir,
Please find enclosed herewith the filled-in Application From for the Post of Professor, Mechanical Engineering for mapping of existing Professor with AGP of Rs. 10,000/- to Rs. 10,500/- as per the Advertisement cited above.

Kindly acknowledge the receipt of the same.

Regards.

30/11/17

Forwarded
A. Gangopadhyay 30/11/17

Prof. Aniruddha Gangopadhyay

Prof. Aniruddha Gangopadhyay
Dean (Faculty Welfare)
National Institute of Technology
Durgapur - 713209, INDIA

(AMAR NATH MULLICK)
Professor
Department of Mechanical Engineering
National Institute of Technology
Durgapur – 713 209
Emp-Id – 4P – 917

Encl.: a) Filled-in Application Form in the prescribed format
Photocopy of the First page of Journals & Conference Proceedings
Filled-in Data Sheet.

राष्ट्रीय प्रोद्योगिकी संस्थान, दुर्गापुर

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR-713209
WEST BENGAL, INDIA, www.nitdgp.ac.in
(An Autonomous Institution of the Govt. of India under MHRD)

Advertisement No.: NITD/Estt./02/09/2017 Date: 25.09.2017

(Note: Incomplete applications are liable to be rejected)

PostAppliedfor& AGP		DD Number	
Department		Amount&Date	
Specialization		Name of issuing bank	

1.	Name in full (in block letters)		AMAR NATH MULLICK				
2.	Father's/Husband's Name		Late Bhola Nath Mullick				
3.	Mother's Name		Smt. Saraju Mullick				
4.	Date of Birth		January 12, 1958				
	Age as on 06.11.2017		59	Year	10	Month	15 Day
5.	a)Marital Status: ~~Married~~/Unmarried		b)Gender: Male/~~Female~~				
6.	a)Permanent address: 77/ 1 B Sri Gopal Mullick Lane, Kolkata 700 012		b)Correspondence address: G + 8 Apartment, Flat No. 504, NIT Campus, Durgapur 713 209				
	Phone (with STD)/ Mobile No.	9830385342/9434788052	E-mail	anmullick@gmail.com/ amarnath.mullick@me.nitdgp.ac.in			
7.	Nationality Indian						
8.	Category under which seeking reservation/relaxation		~~SC/ST/OBC/PWD/WOMEN~~				

9. Educational Qualification (from Matriculation onwards):

Exam. passed	Specialization	Board/University	Passing year	Class/ Division	%marks/ CGPA
Higher Secondary	Science	WBSE	1973	First	60.1
B M E	Mechanical Engg	Jadavpur University	1979	First Class	78.4
M M E	Hydraulics & Fluid Mechanics	Jadavpur University	2000	First Class	73.4
Ph D Apr 02 – Apr 07	Mechanical Engineering	Jadavpur University	2007		

* For PhD Degree mention the duration/period.

SignatureoftheApplicant

10. Detailsofemployments:

(a)Teaching: Total: 17 Years Post PhD: 10 Years 7 Months

Sl.No.	Name and address of employer	Designation	Pay-scale	From	To	Duration	Type of organization
1	National Institute of Technology	Professor	37,400-67,000 with	01.02.13	Till date	Two years	Educational Institute
2	-do-	Associate Professor	37,400-67,000 with	01.02.10	31.01.13	Three years	-do-
3	-do-	Assistant Professor	12,000-420-18,300	01.02.07	31.01.10	Three Years	-do-
4	Future Institute of Technology,	Assistant Professor	12,000-420-18300	16.08.05	31.01.07	One & half years	-do-
5	Netaji Subhash Engineering	Senior Lecturer	10,000-325-15,200	01.02.02	15.08.05	Three & half years	-do-
6	Bengal Institute of Technology	Part-time Faculty		19.10.2K	31.01.02	One & Half Year	-do-

11. Other information (Experiments/ Computational projects added to teaching laboratories/ Courses offered through application of ICT/E-learning packages prepared):
Experiments in newly developed Aerodynamics Lab done by the UG students as a part of their Final Year Project and validate their result through Computational Simulation

12. Research activities:(Specify total Number and attach list of publication in each category as per format given**):**

(a)Paper publications in National/ International Journals (SCI/ SCOPUS indexed journals only):

(Attach photocopy of first page) **See Annexure I**

Sl.No.	Title of paper	Author(s)	NameoftheJournal	Vol.&Year	Pages

Signatureof Candidate

(b)Paper Publications in Conferences (SCI/SCOPUS/Web of Science/Internationally renowned conference):

(Attach photo copy of first page) **See Annexure II**

Sl.No.	Title of paper	Co-author(s), ifany	Name of the Conference	Date

(c)Number of UG and PG students guided / Ongoing:

UG		PG	
Guided	Ongoing	Guided	Ongoing
40	5	5	1

(d) Details of PhD Students (Guided and Ongoing): See Annexure III

Sl.No	NameofStudent	Year	TitleoftheDissertation	Status(Completed /Ongoing)

13. Books/ Monographs/Book chapters written:

Sl.No.	Name of book/monograph/ Book chapters	Name of Co-author(s),If any	Year of Publication	Publisher with address
1	Studies of Flow through air intake using Artificial Intelligence	Sole Author	2012	Lap Lambert Academic Publishing,

14. Seminars/Short Term Courses /Summer Schools/ Winter Schools organized:

Sl. No.	From	To	Name of the course	As Chairman /Coordinator	Numberof Participants
1			EARSE 09	Organising Secretary	32
2			NaCoMM 09	Conference Secretary	80

15.Seminars/Short Term Courses/Summer Schools/Winter school attended, if any

Sl.No.	From	To	Institute/Organization	Sponsored by	Name of the course
1	19.08.13	23.08.13	IIM Lucknow	MHRD	Management Capacity Enhancement
2	2.2.17	8.2.17	IIM Indore	MHRD	Do

16.Research Projects/Sponsoredproject/Consultancyactivities:

Sponsoring Agency	Title of the Project	Period	Amount	Status (Completed/ ongoing)

17.Membership of Professional Bodies:

Sl.No.	Name of Professional body	Membership no. with validity
1	Institute of Engineers (India)	F – 018176 -1 Fellow, Life
2	American Society of Mechanical Engineers	M – 8185555 Member, Annual
3	Institute of Electrical & Electronics Engineers	M – 90499913 Senior Member,
4	National Society of Fluid Mechanics & Fluid Power	M - 154 Member, Life
5	Assiciation of Machines & Mechanism	Member, Life

18.Administrative/InstituteSupportwork: See Annexure IV

Sl.No.	Section /office/Institute level committee	From	To	Positionheld	Responsibilities

19.Any otherrelevantinformation:(Addseparatesheetifrequired)

- Reviewer of the Journals
  - Journal of Mechanical Science & Technology
  - Journal of Applied Fluid Mechanics
  - Conferences organized by ASME.
  - Project Reports of Kerala State Council for Science, Technology and Environment
- Act as Resource Person at

Jadavpur University

Durgapur Institute of Advanced Technology & Management

- Act as Examiner
  - Jadavpur University
- Act as Advisor
  - Tripura Public Service Commission

20.Nameandaddressoftwo References:

1st Referee		2nd Referee	
Name	Prof Bireswar Majumdar	Name	Prof. Sivaji Chakravorty
Position	Professor	Position	Director, NIT, Calicut
Address	Power Engineering	Address	NIT Calicut
E-Mail	b_maj3255@yahoo.com	E-Mail	s_chakravorti@ieee.org
PhoneNo.	+919433213671	PhoneNo.	
E-MailAddress		E-MailAddress	

21.Did you previously apply against earlier advertisement(s)? If yes, give particulars: No

....

DECLARATION

"I hereby declare that the statements made by me in above form are true, complete and correct to best of my knowledge and belief."

Place: Durgapur
Date: November 30, 2017

Signature of Applicant

Name: Prof. Amar Nath Mullick

Page - 67

राष्ट्रीय प्रोद्योगिकी संस्थान, दुर्गापुर
NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR-713209

DATA SHEET
(To be filled by the candidate)

Post Applied for:	Name of Post	AGP
	Professor	
Department:	Mechanical Engineering	
Specialization:	Fluid Mechanics & Hydraulic Machines	

(A) PERSONAL DATA

1	Name and Address with e mail id	Dr. Amar Nath Mullick, 77/1 B Sri Gopal Mullick Lane, Kolkata 700012, anmullick@gmail.com
2.	Date of Birth/Age as on 06-11-2017	January 12,1958
3.	Category (SC/ST/OBC/UR)	PwD : ~~Yes~~/ No Gender: Male/~~Female~~

4.	Education Qualifications:						
	Exam/Degree	Specialization	College/ Institute	University	% of marks/CGPA	Class/ Grade	Year
	Class X						
	Class XII	Science	Hare School		60.1	First Div	1973
	UG	Mechanical Engg	Jadavpur University		78.4	1st Class (Hons)	1979
	PG	Fluid Mech.	-do-		73.8	1st Class	2000
	Ph.D.		-do-				2007
	Others, if any						

5.	Period spent on acquiring Ph. D	5 Years
6.	Post- Doctoral Specialization	
7.	(a) Present Position, with PB	Professor PB 4
	(b) Date of entry into present position position	

	Total Experience	Years: 38 Years 3 Months	Post Ph.D: 10 Years 7 Months
	Teaching Experience	Years: 17 Years	Post Ph.D: 10 years 7 Months
	Research Experience	Years: 17 Years	Post Ph.D: 10 years 7 Months
	Industrial Experience	Years: 21 Years	Post Ph.D:
9	Number of years of post-Ph.D experience outside NITD(for Ph.Ds of NITD)		

10	Publication details	International	National
	Papers in SCI/Scopus indexed Jls.	8	
	Conference Proceedings(SCI/Scopus indexed /Web of science	23	
11	No. of Thesis guided	UG: 40	PG: 5
12	No. of Ph.D thesis	Completed: 2 Submitted: 1	Ongoing: 4
13	No. Books/chapters authored	Books: 1	Book Chapters:
14	No. of patents	Granted:	Filed:
15	R & D Projects/Consultancy jobs	No. of R&D Projects:	Total Amount:
		No. of consultancy jobs:	Total Amount:
16	Fellowship in Professional body	Institute of Engineers (India)	
17	No. of awards/distinction, if any		
18	Cumulative Credit points	130 (114)	
19	Any other relevant information		

Place:Durgapur

Date: November 30, 2017

Signature of Applicant

[Page-08]

NO OBJECTION CERTIFICATE TO BE FURNISHED BY THE CANDIDATE WHO IS ALREADY IN EMPLOYMENT,REGULAR/TEMPORARYBASIS.

Certified that Mr.Amar Nath Mullick Son of Late Bhola Nath Mullick is a permanent employee of the department of Mechanical Engineering of National Institute of Technology, Durgapur since February 01, 2007. Department has no objection if he is appointed in National Institute of Technology Durgapur against the posts advertised, vide advertisement NoNITD/Estt./02/09/2017 dated 25.09.2017

Place:-Durgapur –Date: November 30, 2017

Signature & Seal of forwarding Authority

Page -10

(B) DATA FOR EVALUATION OF CREDIT POINT. *(See the Modified Recruitment Rule for guidelines)*

Post Applied:	Name of Post	AGP
	Professor	Rs 10,000/-
Department:	Mechanical Engineering	
Specialization:	Fluid Mechanics & Hydraulic Machines	

(1). **Externally sponsored R & D projects completed or ongoing / Patent granted:**

Sl. No.	Name of the project along with the name(s) of all the investigators/Patent granted with name of inventor(s)	Duration		Amount in Lakhs	Whether PI?	Total No. of investigators	Credits
		Starting Month & Year	Ending Month & Year				
1							
2							
	Total Credit points						

(2). **Consultancy projects:**

Sl.No.	Name of the work and the year	Agency	Name of the persons involved	Consultancy amount in Lakhs	Credits
1					
2					
	Total Credit points				

(3). **Ph.D. Guidance (Completed including thesis submitted):**

Sl.No.	Title of the Thesis	Name of supervisor(s)	1st Supervisor?	Total No. of supervisors	Date of award/Submission /Viva voce	Credits
1	Flow Investigation through Annular Curved Diffuser	A N Mullick, B Haldar, B Majumdar	A N Mullick	3	2011	5
2	Development of Flow Through Constant Area Duct in Turbulent Regimes	A N Mullick, P K Sinha, B Majumdar	A N Mullick	3	2015	5

3	Numerical Study on the Thermofluidic Transport of Fly Ash-Water Slurry in Horizontal Pipeline	A N Mullick, D Chatterjee	A N Mullick	2	Thesis Submitted 2017	5
Total Credit points						15

(4). Paper Publications in non-paid SCI/Scopus indexed Journals only (Since the last promotion):

Sl. No.	Name of the Journal	Title of the paper with Vol. No. & year	Author (s) Name	First author/ Main Supervisor?	Total No. of authors	Credits
1	International Journal of Thermal Science	Numerical Prediction of Flow and Heat Transfer Characteristics of water-fly Ash Slurry in a 180° return pipe bend, Vol. 113, 2017	B B Nayek, D Chaterjee & A N Mullick	A N Mullick	Three	2
2	Multiphase Science & Technology	Numerical Simulation of Convective Transport of Fly ash-water slurry in Horizontal pipe bend, Vol. 27, 2015	B B Nayek, D Chaterjee & A N Mullick	A N Mullick	Three	2
3	Journal of Computational Multiphase Flows	Numerical analysis of convective transport of fly ash-water slurry through a horizontal pipe, Vol. 7 2015	B B Nayek, S Gupta, D Chatterjee & A N Mullick	A N Mullick	Four	2
4	International Review of Mechanical Engineering	A Computational Analysis of Flow Structure through a constant area S duct, Vol. 8, 2014	A K Biswas, P K Sinha, A N Mullick & B Majumdar	A N Mullick	Four	2
5	International Review of Aerospace Engineering	A Computational Analysis of Flow Structure through a constant area S duct, Vol. 6, 2013	A K Biswas, P K Sinha, A N Mullick & B Majumdar	A N Mullick	Four	2
6	International Review of Mechanical Engineering	CFD Investigation of Flow through a Constant Area Curved Duct, Vol. 6, 2012	A K Biswas, P K Sinha, A N Mullick & B Majumdar	A N Mullick	Four	2

7	Journal of Aerospace Engineers, Institute of Engineers (India)	Flow investigation through an annular diffuser in subsonic flow regime, Vol. 90, 2009	P K Sinha, A N Mullick, B Halder & B Majumdar	A N Mullick	Four	2
8	International Journal of Turbo & Jet-Engines	An experimental Investigation on the Airflow in Twin Submerged Side Intake at Subsonic Speed, Vol 24, 2007	A N Mullick, S Chakravorty & B Majumder	A N Mullick	Three	2
Total Credit points						16

(5).**Paper Publications in Conferences (SCI/SCOPUS/ Web of Science/ Internationally renowned conference):**

Sl.No.	Name of the Conference	Title of the paper with Date	Author (s)Name	First author/ Main Supervisor?	Total No. of authors	Credits
1	20th International Conference on Aerospace & Mechanical Engineering	Numerical Prediction of Flow Behaviour of Coal Water and Copper Ore Water Slurries, June 2018	R Mishra, K C Ghanta, A N Mullick & S L Sinha	A N Mullick	Four	0.6
2	5th International Conferences on Advances in Civil, Structural and Mechanical Engineering, CSM 2017	Numerical Prediction on Two Phase Flow Through an 180° Pipe Bend, Sept 2017	B B Nayek, D Chatterjee & A N Mullick	A N Mullick	Three	0.6
3	41st National & 5th International Conference on FMFP 2014 to be held at IIT Kanpur	Numerical Investigations of Heat Transfer between wall and water-fly ash slurry flow in Horizontal pipes, Dec 2014	B B Nayek, S Gupta, D Chatterjee & A N Mullick	A N Mullick	Four	0.6
4	International Conference on Advances in Refractories and Clean Steel Making at Ranchi	CFD Investigation of Flow in a Continuous Casting Tundish, June 2013	Deepak Verma & A N Mullick	A N Mullick	Two	0.6
5	International Conference on Recent Advances in Mechanical Engineering held at Chennai	A Comparison between C-shaped Constant Area Duct and Diffuser using CFD Analysis, April 2012	A K Biswas, Ashoke K Raman N K Das A N Mullick P Ray B Halder & B Majumdar	A N Mullick	Seven	0.6
6	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012) held at Singapore	A Numerical Analysis of Flow Development through a Constant Area Duct, March 2012	A K Biswas, Ashoke K Raman A N Mullick & B Majumdar	A N Mullick	Four	0.6
7	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012)	Numerical Validation of Flow through S-shaped Diffuser, March 2012	N K Das A K Biswas, Ashoke K Raman A N Mullick B Halder P Ray & B Majumdar	A N Mullick	Seven	0.6

Page-13

8	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012)	Ana s of Flow Performance through Annular Curved Diffusing Duct using CFD, March 2012	P K Sinha A N llick B Halder B Majumdar	A N Mullick	Four	0.6
9	International Conference on Design and Advances in Mechanical Engineering held at Tiruvannamalai	A Comparative Study of Turbulence Models Performance through a Curved Diffuser in Turbulent Flow Regime, Dec 2011	N K Das A K Biswas, Ashoke K Raman A N Mullick B Halder P Ray & B Majumdar	A N Mullick	Seven	0.6
10	4th International Meetings of Advances in Thermofluids (IMAT 2011), Malaka, Malaysia	Wall Y^+ Approach for Dealing with Turbulent Flow Through a Constant Area Duct, Oct 2011	Isha Shukla, S S Tupkari, Ashoke K Raman & A N Mullick	A N Mullick	Four	0.6
11	4th International Meetings of Advances in Thermofluids (IMAT 2011), Malaka, Malaysia	Numerical Investigation of flow through a Curved Annular Diffuser, Oct 2011	P K Sinha A N Mullick B Halder B Majumdar	A N Mullick	Four	0.6
12	5th International Conference on Advance Mechanical Engineering at SVNIT, Surat	A Numerical Analysis of Flow Development through a Constant Area Duct, June 2011	Ashoke K Raman A K Biswas & A N Mullick	A N Mullick	Three	0.6
13	5th International Conference on Advance Mechanical Engineering at SVNIT, Surat	Numerical Investigation of flow through Annular Curved Diffusing Duct, June 2011	P K Sinha A N Mullick B Halder B Majumdar	A N Mullick	Four	0.6
14	International Conference on Modeling, Optimization & Computing (ICMOC 2010), at NIT Durgapur, India	Characteristics of Fluid Flow through Microchannels, Oct 2010	Meenkshi Mour, Debarun Das & A N Mullick	A N Mullick	Three	0.6
15	International Conference on Modeling, Optimization & Computing (ICMOC 2010), at NIT Durgapur, India	Flow Investigation through Annular Curved Diffusing Duct, Oct 2010	P K Sinha A N Mullick B Halder B Majumdar	A N Mullick	Four	0.6
16	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	Flow Investigation through annular curved diffuser, Dec 2010	P K Sinha A N Mullick B Halder B Majumdar	A N Mullick	Four	0.6
17	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	Flow development in a constant area duct, Dec 2010	A K Biswas, A N Mullick & B Majumdar	A N Mullick	Three	0.6
18	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	An experimental investigation on a flow through a curved diffuser, Dec 2010	N K Das A N Mullick B Halder P Ray & B Majumdar	A N Mullick	Five	0.6
19	1st International Conference on New Frontiers in Biofuels, at Delhi Technological University, India	Biofuels – a healthier and wealthier future for India, January 2010	Shankarnsh Srivastava Nirupam Kar & A N Mullick	A N Mullick	Three	0.6
20	ASME International Mechanical Engineering Congress & Exposition 2009 held at Florida USA IMECE2009 - 12456	Flow Investigation through a 30° Turn Diffusing Duct in Subsonic Flow Regime, Nov. 2009	P K Sinha A N Mullick B Halder B Majumdar	A N Mullick	Four	0.6
Total Credit points						12 (10)

Page - 14

(6). **Administrative Responsibilities Entrusted:**
[HOD, Dean, Chief Warden, Professor in-charge of T & P, Advisor(Estate), CVO, PI (Exam), TEQIP Coordinator]
[Since the last promotion]:

Sl. No.	Details of Responsibility Entrusted	Authority Department/Institute	Duration		Credits
			Starting Date	Ending Date	
1	Served as Professor-in-Charge Post Graduate Examination	Institute	Nov 09	Apr 14	16
2	Head Of The Department	Institute	Dec 16	Till Now	4
	Total Credit points				20(16)

(7). **Administrative Responsibilities Entrusted:**
(Warden, Assistant warden, Associate Dean, Chairman/Convener Institute Academic Committees, In-Charge Computer centre/IT services/library/Admission/student activities and other Institutional activities)
[Since the last promotion]:

Sl. No.	Details of Responsibility Entrusted	Authority Department/Institute	Duration		Credits
			Starting Date	Ending Date	
1	Served as Associate Dean, Student Welfare of the Institute	Institute	Aug 09	Apr 14	8
2	Served as Joint Centre-in-charge of CCMT 2012 & 2013	Institute	June 2012	Dec 2013	4
	Total Credit points				12(8)

(8). **Administrative Responsibilities Entrusted:**

[Chairman and convener of different standing committee and special committee (ex officio status will not be considered), faculty in-charges (Each for one year duration) of different units or equivalent]
[Since the last promotion]:

Sl. No.	Details of Responsibility Entrusted	Authority Department/Institute	Duration		Credits
			Starting Date	Ending Date	
1	Convener of Institute Scholarship Committee	Institute	Nov 09	Apr 14	3
2	Member Anti-ragging Committee	Institute	Aug 09	Apr 14	3
	Total Credit points				6(3)

Page-15

(9) **Administrative Responsibilities Entrusted:**
(Departmental activities identified by HOD like lab. In-charges or department level committee for a minimum period of one year)
[Since the last promotion]:

Sl. No.	Details of Responsibility Entrusted	Authority Department/College	Duration		Credits
			Starting Date	Ending Date	
1	Served as Lab-in-Charge Fluid Mechanics & Hydraulic Machine Lab and Aerodynamics Lab	Department	July 08	June 15	3
2	Served as Chairman Departmental Purchase Committee	Department	Nov 09	July 15	3
Total Credit points					6(3)

10. **Conducted Workshops/FDP/Short term courses of minimum 05 working days duration offered as coordinator or convener**

(Since the last promotion):

Sl.No.	Title of the Course	Duration		Co-ordinator (s)/Convenor	Credits
		Starting Date	Ending date		
1					
2					
Total Credit points					

11. **Conduct of national programmes like GIAN etc. as course coordinator:**(Since the last promotion):

Sl.No.	Title of the Course	Duration		Co-ordinator (s)	Credits
		Starting Date	Ending date		
1					
2					
Total Credit points					

12. **Conducted National/International conference as chairman or secretary :** (Since the last promotion):

Sl.No.	Title of the Conference	Duration		Chairman/Secretary	Credits
		Starting Date	Ending date		
1	NaCoMM 2009	17.12.09	18.12.09	Secretary	3
2	EARSE 2009	15.01.09	16.01.09	Secretary	3
Total Credit points					6

13. **Length of service over and above the relevant minimum teaching experience required for the given cadre:**(Since last promotion)

Sl. No	Years of service over and above the relevant minimum teaching experience required for the given cadre	Credits
1	Total years of Service 38 yrs 3 mths	10
Total Credit points		10

14. **Establishment of new Laboratories:** (Since the last promotion)

Sl.No.	Name of Lab. established	Funded by	Co-ordinator (s)	Credits
1	Established Aerodynamics Lab	TEQIP & Institute		4
2				
Total Credit points				4

15. **Details of Theory Courses taught over and above 6 credits hrs.:**(Since the last promotion)

Sl.No.	Name of Course	No. of hrs/week with credit	Credits
1	B Tech & M Tech	8	6
2			
Total Credit points			6

(16).PG Dissertation Guided: (Since the last promotion)

Sl.No.	Title of the Project	Name of supervisor(s)	Academic session	Credits
1	A Numerical Study of Flow development through a Bifurcated Duct	A N Mullick	2011	0.5
2	Rheological Behaviour of Al-Mg_2Si Composite	A N Mullick	2015	0.5
3	Hydrodynamic Viscous Friction Analysis of Piston Ring and Cylinder Liner Contact	A N Mullick & B Bera	2015	0.5
4	Effect of Drag Coefficient on the Aerodynamic performance of the Vehicle	A N Mullick & S Banerjee	2016	0.5
5	Aerodynamic Drag reduction of Bluff Body	A N Mullick	2017	0.5
6	Numerical Investigation of Flow Characteristics of Butterfly Valve at different closure angle	A N Mullick	2017	0.5
	Total Credit points			3

(17).UG Projects guided:(Since the last promotion)

Sl.No.	Title of the Project	Name of supervisors	Year of Submission	Credits
1	Characteristics of Fluid Flow Through Microchannels – a paradigm	A N Mullick	2010	0.25
2	A Computational Study on Horizontal Axis Wind Turbine	A N Mullick	2011	0.25
3	Numerical Prediction of Subsonic Flow through Air Intake Constant Area Duct in Turbulent Flow Regimes	A N Mullick	2011	0.25
4	A Study of Continuous Casting of Steel	A N Mullick	2012	0.25
5	Numerical Study of Droplet Motion in Gas Flow Channel of PEM Fuel Cell	A N Mullick	2012	0.25
6	Design and Manufacturing of a Low Cost Domestic Solar Water Heating System with Parabolic Reflector	A N Mullick	2013	0.25
7	Flow over a Flat Plate with Square Obstruction	A N Mullick	2013	0.25
8	MHD flow and Heat Transfer past a Square Cylinder in a Duct at High Hartmann & Reynolds Number	A N Mullick	2014	0.25
9	Design of Automated Floor Cleaner & Mopper	A N Mullick	2014	0.25
10	CFD Analysis of Turbulent Flow Over a Cube Placed on a Flat Plate	A N Mullick	2014	0.25
11	Design of Centrifugal Pump – a Case study	A N Mullick	2015	0.25
12	Comparison of Flow of Water and Slurry Through a Pipe – a Case Study	A N Mullick		0.25
13	Mesh Optimization of Diffuser	A N Mullick	2016	0.25
14	Wind as an Alternate Source of Energy	A N Mullick	2016	0.25
15	Characteristics of Fluid Flow Through Microchannels & Electro-osmotic process	A N Mullick	2017	0.25
16	Study of Versajet II Hydo-surgery System	A N Mullick	2017	0.25

Page - 17

Total Credit points		4

(18). **Text/ Reference book published on relevant subject from reputed international publishers:** (Since the last promotion)

Sl.No.	Title of the Text/Reference book	Publishers	Year of publication	Credits
1	Studies of Flow Through Air Intake using Artificial Intelligence	Lap Lambert Academic Publishing	2012	6
2				
Total Credit points				6

(19). **Text/ Reference book published from reputed national publishers or book chapters in the bookspublished byreputed international publishers:** (Since the last promotion)

Sl.No.	Title of the Text/Reference book	Publishers	Year of publication	Credits
1				
2				
Total Credit points				

(20). **Details of significant outreach activities:**(Since the last promotion)

Sl.No.	Activities	Year	Credits
1	Reviewer of Journal of Applied Fluid Mechanics	2015-17	1
2	Reviewer of Journal of Institute of Engineers Series C	2015 -17	1
3	External Examiner of Ph D Dissertation at Jadavpur University	2016	1
4	Acted as Advisor at Tripura Public Service Commission	2017	1
Total Credit points			4

(21). **Fellowships of IEEE, FNA, FNAE, FNASc:**

Sl.No.	Professional Society	Year	Credits
1			
2			

Total Credit points	

(22). **_lacement percentage (only for the placement cell officers/faculty in-charge of placement):**(Sincethelast promotion)

Sl.No.	Year	Percentage of placement(% to be based on total number of students passing out and single job offer)	Credits
1			
2			
Total Credit points			

(C).Self-Assessment Score of the applicant:
[Cumulative Credits Points of (1) to (22) above] 130 (114)

30/11

Place & Date Durgapur, November 30, 2017

Signature of applicant

Page-19

[Page - 20]

Annexure I

12 (a) Paper Publication in National/International Journals (SCI/SCOPUS Indexed Journals only)

S.No.	Title of paper	Author(s)	Name of the Journal	Vol. &Year	Pages
1	Numerical Prediction of Flow and Heat Transfer Characteristics of water-fly Ash Slurry in a 180° return pipe bend	B B Nayek, D Chaterjee & A N Mullick	International Journal of Thermal Science	Vol 113, 2017	100 - 115
2	Numerical Simulation of Convective Transport of Fly ash-water slurry in Horizontal pipe bend	B B Nayek, D Chaterjee & A N Mullick	Multiphase Science & Technology	Vol 27. 2015	159 - 186
3	Numerical analysis of convective transport of fly ash-water slurry through a horizontal pipe	B B Nayek, S Gupta, D Chatterjee & A N Mullick	Journal of Computational Multiphase Flows	Vol 7, 2015	79 - 96
4	A Computational Analysis of Flow Structure through a constant area S duct	A K Biswas, P K Sinha, A N Mullick & B Majumdar	International Review of Mechanical Engineering	Vol. 8 No. 2 2014	339 - 406
5	A Computational Analysis of Flow Structure through a constant area S duct	A K Biswas, P K Sinha, A N Mullick & B Majumdar	International Review of Aerospace Engineering	Vol. 6 No. 3 June 2013	145 - 152
6	CFD Investigation of Flow through a Constant Area Curved Duct	A K Biswas, P K Sinha, A N Mullick & B Majumdar	International Review of Mechanical Engineering	Vol. 6 No. 7 November 2012	1654 - 1660
7	Flow investigation through an annular diffuser in subsonic flow regime	P K Sinha, A N Mullick, B Halder & B Majumdar	Journal of Aerospace Engineers, Institute of Engineers (India)	Vol. 90, November 2009	24 - 29
8	An experimental Investigation on the Airflow in Twin Submerged Side Intake at Subsonic Speed	A N Mullick, S Chakravorty & B Majumder	International Journal of Turbo & Jet-Engines	Vol. 24, Nos. 3-4, 2007	207 - 222

Page-21

International Journal of Thermal Sciences 113 (2017) 100–115

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

Numerical prediction of flow and heat transfer characteristics of water-fly ash slurry in a 180° return pipe bend

Bibhuti Bhusan Nayak [a], Dipankar Chatterjee [b,*], Amar Nath Mullick [a]

[a] Department of Mechanical Engineering, National Institute of Technology, Durgapur, 713209, India
[b] Advanced Design and Analysis Group, CSIR-Central Mechanical Engineering Research Institute, Durgapur, 713209, India

ARTICLE INFO

Article history:
Received 22 April 2016
Received in revised form
28 November 2016
Accepted 28 November 2016

Keywords:
Liquid-solid slurry flow
Heat transfer coefficient
Numerical simulation
Eulerian model
180° curved pipes
Pressure drop

ABSTRACT

A three-dimensional numerical simulation is performed to predict the thermofluidic transport characteristics of water-fly ash slurry in an 180° return bend. U pipelines of diameter 53 mm with radius ratios of 2.98 and 5.6 are considered that may replicate a shell and tube type heat exchanger. The pressure drop and heat transfer characteristics are predicted and the effects of Dean, Nusselt and Reynolds numbers on the vortex structure formation and heat transfer are studied. The numerical simulation is carried out by deploying the granular Eulerian multiphase model following a finite volume approach. The turbulent transport is addressed using the RNG $k-\varepsilon$ turbulence model. The results revealed that the heat transfer coefficient of pipe bends of smaller radius ratio is 53.28% more than the larger radius ratio for the solid concentration of 10% and velocity of 1 m/s. Its value increases with increase in the particle concentration and velocity due to the presence of a secondary flow in the bends. The Dean number increases with decreasing the radius ratio and the average Nusselt number increases with increasing the Reynolds number. With increasing Dean Number, the Nusselt number increases with decreasing the radius of curvature for the same particle concentration. When the particle concentration increases, the average Nusselt number also increases. The average Nusselt number in the return bend appears to be higher than that in the inlet and outlet pipes due to the presence of the secondary flows.

1. Introduction

The U pipelines are frequently found in shell and tube type heat exchangers which are widely used in oil refineries, refrigeration and air-conditioning and thermal processing plants due to their compact structure and high heat transfer coefficient [1]. Unlike the straight tubes, the bent tubes offer certain interesting fluid dynamic aspects which significantly influence the overall thermo-fluidic transport. Because of the curvature effect in bends, a secondary flow originates in the bend portion as a consequence of the centrifugal force. The vortices produced due to the secondary flow causes an interchange of the slow moving fluid near the wall and the faster moving fluid at the central core. This causes a substantial increase in the heat transfer rate between the walls and the fluid flowing through the tubes. However, it is also observed that there is an excess pressure loss in the curved pipes due to the presence of the secondary flow. In case of two phase solid-liquid flow, this secondary flow also influences the distribution of the solids leading to excessive wear. Usually, the heat transfer between the wall and the fluid is convective; however, the conductive transport also becomes important owing to the no-slip condition at the inner boundary layer. When the turbulence intensity increases, the secondary flow also increases at the bend portions [2] augmenting the convective mode of heat transfer. These fundamental and technological issues need to be addressed through a systematic study on various thermo-fluidic aspects during pipeline transportation of solid materials in the form of slurry. In this work, the hydrodynamic and thermal transport characteristics are studied for the flow of water-fly ash slurry in an 180° return bend.

As such, in order to design the heat exchanger for the fly ash slurry flow, it is essential to estimate the heat transfer and pressure drop characteristics of the fly ash produced by the combustion of coal in thermal power plants. The present study aims at performing the numerical investigation of the influence of dispersed particles on the heat transfer of liquid-solid slurry flow in a horizontal U-bend pipe using coal fly ash particles of diameter 13 μm suspended in water at concentrations ranging from 10 to 50%. The pipes can

* Corresponding author.
E-mail address: d_chatterjee@cmeri.res.in (D. Chatterjee).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.11.019
1290-0729/© 2016 Published by Elsevier Masson SAS.

Page - 22

Multiphase Science and Technology, 27 (2-4): 159–186 (2015)

NUMERICAL SIMULATION OF CONVECTIVE TRANSPORT OF FLY ASH–WATER SLURRY IN HORIZONTAL PIPE BENDS

Bibhuti Bhusan Nayak,[1] Dipankar Chatterjee,[2,] & Amar Nath Mullick[1]*

[1] *Department of Mechanical Engineering, National Institute of Technology, Durgapur, Durgapur-713209, India*

[2] *Simulation & Modeling Laboratory, CSIR-Central Mechanical Engineering Research Institute, Durgapur-713209, India*

[*] Address all correspondence to: Dipankar Chatterjee,
E-mail: d_chatterjee@cmeri.res.in

A 3D numerical study is conducted to analyze the thermo-fluidic transport associated with the flow of water–fly ash slurry in 90 deg curved horizontal pipes having different radius ratios (ratio between radius of curvature and radius of pipe) of 2.98 and 5.6. The above phenomena in bends is also compared with straight pipe of the same length and diameter for the flow of spherical fly ash particle of sizes 13 and 34 μm at velocities ranging between 1 and 5 m/s and particle concentrations within 10–50% by volume for each velocity. The simulation is carried out by deploying a Eulerian multiphase model available in the commercial computational fluid dynamics code Ansys Fluent. The pipe wall is kept at an isothermal condition of 400 K, whereas the slurry enters the pipeline at a temperature of 300 K. The results indicate that the pipes having bends enhance the heat transfer performance at the expense of the increased pressure drop compared to the straight pipes and also the pressure drop and heat transfer increase with decreasing radius of curvature due to the increase of the secondary flow in the pipe bends. It is also observed that the pressure drop is always greater when the slurry contains larger size particles. On the contrary, the heat transfer coefficient of the slurry having a smaller size of particles is found more in comparison to the larger-size particle slurry.

KEY WORDS: *slurry transport, heat transfer, numerical simulation, Eulerian model, pipe bends, secondary flow*

1. INTRODUCTION

Pipelines are very common means for the long-distance hydraulical transportation of various solid materials such as the coal, fly ash, limestone, zinc tailings, rock phosphate Gilsonite, copper and iron concentrate, etc. Pipe bends are an integral part of any pipeline network system since they provide flexibility in routing. However, because of the curvature effect in bends, a secondary flow originates in the bend portion as a consequence of the centrifugal force. This secondary flow may cause a significant enhancement in

[Page-23]

Numerical analysis of convective transport of fly ash-water slurry through a horizontal pipe

B. B. Nayak, S. K. Gupta, D. Chatterjee and A. N. Mullick

Reprinted from

The Journal of
Computational Multiphase Flows

Volume 7 · Number 2 · 2015

Multi-Science Publishing

Page-24

Numerical Analysis of Convective Transport of Fly Ash-water Slurry Through a Horizontal Pipe

Bibhuti Bhusan Nayak[1], Satish Kumar Gupta[1], Dipankar Chatterjee[2*] and Amar Nath Mullick[1]

[1]Department of Mechanical Engineering
National Institute of Technology, Durgapur
Durgapur-713209, India
[2]Simulation & Modeling Laboratory
CSIR-Central Mechanical Engineering Research Institute,
Durgapur-713209, India

Received date: 2 November 2014; Accepted date: 22 April 2015

Abstract

The thermal transport of solid-liquid suspension under turbulent flow condition is not well understood because of the complex interaction between the solid particles and the turbulent carrier fluid. The solid particles may enhance or suppress the rate of heat transfer and turbulence depending on their size and concentration. In the present paper, a three-dimensional numerical simulation is carried out in order to study the pressure drop and heat transfer characteristics of a liquid-solid slurry flow in a horizontal pipe. The simulation is performed by using the algebraic slip mixture (ASM) model which is a part of the finite-volume based CFD software Ansys Fluent. The turbulence is handled by the RNG $k - \varepsilon$ model. A hexagonal shape and cooper type non-uniform three-dimensional grid is created to discretize the computational domain. Spherical fly ash particles, with mass median diameter of 13μm for an average flow velocity ranging from 1-5 m/s and particle concentrations within 0-40% by volume for each velocity, are considered as the dispersed phase. The results illustrate that higher particle concentration in the flow causes an increase in the heat transfer and pressure drop. Moreover, both heat transfer and pressure drop are seen to show a positive dependence on the mean velocity of the flow.

1. INTRODUCTION

Liquid–solid two phase flows, referred to as slurries, are common to many engineering and natural processes. Pressure drop and the rate of enhancement or suppression of heat transfer due to the presence of the solid particles in slurry flows depend on their concentration and size distribution. The knowledge of these two characteristics is essential for the design of slurry pumps, heat exchangers, driers, fluidized beds and slurry pipeline reactors which helps to maximize the economic benefit. Addition of micron-size solid particles into gases or liquids at small volumetric fractions is known to enhance heat transfer due to the thinning of the viscous sub-layer and creating a high thermal conductivity in that layer. Owing to the shortages in the energy and material in near future, development of various heat transfer enhancement techniques has grabbed the attention of many researchers.

Harada et al. [1] experimentally investigated heat transfer from wall to water suspensions of glass beads (particle diameter, d = 0.06 to 1 mm) or ion exchange resin (d = 0.8 mm) flowing through horizontal pipes (pipe diameter, D = 14mm, 19mm, 25mm) with Reynolds numbers within the range 3000 to 50000 and volume fraction of solid 0 to 0.1. They concluded that for particle sizes larger than 0.35 mm in diameter, the heat transfer coefficient was always 10-30 % larger than water

* Corresponding author: E-mail address: d_chatterjee@cmeri.res.in

International Review of Mechanical Engineering (I.RE.M.E.), Vol. 8, N. 2
March 2014
ISSN 1970 - 8734

Praise Worthy Prize

A Computational Analysis of Flow Structure Through a Constant Area DS-Duct

A. K. Biswas[1], Prasanta K. Sinha[2], A. N. Mullick[3], B. Majumdar[4]

Abstract – *In the present investigation the distribution of wall static pressure and mean velocity are experimentally studied on a DS-shape constant area rectangular curved duct of 22.5°/22,5°/22.5°/22.5° angle of turn with an aspect ratio of 2.0. The experiment is carried out at mass averaged mean velocity of 40m/s. The velocity distribution shows the Bulk of flow shifting from wall-2 to wall-1 in the first bend and third bend of the duct and wall-1 to wall-2 in the second bend and forth bend of the duct along the flow passage of the DS-duct is very instinct. Flow at end of the DS-duct is psurely uniform in nature due to non existence of secondary motion. The experimental results then were numerically validated with the help of Fluent, and the comparison of experimental and predicted results for DS duct shows good quality agreement between the experimental and predicted results.*

Keywords: *Constant Area DS-Duct, Secondary Motion, Wall Pressure, K-E Model, Fluent Solver*

Nomenclature

Dn	Dean Number
L	Centerline length
R_c	Mean Radius of curvature
R_e	Reynold's number ($U_{av}D/v$)
U_{av}	Inlet Average Velocity
W	Inlet Width
b	Height of duct
$\Delta\beta$	Angle of turn of the curvature
v	Kinematic Viscosity

I. Introduction

Constant area Double S-ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized. Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector.

As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices. Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry albeit with high efficiency.

In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

Double S-ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular. Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices. Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex geometry (like inlets, nozzles,

Manuscript received and revised February 2014, accepted March 2014

Praise Worthy Prize

International Review of Aerospace Engineering (I.RE.AS.E), Vol. 6, N. 3
ISSN 1973-7459 June 2013

A Computational Analysis of Flow Development Through a Constant Area S-Duct

A. K. Biswas[1], Prasanta K. Sinha[2], A. N. Mullick[1], B. Majumdar[3]

Abstract – *In the present investigation the distribution of wall static pressure and mean velocity are experimentally studied on a S-shape constant area rectangular curved duct of 45°/45° angle of turn with an aspect ratio of 2.0. The experiment is carried out at mass averaged mean velocity of 40m/s. The velocity distribution shows the bulk flow shifting from outer wall to the inner wall in the first half and from inner wall to the outer wall in the second half along the flow passage of sigmoid duct is very instinct. Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the form of counter rotating vortices have been generated within the curved duct. The experimental results then were numerically validated with the help of Fluent, which shows a good agreement between the experimental and predicted results.*

***Keywords**: Constant Area S-Duct, Secondary Motion, Wall Pressure, k-ε Model, Fluent Solver*

Nomenclature

Dn	Dean Number
L	Centerline length
R_c	Mean Radius of curvature
R_e	Reynold's number ($U_{av}D/v$)
U	Velocity of fluid
U_{av}	Inlet Average Velocity
W	Inlet Width
b	Height of duct
f	Body force per unit mass
$\Delta\beta$	Angle of turn of the curvature
v	Kinematic Viscosity
μ	Dynamic Viscosity
ρ	Mass density of fluid
k	Kinetic energy
ε	Dissipation rate
PIV	Particle Image Velocity-meter
LDV	Laser Doppler Velocity-meter

I. Introduction

Constant area Sigmoid ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized.

Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector.

As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices.

Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

S-ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and

Manuscript received and revised May 2013, accepted June 2013

Page - 21

Praise Worthy Prize

International Review of Mechanical Engineering (I.RE.M.E.), Vol. 6, N. 7
ISSN 1970 - 8734
November 2012

CFD Investigation of Flow through a Constant Area Curved Duct

A. K. Biswas[1], Prasanta K. Sinha[2], A. N. Mullick[3], B. Majumdar[4]

Abstract – *In the present investigation the distribution of wall static pressure and mean velocity are experimentally studied on a C-shape constant area rectangular curved duct of 90° angle of turn. The experiment is carried out at mass averaged mean velocity of 40m/s. The velocity distribution shows the bulk flow shifting from outer wall to the inner wall along the flow passage of curved duct is very instinct. Due to the imbalance of centrifugal force and radial pressure gradient, secondary motions in the form of counter rotating vortices have been generated within the curved duct. The experimental results then were numerically validated with the help of Fluent, which shows a good agreement between the experimental and predicted results.*

Keywords: *Constant Area Curved Duct, Secondary Motion, Wall pressure, k-ε Model, Fluent Solver*

Nomenclature

Dn	Dean Number
L	Centerline length
R_c	Mean Radius of curvature
R_e	Reynold's number ($U_{av}D/v$)
U_{av}	Inlet Average Velocity
W	Inlet Width
h	Height of duct
$\Delta\beta$	Angle of turn of the curvature
ν	Kinematic Viscosity

I. Introduction

Constant area curved ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized. Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector. As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices.

Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

C-ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures.

Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular.

Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices.

Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex geometry (like inlets, nozzles, diffusers, contractions elbows etc) albeit with high efficiency. In these applications, design of the ducts is based on the

Manuscript received and revised October 2012, accepted November 2012

Posc 28

Flow Investigation through an Annular Diffuser in Subsonic Flow Regime

P K Sinha, *Fellow*

Dr A N Mullick, *Member*

Dr B Halder, *Fellow*

Dr B Majumdar, *Member*

Annular diffusers are an integral component of the gas turbine engines of high-speed aircraft. These facilitate effective operation of the combustor by reducing the total pressure loss. The performance characteristics of these diffusers depend on their geometry and the inlet conditions. In the present investigation, the distribution of radial velocity, transverse velocity, mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.44 and centreline length was chosen as three times of inlet diameter. The measurements of radial velocity, transverse velocity, mean velocity, static pressure and total pressure distribution, were taken at Reynolds number of 1.9×10^5 based on inlet diameter and average inlet velocity 38 m/s. The mean velocity and Y and Z components of mean velocities were measured with the help of pre-calibrated five-hole pressure probe. Results show that the mean velocity distribution is symmetrical and uniform at the inlet section and there after high velocity cores are accumulated at the hub wall. This may be due to formation of vortices between these zones. The transverse velocity distribution shows the development of secondary motion in the form of contra-rotating vortices. At the plane between outward casing wall and top inward casing wall and hub wall certain rolls of contra-rotating vortices are observed. The mass average static pressure recovery within the curved diffuser is continuous from Section A to Section C. Performance parameter like coefficient of mass average static pressure recovery and coefficient of mass average total pressure loss are found to be 23% and 17%, respectively.

Keywords : Annular curved diffuser; Subsonic flow; Contra-rotating vortices

NOTATIONS

- A_r : area ratio
- A_s : aspect ratio
- cc : concave or inward wall
- C_{PR} : coefficient of pressure recovery
- cv : convex or outward wall
- D : inlet diameter of the diffuser
- L : centreline length of the diffuser
- P : pressure sensed by five hole probe
- Re : Reynolds number
- U : velocity of air
- α : pitch angle
- β : Yaw angle
- ρ_{air} : density of air
- ρ_m : density of manometric liquid
- ζ : coefficient of pressure loss
- $\Delta\beta$: angle of turn of the centreline

Subscripts

- av : average
- e : exit
- i : ith data point in a given section
- S : static
- T : total
- x,y,z : directions along the 3D Cartesian

INTRODUCTION

Diffusers are used in many engineering systems to decelerate the flow and to convert the dynamic pressure into static pressure. The annular curved diffuser is an essential

P K Sinha is with the Durgapur Institute of Advanced Technology and Management, Durgapur 713 212; Dr A N Mullick and Dr B Halder are with the Department of Mechanical Engineering, NIT, Durgapur 713 209; and Dr B Majumdar is with the Department of Power Engineering, Jadavpur University, Kolkata 700 098.

This paper was presented and discussed at the Twenty-second National Convention of Aerospace Engineers held at Ranchi during November 27-29, 2008.

Page-29

©Freund Publishing House Ltd. International Journal of Turbo and Jet Engines, 24, 207-223 (2007)

An Experimental Investigation on the Airflow in Twin Submerged Side Intake at Subsonic Speeds

A N Mullick [a,*], S Chakravorty [b] & B Majumdar [c]

[a] *National Institute of Technology, Durgapur, Applied Mech. Dept. India, 713209*
[b] *Jadavpur University, Electrical Engineering Department, India 700032*
[c] *Jadavpur University, Power Engineering Department, India 700098.*

tract

The intake of an air vehicle is an important component, which affects not only the propulsion but also the aerodynamic characteristics of the vehicle. Single engine fighter aircrafts embedded in the fuselage typically have Y-shaped popularly known as twin intake ducts, which symmetrical about their central plane. In the present paper an experimental study was made of an incompressible subsonic flow through a model twin intake duct of rectangular cross-sectional area of $A_s = 2.0$ and $A_r = 1.5$. The operating R_e based on the inlet hydraulic mean depth and mass averaged mean velocity is maintained at 3.17×10^5. The geometry of the model twin intake is taken as two S-shaped diffusing passages of $\Delta\beta = 30°/30°$ with N = 300 mm of each. The two limbs of the twin intake duct merged at a section beyond which the duct has single arm symmetric about the vertical plane. The present work focuses on the performance and understanding of the flow characteristics of Y-shaped twin intake. A pre-calibrated MHP using function estimation tool like Wavelet Network method is used to measure various flow parameters along the axis of the duct. The distribution of longitudinal velocity, transverse velocity, static pressure and total pressure in the form of contours are drawn for analysis. The results show that the flow is uniform at the exit of the twin intake. It has also been observed that the wall pressure recoveries along the interior walls are less compared to the exterior walls, which is mainly due to the combined effect of the centrifugal force and generation of secondary motion. The overall mean coefficient of pressure recovery of both the limbs of the twin intake has been observed as about 41%.

ywords: Twin intake duct; Multi hole probe; Merger plane; Artificial Neural Network; Wavelet Network.

Nomenclature & Symbols

a	Width of the twin intake
A_r	Area ratio
A_S	Aspect ratio
b	Depth of the twin-intake
C_l	Coefficient of total pressure loss
C_p	Coefficient of static pressure recovery
$C_{p(theo)}$	Ideal coefficient of static pressure recovery
d_h	Hydraulic mean depth [2ab/(a+b)]
ew	Exterior wall
iw	Interior wall
M_a	Mach Number
N	Centerline length
R_c	Mean radius of curvature
R_e	Reynolds Number $[Ud_h/\mu]$
st.	Measuring station
MHP	Multi hole probe
U	Velocity of fluid
u, v, w	Component of velocity along x, y and z directions
μ	Kinematic viscosity
ρ	Density
β	Curvature ratio $[\beta = R_c/d_h]$
$\Delta\beta$	Angle of turn
ξ_0	Diffuser effectiveness

* Amarnath Mullick <anmullick@gmail.com>

Annexure II

12 (b) Paper Publication in Conferences (SCI/SCOPUS/Web of Science/Internationally Renowned Conferences)

S.No.	Title of paper	Co-author(s), if any	Name of the Conference	Date
1	Numerical Prediction of Flow Behaviour of Coal Water and Copper Ore Water Slurries	R Mishra, K C Ghanta, A N Mullick & S L Sinha	20th International Conference on Aerospace & Mechanical Engineering	June 2018
2	Numerical Prediction on Two Phase Flow Through an 180° Pipe Bend	B B Nayek, D Chatterjee & A N Mullick	5th International Conferences on Advances in Civil, Structural and Mechanical Engineering, CSM 2017	September 2017
3	Numerical Investigations of Heat Transfer between wall and water-fly ash slurry flow in Horizontal pipes	B B Nayek, S Gupta, D Chatterjee & A N Mullick	41st National & 5th International Conference on FMFP 2014 to be held at IIT Kanpur	December 2014
4	CFD Investigation of Flow in a Continuous Casting Tundish	Deepak Verma & A N Mullick	International Conference on Advances in Refractories and Clean Steel Making at Ranchi	June 2013
5	A Comparison between C-shaped Constant Area Duct and Diffuser using CFD Analysis	A K Biswas, Ashoke K Raman N K Das A N Mullick P Ray B Halder & B Majumdar	International Conference on Recent Advances in Mechanical Engineering held at Chennai	April 2012
6	A Numerical Analysis of Flow Development through a Constant Area Duct	A K Biswas, Ashoke K Raman A N Mullick & B Majumdar	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012) held at Singapore	March 2012
7	Numerical Validation of Flow through S-shaped Diffuser	N K Das A K Biswas, Ashoke K Raman A N Mullick B Halder P Ray & B Majumdar	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012)	March 2012
8	Analysis of Flow Performance through Annular Curved Diffusing Duct using CFD	P K Sinha A N Mullick B Halder B Majumdar	International Conference on Fluid Dynamics and Thermodynamics Technology (FDTT 2012)	March 2012
9	A Comparative Study of Turbulence Models Performance through a Curved Diffuser in Turbulent Flow Regime	N K Das A K Biswas, Ashoke K Raman A N Mullick B Halder P Ray & B Majumdar	International Conference on Design and Advances in Mechanical Engineering held at Tiruvannamalai	December 2011
10	Wall Y^+ Approach for Dealing with Turbulent Flow Through a Constant Area Duct	Isha Shukla, S S Tupkari, Ashoke K Raman & A N Mullick	4th International Meetings of Advances in Thermofluids (IMAT 2011), Maleka, Malaysia	October 2011
11	Numerical Investigation of flow through a Curved Annular Diffuser	P K Sinha A N Mullick B Halder B Majumdar	4th International Meetings of Advances in Thermofluids (IMAT 2011), Maleka, Malaysia	October 2011

12	A Numerical Analysis of Flow Development through a Constant Area Duct	Ashoke K Raman A K Biswas & A N Mullick	5th International Conference on Advance Mechanical Engineering at SVNIT, Surat	June 2011
13	Numerical Investigation of flow through Annular Curved Diffusing Duct	P K Sinha A N Mullick B Halder B Majumdar	5th International Conference on Advance Mechanical Engineering at SVNIT, Surat	June 2011
14	Characteristics of Fluid Flow through Microchannels	Meenkshi Mour, debarun Das & A N Mullick	International Conference on Modeling, Optimization & Computing (ICMOC 2010), at NIT Durgapur, India	October 2010
15	Flow Investigation through Annular Curved Diffusing Duct	P K Sinha A N Mullick B Halder B Majumdar	International Conference on Modeling, Optimization & Computing (ICMOC 2010), at NIT Durgapur, India	October 2010
16	Flow Investigation through annular curved diffuser	P K Sinha A N Mullick B Halder B Majumdar	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	December 2010
17	Flow development in a constant area duct	A K Biswas, A N Mullick & B Majumdar	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	December 2010
18	An experimental investigation on a flow through a curved diffuser	N K Das A N Mullick B Halder P Ray & B Majumdar	37th National & 4th International Conference on FMFP 2010, at IIT Chennai, India	December 2010
19	Biofuels – a healthier and wealthier future for India	Shankarnsh Srivastava Nirupam Kar & A N Mullick	1st International Conference on New Frontiers in Biofuels, at Delhi Technological University, India	January 2010
20	Flow Investigation through a 30° Turn Diffusing Duct in Subsonic Flow Regime	P K Sinha A N Mullick B Halder B Majumdar	ASME International Mechanical Engineering Congress & Exposition 2009 held at Florida USA	November 2009
21	A Computational Approach to analyze the Secondary Motion in a Twin Intake Duct	A N Mullick & B Majumdar	of ICTACEM 2007, at IIT Kharagpur	December 2007
22	A Computational Approach to analyze the Performance of a Twin Intake Duct	A N Mullick B Majumdar & S Chakravorty	7th Asian Computational Fluid Dynamics Conference at IISc, Bangalore	November 2007
23	A study of Wall Pressure Distribution of an Axisymmetrical Y-shaped Air Intake	A N Mullick & B Majumdar	33rd National and 3rd International Conference on FMFP, IIT, Mumbai	December 2006

Page - 32

WASET

waset.org

ACCEPTANCE LETTER

WORLD ACADEMY OF SCIENCE, ENGINEERING AND TECHNOLOGY

November 22, 2017

Mr. Rahul Mishra
Shri Shankaracharya Engineering College Bhilai Chhattisgarh
India

Herewith, the international scientific committee is happy to inform you that the peer-reviewed draft paper code 17ID100305 entitled (Numerical Prediction of Flow Behavior of Coal Water and Copper Ore Water Slurries by Rahul Mishra, K. C. Ghanta, A. N. Mullick, S. L. Sinha) has been accepted for oral presentation as well as inclusion in the conference proceedings of the ICAME 2018 : 20th International Conference on Aerospace and Mechanical Engineering to be held in San Francisco, USA during June, 6-7, 2018. The high-impact conference papers will also be considered for publication in the special journal issues at http://waset.org/Publications.

Conference Registration and Writing Formatted Paper:

1. Conference registration documents should be submitted to:
http://waset.org/apply/2018/06/san-francisco/ICAME?step=2
2. Word Template File should be Downloaded at
http://waset.org/downloads/template.docx
3. Latex Style File should be Downloaded at http://waset.org/downloads/latex.zip
4. Copyright Transfer Statement Document should be Downloaded at
waset.org/publications/copyright?paperCode=17ID100305

Letter of Invitation and Visa Requirements:
If you need an invitation letter to get an entrance Visa, please fill in the online form to get a letter at http://waset.org/apply/2018/06/san-francisco/ICAME?step=1.

We look forward to your participation in the ICAME 2018 : 20th International Conference on Aerospace and Mechanical Engineering.

Sincerely,

International Scientific Committee
ICAME 2018 San Francisco, USA
http://waset.org/conference/2018/06/san-francisco/ICAME

Numerical Prediction of Flow Behavior of Coal Water and Copper Ore Water Slurries

Rahul Mishra, K. C. Ghanta, A. N. Mullick, S. L. Sinha

Abstract—Many researchers are working in the field of slurry flow regarding the rheological parameters of multi-size particulate flow. It is imperative part of pipeline industry; moreover, one can say that pipeline industry has some reliance on multi-size particulate flow. Analyzers have carried out modeling of the erosion of pipes and bends. Computational fluid dynamics (CFD) has been used extensively for particulate slurry erosion divination in pipes and bends, since early 90s with emphasis on various analytical and empirical models. The review of literature in this field has been carried out to understand the extent of work that has been reached and the scope available to work in this field. The present paper introduces a simplified 3-D algebraic slip mixture (ASM) model along with RNG k-ε turbulent model to obtain the numerical solution in coal-water and copper ore-water slurry flow. The pressure drop, viscosity, density, volume fraction, Reynolds number, and Euler number have been compared with the experimental work available in the literature. The solutions have been found to be in good agreement with the experimental data.

Keywords—Rheological parameters, Pressure drop, Volume fraction, Particle size distribution, Reynolds Number, Euler's Number.

NOMENCLATURE

Symbol	
C_{ov}	Coefficient of variance
d	Particle diameter (micron/mm)
D	Pipe diameter (m)
f_m	Friction factor for solid liquid mixture
g	Acceleration due to gravity (cm/sec^2)
K_1, K_2, K, m	Constants
L	Length of test section (m)
ΔP_m	Pressure drop due to flow of solid liquid mixture (bar)
R	Pipe radius (cm/m)
Re	Mixture Reynolds Number
s	Specific gravity
S	Modulus of the mean rate of strain tensor time (Sec.)
V_f	Mean fluid velocity
V_m	Mean suspension velocity
X_V	Volume fraction of solids
F	Body force (N/m^2)
U_m	Mass averaged velocity (cm/sec).
Greek Letters	
$\alpha, \beta, \gamma, \delta$	Constants
Θ	Deflection (degree)
μ_L	Viscosity of liquid (CP)
μ_m	Viscosity of suspension (CP)
ρ_m	Density of mixture (kg/m^3)
ρ	Density of liquid (kg/m^3)
ρ_s	Density of solid (kg/m^3)
ε	Dissipation rate of turbulent kinetic energy (cm^2/sec^3)

Rahul Mishra is with the Department of Mechanical Engineering, Shri Shankaracharya Engineering College-490020, Bhilai, C. G., India (corresponding author, phone. (0788)-2279336, +91-9826175806, 9827448803, e-mail: mishrarahul87@gmail.com).

K. C. Ghanta is with the Department of Chemical Engineering, National Institute of Technology Durgapur-713209, W. B., India.

A. N. Mullick is with the Department of Mechanical Engineering, National Institute of Technology Durgapur-713209, W. B., India.

S. L. Sinha is with the Department of Mechanical Engineering, National Institute of Technology Raipur-492010, C.G., India

I. INTRODUCTION

MANY gargantuan pipelines were built around the world for the flow of slurry. As compared to rail and road transport, pipeline transport had been considered more economical and eco-friendly. To delineate the pipelines and its associated facilities (pumps, etc.), designers need meticulous information regarding pressure drop, erosion zone, etc. at the early design phase. Also, the delineating engineers need to know meticulously the critical velocity so that the multi-size particulate slurry flow inside the pipeline can be attuned in order to have minimum pressure drop. The research had been performed in order to predict what has been done for flow of slurry through pipelines so that the pressure drop can be reduced, and how the erosion gets reduced. The present work can be considered as a step forward for better understanding of pressure drop and erosion prediction in slurry pipelines. Attempts have been made to know what has been done in the said field and what all are the findings so far; so that researcher could get the way in which advancement can be carried out. The rheological study of coal water and copper ore water slurry was done and it has been observed that the behavior of copper ore water slurry was more complex as compared to coal water slurry because of the coarse shape of copper ores. Fluid velocity and fluid viscosity are the main factors causing resistance to fluid flow. As the flow velocity or fluid viscosity increases, the pressure drop across the section also increases and vice versa. The pipeline flow may consist of wide particle size distribution which seriatim creates lots of complexities because large particle reduces the viscosity if it is having uniform shape, but if the shape is not uniform, the viscosity increases, which further increases the pressure drop; also, if volume fraction increases, the eddies and friction will increase, which then increases the pressure drop and due to friction, the erosion is also increased. These two phenomena go abreast that is why the present area of research is taken though lots of research had been going on this field but still there is a lot of scope for improvement. The present study

Paper ID CSM-17-649

Numerical Prediction on Two Phase Flow through an 180° pipe bend

B B Nayak, D Chatterjee and A N Mullick

***Abstract*— A three dimensional numerical study of flow characterized of fly ash-water slurry in a horizontal 180° pipe bend having radius ratio of 5.6 is described in this paper. The flow constitutes the fly ash particles of size 13 μm in water at velocities ranging between 1.0-5.0 m/s and particle concentrations ranging between 10-50% by volume for each velocity of flow. The numerical simulation is carried out by deploying the Eulerian Multiphase Model of CFD code in ANSYS Fluent code. The results illustrate that the overall pressure drop is 62% in the pipe bend at all particle velocities and concentrations. The velocity and concentration distributions at various positions (0°, 90° and 180°) of the bend were illustrated.**

Keywords—*U-bend, two phase flow, numerical simulation. Eulerian Multiphase approach*

I. Introduction

The 180°pipe bends are integral part of the piping system used for transportation and in heat exchangers in almost all Industrial applications. For all high Reynolds Number flow a secondary flow originates in the bend portion due to the imbalance in centrifugal force in the curvature of bend and which in turn leads to the excess pressure loss across the curved portion of the pipes. In the present work, pressure drop is predicted in a horizontal 180° pipe bends for the flow of water-fly ash slurry. Azzola *et al.* [1] used the Laser Doppler Velocimetry technique to measure the longitudinal and circumferential velocity components for developing turbulent flow in 180° pipe bend. They observed the reversal of secondary flow in circumferential velocity profile and it is independent of Reynolds Number. Jayanti et. al. [2] studied the numerical simulation of gas particle motion in 90° and 180° circular cross section pipe bends and concluded that the secondary flow induced in the gas phase due to the curvature effect. Clarke and Finn [3] did the numerical simulation of laminar flow of potassium formate through heat exchanger U-bends and observed an enhancement of heat transfer at downstream of the bend. Al-Yaari and Abu-Sharkh [4] numerically simulated oil-water two-phase flow in 180° bends using Eulerian-Eulerian approach and observed as the bend to pipe radius ratio increases, the tendency of separation of oil water system decreases. Muzumder [5] performed CFD simulation of dilute gas-solid flow through a U-Bend to study the dynamic behavior of the entrained solid particles in the flow and observed that the liquid and the gas flow rates, gravity and centrifugal forces have a strong effect on the flow behavior. Pietrzak and Witczak [6] performed experimental research on multiphase flow in 180° U-bends using three flowing media, namely, air, water and oil obtained the flow pattern and pressure drop correlations for horizontal, upward and downward flows. . Daneshfaraz [7] study numerically the velocity profile and pressure distribution on 3-D bends with different diversion angles of 90°, 135°, 180° and Reynolds number range of 100 to 1900 using CFD. He concluded that by increasing the section angle, the velocity profile and the pressure distribution incline to the outer wall and maximum deviation from inlet velocity profile occurs at 45° section angle. The maximum velocity occurs at 0.7 to 0.9 of the pipe diameter from linear wall and the maximum pressure loss occurs at angle between 22.5° and 45° with increasing Reynolds Number. Cvetkovski *et al.* [8] carried out numerical simulation of flow in U-bend pipes used in ground source and surface heat pumps for heating and cooling purposes and observed that at low turbulence, the Dean Number has a significant effect on the heat transfer in the curved section. They also concluded that the heat transfer at the curved section has of the pipe has more significant effect on Dean Number than the Reynolds Number

So far, the study of flow behavior of two-phase fly ash-water slurry flow in 180° bends at high concentrations is limited. The present study is a numerical prediction of flow characteristics of the two-phase flow in an 180° horizontal pipe bend using fly ash of particles diameter 13 μm suspended in water at high velocities and concentrations factor ranging from 10 to 50%.

II. Physical problem

The horizontal 180° bend pipe of length about L = 11m with an inner diameter of D = 0.053 m having radius ratios (R/r) of 5.6 is chosen as the slurry transportation device as shown in Figure 1.

A. *Governing Equation*

Continuity equations for the solid and liquid phases

$$\nabla \cdot (\alpha_s \rho_s \overrightarrow{v_s}) = 0 \qquad \nabla \cdot (\alpha_f \rho_f \overrightarrow{v_f}) = 0$$

Momentum equations for the solid and liquid phases

$$\nabla.(\alpha_s \rho_s \vec{v}_s \vec{v}_s) = -\alpha_s \nabla P - \nabla P_s + \nabla \bar{\bar{\tau}}_s + \alpha_s \rho_f \vec{g} + K_{fs}(\vec{v}_f - \vec{v}_s) + C_{vm} \alpha_s \rho_f (\vec{v}_f.\nabla \vec{v}_f - \vec{v}_s.\nabla \vec{v}_s) + C_L \alpha_s \rho_f (\vec{v}_s - \vec{v}_f) x (\nabla x \vec{v}_f)$$

B B Nayak and A N Mullick
National Institute of Technology Durgapur
India
amaranth.mullick@me.nitdgp.ac.in
D Chatterjee
CMERI, India

Page - 35

Fifth International and forty first national conference on Fluid Mechanics and Fluid Power, FMFP 2014, IIT Kanpur, India, 12 – 14 December, 2014

Numerical investigations of heat transfer between wall and water-fly ash slurry flow in horizontal pipes

Bibhuti Bhusan Nayak[1], Amar Nath Mullick[1, a], Satish Kumar Gupta[1], Dipankar Chatterjee[2]
[1]Department of Mechanical Engineering, National Institute of Technology, Durgapur-713209, India
[2]Simulation & Modeling Laboratory, CSIR-Central Mechanical Engineering Research Institute, Durgapur-713209, India
[a]Corresponding author, e-mail: anmullick@gmail.com,

ABSTRACT

In this paper, heat transfer and pressure drop phenomena are studied for a two phase flow of water- fly ash slurry in a horizontal pipe using the computational fluid dynamics technique (CFD). The presence of solid particles in the slurry may enhance or suppress the rate of heat transfer and turbulence depending on their size and concentration. The simulation is carried out by deploying the Eulerian multi phase modeling capability of the finite volume based commercial CFD solver ANSYS FLUENT. The turbulence is handled by the RNG k-ε model. Spherical fly ash particles, with mass median diameters of 13 μm for an average flow velocity ranging from 1-5 m/s and particle concentrations within 0-40% by volume for each velocity, are considered as the dispersed phase. The wall is kept at an isothermal condition of 400 K whereas the slurry enters the pipeline at a temperature of 300 K. The results illustrate that higher particle concentration in the flow causes an increase in the heat transfer and pressure drop. Moreover, both heat transfer and pressure drop are seen to show a positive dependence on the mean velocity of the flow.

Keywords: Liquid-solid slurry flow, Heat transfer, Numerical simulation, Eulerian model

1. Introduction and Literature Review

Heat transfer involving in the design of many industrial engineering equipment such as heat exchangers, fluidized beds, driers and slurry pipeline reactors can be accomplished through fluid like water. Various techniques have been employed to improve heat transfer rate of the fluid by which one can reduce the size of the equipment and thereby the efficiency. One of the heat transfer enhancement techniques is the addition of micro-sized solid particles to the heat transfer fluid. The solid particles migrate and disturb the boundary layer of the fluid creating high thermal conductivity in that layer resulting in the heat transfer enhancement. The enhancement of the heat transfer between the pipe wall and the slurry flow in horizontal pipes depends on the volume fraction distribution and size of the solid particles throughout the fluid. In this paper the heat transfer of the slurry which is the mixture of liquid and solid is studied computationally and presented in terms of the flow pattern and temperature profile in the fluid.

Harada et al. [1] experimentally investigated heat transfer from wall to water suspensions of glass beads (particle diameter, $d = 0.06$ to 1 mm) or ion exchange resin ($d = 0.8$ mm) flowing through horizontal pipes (pipe diameter, D = 14mm, 19mm, 25mm) with Reynolds numbers in the range 3000 to 50000 and volume fraction of solid 0 to 0.1. They concluded that for particle sizes larger than 0.35 mm in diameter, the heat transfer coefficient was always 10-30 % larger than water flow without particulates. Rozenblit et al. [2] experimentally studied the heat transfer associated with solid-water mixture (plastic acetal particles of diameter 3 mm suspended in clear water) in a horizontal pipe of length 10 m at different solid concentrations of 6%, 9%, 12% and 15% for the Reynolds number range of 19000 to 55000 using an electro-resistance sensor and infrared imaging. They observed that heat transfer coefficient increases with particle volume concentration and depends on the distribution of the solid phase along the pipe cross-section. Ku et al. [3] experimentally investigated the heat transfer coefficients of liquid-solid mixtures using a double pipe heat exchanger with suspension flows in the inner pipe. Experiments were carried out using spherical fly ash particles of diameter in the range of 4 to 78 μm, volume concentration of solids from 0 to 50% and Reynolds number in the range of 4000 to 11000. Heat transfer was found to increase with decreasing particle diameter and increasing volume concentration. Ling et al [4] numerically investigated the sand-water slurry flows in a horizontally straight pipeline of length of 1.4 m, inner diameter 0.0221 m, solid volume fraction range of 10 to 20%, mean velocity range of 1 to 3 m/s, mean particle diameter of 1.1×10^{-4} m using two different sands of silica and zirconia of densities 2380 and 4223 kg/m^3 respectively in a fully developed turbulent flow region. Verma et al.

Page - 36

(i)

CFD Investigation of Fluid Flow in a Continuous Casting Tundish

Deepak Kumar Verma[1], Satish Kumar Gupta[2]&A N Mullick[2]
[1]Alloy Steel Plant, Durgapur- 713 208
[2]National Institute of Technology, Durgapur 713 209
Email: amaranth.mullick@me.nitdgp.ac.in

ABSTRACT

The flow of molten metal in the tundish is highly influenced by the temperature distribution and separation of non-metallic inclusions, which ultimately leads to the improvisation of steel cleanliness and smooth casting operation. To study the fluid flow phenomena in continuous casting machines through trial processes is very difficult due to high temperature and complicated system. Moreover the measurements of the flow properties are also difficult and expensive. CFD (Computational Fluid Dynamics) is one of the tools, which may predict results of flow patterns, heat transfer and temperature distribution of molten metal after proper simulation and validation. In Continuous casting process tundish acts as a constant supply reservoir of molten metal. Design of the tundish usually involves combination of dam and weir as flow modifier. In the present work, three-dimensional CFD analysis of a slab tundish is carried out to study the effect of dam and combination of dam and weir by using commercial CFD software FLUENT, by using unsteady, segregated k-ε turbulent model with turbulent intensity 4% and turbulent length scale as 0.0005 m. The simulation is carried out with second order implicit upwind method at 1823K and velocity of the molten metal is considered as 1 m/s. The shroud bottom is considered as the inlet of the model. The tundish geometry is modelled exactly same as plant operating tundish in Alloy Steels Plant, Durgapur. The results are compared with the observations obtained by Singh *et. al* [6] keeping the parameters same as they adopted in their studies. It has been observed that both the results have a good agreement in qualitatively as well as quantitatively.

KEYWORDS: Computational Fluid Dynamics, Continuous Casting Tundish, Fluid Flow, Heat Transfer.

INTRODUCTION

The concept of continuous casting is of current interest because of its potential to obtain high yield, low cost, quality steel with compact design. Flow and temperature distributions of molten metal in a single strand slab tundish with dam, combination of dam and weir and bare tundish have been predicted from basic principles by solving the three-dimensional conservation equations of mass, momentum and energy. In order to obtain clean steel in the mould and to prevent premature stoppage of casting due to clogging and breakout, it is important to study the distribution of molten metal flow and heat transfer through the tundish. Using of flow modifier in the tundish is important for prevention of flow short circuiting and thus increases residence time of the molten metal before entering to the mould. Also it enhances mixing of the molten metal which is required for uniform temperature distribution and inclusion flotation due to buoyancy effect [12]. The flow induced by flow control devices enhances cross-stream mixing of the molten metal in the tundish which reduces the dead zone

Page - 37

(7)

A COMPARISON BETWEEN C-SHAPED CONSTANT AREA DUCT AND DIFFUSER BY CFD ANALYSIS

A K Biswas[1], Ashoke K Raman[1], N K Das[2], A N Mullick[1+], P Ray[1], B Halder[1] And B Majumdar[3]
[1] National Institute Of Technology Durgapur,[2] BCET Durgapur,[3] Jadavpur University, Kolkata
[+] anmullick@gmail.com

ABSTRACT

In the present paper an extensive study of rectangular cross-sectioned C-duct and C-diffuser is made by the help of 2-D mean velocity contours. Study of flow characteristics through constant area duct is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. C-ducts are used in aircraft intakes, combustors, internal cooling systems of gas turbines, ventilation ducts, wind tunnels etc., while diffuser is mechanical device usually made in the form of a gradual conical expander intended to raise the static pressure of the fluid flowing through it. Flow through curved ducts is more complex compared to straight duct due to the curvature of the duct axis and centrifugal forces are induced on the flowing fluid resulting in the development of secondary motion (normal to the primary flow direction) which is manifested in the form of a pair of contra-rotating vortices. For a diffuser in addition to the secondary flow, the diverging flow passage, which causes an adverse stream wise pressure gradient, can lead to flow separation. The combined effect may result n non uniformity of total pressure and total pressure loss at the exit.

A comparative study of different turbulent models available in the Fluent using y+ as guidance in selecting the appropriate grid configuration and turbulence models are done. Standard k-ε model and RSM models are used to solve the closure problem for both the constant area duct and the diffuser. It has been observed that the Standard k-ε model predicts the flow through the constant area duct and the diffuser within a reasonable domain of the y+ range.

1 INTRODUCTION

Diffuser is a mechanical device usually made in the form of a gradual conical expander intended to raise the static pressure of the fluid flowing through it. Specific use of a diffuser with a proper area ratio should also ensure an adequate amount of uniform flow at the outlet together with a considerable static pressure. Diffusers play an important role in many engineering and industrial applications to accomplish the objectives under geometric constraints. In aircraft gas turbines, high velocity air from the wing or fuselage first flows through a diffusing intake where it is decelerated for increasing the pressure, and then fed to the engine compressor. In the design of ventilation and air conditioning systems, diffusers are commonly used for discharging the conditioned air into the space to be cooled in order to reduce the air velocity and increase the static pressure. In these applications, on one hand there is restriction on space as well as design compatibility to match with the shortest possible duct length, while on the other hand, given cross-sectional shapes at inlet and outlet of the duct has to be satisfied; this compelled to the use of curved diffusers. C-ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular. Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices. Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex geometry (like inlets, nozzles, diffusers, contractions, elbows etc) albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

Page - 38

2012 International Conference on Fluid Dynamics and Thermodynamics Technologies (FDTT 2012)
IPCSIT vol.33(2012)©(2012) IACSIT Press, Singapore

A Numerical Analysis of Flow Development Through a Constant Area S-Duct

A. K. Biswas[1], Ashoke K. Raman[1] A. N. Mullick[1+] and B. Majumdar[2]

[1]National Institute of Technology, Durgapur 713 209, West Bengal, India
[2]Jadavpur University, Kolkata, West Bengal, India

Abstract. This paper presents the results of an experimental work with measurement of mean velocity contours in 2-D form and validation of the same with numerical results based on the y^+ approach at fully developed flow for various turbulent models like, k-ε model, k-ω model, RNG k-ε model and Reynolds Stress Model (RSM), are used to solve the problem. All the turbulence models are studied in the commercial CFD code of Fluent. The experiment is carried out at mass averaged mean velocity of 40m/s and the geometry of the duct is chosen as rectangular cross-section of 45°/45° curved constant area S-duct. In the present paper the computational results obtained from the different turbulence models are compared with the experimental results. In addition to this for validation of the numerical simulation near wall treatments for fully developed flow or log-law region are also investigated for wall $30<y^+<300$ in the region where turbulent shear dominates. It is concluded from the present study that the mesh resolving the fully turbulent region is sufficiently accurate in terms of qualitative features. Here RSM turbulence model predicts the best results while comparing with the experimental results. RSM model also predicts the flow properties more consistently because it accounts for grid independence test.

Keywords: Constant area S-duct, turbulence Models, wall function, Fluent Solver

1. Introduction

Constant area curved ducts are used for many engineering applications like small aircraft intakes, combustors, internal cooling system of gas turbines, HVAC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. In order to improve the performance of a duct it is absolutely necessary to design the duct with proper geometry so that the losses due to friction and eddies are minimized. Depending upon its application, the shape of the duct is chosen either straight or curved or annular or polar or sector. As a matter of fact the flow through a curved duct is more complex compared to straight duct due to the curvature of the centerline. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion, which is manifested in the form of a pair of contra-rotating vortices. Depending on the objective, hydro-mechanical systems often demand for the design of ducts with complex geometry albeit with high efficiency. In these applications, design of the ducts is based on the mathematical formulation of the flow field for the prescribed condition.

Rowe [1] carried out experiments on circular 90° and 180° turn curved ducts with $R_e=0.4\times10^5$ and reported the generation of contra-rotating vortices within the bends. Bansod & Bradshaw [2] studied the flow characteristics within the 22.5°/22.5° S-shaped constant area ducts of different lengths and radii of curvature.

+ Corresponding author. Tel.: + 919830385342

E-mail address: anmullick@gmail.com or amaranth.mullick@me.nitdgp.ac.in

Page - 30

2012 International Conference on Fluid Dynamics and Thermodynamics Technologies (FDTT 2012)
IPCSIT vol.33(2012)©(2012) IACSIT Press, Singapore

Numerical Validation of Flow Through an S-shaped Diffuser

Nirmal K Das[1], Ashoke K Raman[2], Amar N Mullick[2+], B. Halder[2], P. Ray[3] and B. Majumdar[4]

[1] Department of Mechanical Engineering, BCET, Durgapur, India
[2] Department of Mechanical Engineering, NIT Durgapur, India
[3] Department of Civil Engineering, NIT Durgapur, India
[4] Department of Power Engineering, Jadavpur University, India

Abstract. Curved diffusers are mainly used in both open cycle and closed cycle gas turbines. It also uses as air intake of single engine aircraft. The present paper presents the result of an experimental work of mean velocity contours in 2-D form and also the validation of the experimental result with numerical simulations of different turbulence models based on the wall y+ approach. Different turbulence models available in Fluent code are used to solve the closure problem. Wall y+ values are considered as a guiding parameter to select the appropriate grid configuration and turbulence model. The geometry of the diffuser is chosen as rectangular cross section with an area ratio of 2.0. The experiment is carried out at Re=1.56x105 based on the mass averaged velocity and hydraulic mean depth of the inlet section. In numerical analysis wall y+ value in the laminar sub-layer near the solid wall and at the fully developed layer are evaluated and checked for the values. Based on these criteria it may be concluded that the mesh resolved in the fully turbulent region are sufficiently accurate in terms of the qualitative features of the mesh independency test. On examining the contours at the exit section of all the turbulence models it has been observed that the RSM model predicts the best result as it has got a good qualitative matching with the experimental result.

1. Introduction

Diffuser is a mechanical device usually made in the form of a gradual conical expander intended to convert the dynamic pressure into static pressure of the fluid flowing through it. Diffusers are made in different shapes namely axial, radial, and curved to conform to the constraints imposed by the aspects of design. Subsonic curved diffusers, as air intakes, find wide applications in the field of aircraft design especially in military aircrafts in which the engine is frequently carried in the fuselage and the intake is located in an offset poison. The performance of such diffusers, not only in terms of the total pressure delivered but also, and more significantly, in terms of the quality (uniformity, velocity and direction) of flow at the engine face affects the response of the engine. Flow development in S-shaped diffusers is complicated and is influenced by different geometrical parameters like area ratio (AR), aspect ratio (AS), total divergence angle (2θ), angle of turn of the centre line ($\Delta\beta$), inflexion in the curvature, and dynamical parameters like inlet Mach number (Ma), inlet turbulence and specifically the angle of attack when mounted on a combat air craft.

In a curved diffuser, due to the presence of centerline curvature, fluid near the flow axis is acted upon by a larger centrifugal force than the fluid near the walls. This centrifugal pressure difference (transverse pressure gradient) forces the faster moving fluid to move outwards pushing the fluid in the boundary layer at the outer wall around the sides towards the inner wall; thus a significant secondary flow (normal to the primary flow direction) is produced.

+ Corresponding author. Tel.: + 919830385342
E-mail address: anmullick@gmail.com or amaranth.mullick@me.nitdgp.ac.in

Page -40

2012 International Conference on Fluid Dynamics and Thermodynamics Technologies (FDTT 2012)

IPCSIT vol.33(2012)©(2012) IACSIT Press, Singapore

Analysis of Flow Performances Through Annular Curved Diffusing Duct Using CFD

P. K. Sinha[1], A.N. Mullick[2+], B. Halder[2] and B. Majumdar[3]

[1]Department of Mechanical Engineering, KATM, Panagharr-713148, India

[2]Department of Mechanical Engineering, NIT, Durgapur – 713 209, India

[3]Department of Power Engineering, Jabalpur University, Kolkata, India

Abstract. Curved annular diffusers are an integral component of the gas turbine engines of high-speed aircraft. These facilitate effective operation of the combustor by reducing the total pressure loss. The performances of these diffusers depend on their geometry and the inlet conditions. In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.356. The experimental results then numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with different area ratios varying from 1.2 to 3.50.

The measurements were taken at Reynolds number 2.45 x 105 based on inlet diameter and mass average inlet velocity. Predicted results of coefficient of mass averaged static pressure recovery and coefficient of mass averaged total pressure loss are in good agreement with the experimental results. Standard k-ε model in Fluent solver was chosen for validation. From the parametric investigation it is observed that for the increase in area ratio from1.15 to 2.15, static pressure recovery increases sharply and after this the pressure recovery increases slowly up to 2.8 and beyond the area ratio 2.8 the pressure recovery decreases steadily.

Keywords: Annular curved diffuser, k-ε model, Fluent solver, Five-hole probe.

1. Introduction

Diffusers are used in many engineering application to decelerate the flow or to convert the dynamic pressure into static pressure. Depending on application, they have been designed in many different shapes and sizes. The annular curved diffuser is one of such design and is an essential component in many fluid handling systems. Annular diffusers are an integral component of the gas turbine engines of high-speed aircraft. It facilitates effective operation of the combustor by reducing the total pressure loss. The performance characteristics of these diffusers depend on their geometry and the inlet conditions. Part turn or curved diffusers are used in wind tunnels, compressor crossover, air conditioning and ventilation ducting systems, plumes, draft tubes, etc. The objective of the present study is to investigate the flow characteristics within a circular cross sectioned annular curved diffuser. The performance of an annular curved diffuser is characterizes by static pressure recovery and total pressure loss coefficient.

+ Corresponding author. Tel.: + 919830385342

E-mail address: anmullick@gmail.com or amaranth.mullick@me.nitdgp.ac.in

Page-41

Proceedings of the International Conference on
Design and Advances in Mechanical Engineering (ICDAAME 2011),
SKP Engineering College, Tiruvannamalai, TN., India. 16 - 17 December, 2011. pp.413-418.

A Comparative Study of Turbulence Models Performance through a Curved Diffuser in Turbulent Flow Regime

Nirmal K. Das[1], Ashoke K. Raman[2], A.N. Mullick[2], B. Halder[2], P. Ray[3] and B. Majumdar[4]

[1] *Department of Mechanical Engineering, BCET, Durgapur, India,*
[2] *Department of Mechanical Engineering, NIT, Durgapur, India.*
[3] *Department of Civil Engineering, NIT, Durgapur, India,*
[4] *Department of Power Engineering, Jadavpur University, Kolkata, India.*
Email: anmullick@gmail.com

ABSTRACT

The present paper represents the results of an experimental work with measurement of mass averaged mean velocity contours in 2-D form and validation of the numerical results with the experimental one by using a commercial CFD code Fluent. Mean velocities were measured at the different sections of a C-shaped diffuser of rectangular cross-section with inlet aspect ratio and area ratio of 2.0. The measured data is then processed through a computer programme to obtain the values of mean velocities and its components. The experiment was carried out at 40m/s air velocity and the experiment was carried out in an open jet blow down type wind tunnel. The test diffuser was fabricated from transparent Perspex sheet of 5 mm thick. The cross-sectional area was increased by linearly varying the width from 0.05m at inlet to 0.1m at the exit, over the total centre line length of 0.6m, while the height was kept constant at 0.1m. The width was equally distributed normal to the centerline. For mean velocity measurements, the diffusers together with the inlet and outlet pieces were divided in six sections, namely, Inlet Section, Section - A, Section - B, Section - C, Section - D, and Outlet Sections. The experimental result depicts the probable generation of secondary motion, which is an efficacy of the centrifugal effect due to centerline curvature and the subsequently developed favorable transverse pressure gradient and shifting of the bulk flow towards the outer wall was observed.

The experimental results were then considered as benchmark for the validation of numerical results for different turbulence models. A comparative study of different turbulent models available in the Fluent using y+ as guidance in selecting the appropriate grid configuration and turbulence models are done. Standard k-ε model, RNG k-ε model and RSM models are used to solve the closure problem. It has been observed that considering the near wall treatment and mesh refinement and adaptation process the mesh resolving the fully turbulent region is sufficiently accurate in terms of qualitative features and out of the three models mentioned earlier, RSM model predicts the best results while comparing with the experimental results.

Keywords— Curved Diffuser, Computational Fluid Dynamics, k-ε Turbulence Model, RNG k-ε Turbulence Model

1. INTRODUCTION

Diffuser is a mechanical device usually made in the form of a gradual conical expander intended to convert the dynamic pressure into static pressure of the fluid flowing through it. Specific use of a diffuser with a proper area ratio should also ensure an adequate amount of uniform flow at the outlet together with a considerable static pressure. Diffusers play an important role in many engineering and industrial applications to accomplish the objectives under geometric constraints. In aircraft gas turbines, high velocity air from the wing or fuselage first flows through a diffusing intake where it is decelerated for increasing the pressure, and then fed to the engine compressor. In the design of ventilation and air conditioning systems, diffusers are commonly used for discharging the conditioned air into the space in order to reduce the air velocity and increase the static pressure. Depending upon applications, part bend or 90° curved (C-shaped) diffusers can sometimes be found in some closed-circuit wind tunnel, ducting systems, draft tubes, etc. whereas S-shaped diffusers find applications as intake ducts in combat aircraft engine intakes and as interconnecting ducts between components of gas turbine engines when a shift of axis between the intake and the engine exists. Flow development in the curved diffusers is a complicate process influenced by different geometrical parameters like 2θ, Δβ, AR, AS, centerline shape etc. as well as the dynamical parameters like inlet Mach number, inlet turbulence etc. In a curved diffuser, due to the presence of centerline

Page - 42

The 4th International Meeting of Advances in Thermofluids
Melaka, Malaysia - 3rd & 4th October, 2011

Wall Y+ Approach for dealing with Turbulent Flow Through a Constant Area Duct

Isha Shukla, Swapnil S. Tupkari, Ashok K. Raman, and A. N. Mullick*

Department of Mechanical Engineering, National Institute of Technology, Mahatma Gandhi Avenue, Durgapur 713 209, West Bengal, India
**Corresponding author: amarnath.mullick@me.nitdgp.ac.in*

Abstract. The study of flow development in curved ducts has been carried out since the last century. It is of fundamental interest because of its numerous applications in fluid engineering, such as flow through pipeline, in heat exchangers, ventilators, gas turbines, aircraft intakes, gas turbines and centrifugal pumps. The flow development through this type of curved ducts depends on its geometrical and dynamical parameters. In the present paper an approach has been made for dealing with turbulent flows within a curved duct with a rectangular cross-section and the result obtained from the experimental work has been compared and validated through numerical simulation by using Fluent CFD codes. The experiment is carried out at mass average velocity based on the inlet cross section as 40m/s. In the present study using the wall y^+ as guidance in selecting the appropriate grid configuration and corresponding turbulence models are investigated. The standard k-ε, standard k-ω, Reynolds Stress Model (RSM) and Spalart-Almaras (SA) turbulence models are used to solve the closure problem. Their behaviours together with the accompanying near-wall treatments are investigated for wall y^+ value less than 5 covering the viscous sub layer and y^+ value ranging 5 to 30 in the buffer region. Notably, adopting a wall y^+ in the log-law region, where y^+ value is greater than 30, has also been taken care during the study. After various trials the optimum results were obtained for the K-ω model with a mesh count of 0.54 millions. In this, the value of y^+ was almost within the required range, i.e. $5<y^+<30$. Thus, it was the best mesh that is obtained by wall y+ approach and it has been observed that the K-ω model is also best turbulent model among the four turbulent models mentioned earlier, for predicting the flow through this type of geometry.In the second part of the study an attempt has been made to find out the best model by comparing the computational and experimental results. The mean velocity, static pressure, total pressure and velocity vector distributions at different sections for the 3 models K-ε, K-ω, SA and RSM have been presented in the form of contours and in vector plots.

Keywords: Computational Fluid Dynamics, Spalart-Almaras Turbulence Models, K-ε Turbulence Model, k-ω Turbulence Model, Reynolds Stress Model.
PACS: 47.11.-j

NOMENCLATURE

H	Hydraulic Mean Depth	Re_H	Reynolds Number ($=Hu_B/\upsilon_{air}$)
k	Turbulent kinetic energy ($m^2 s^{-2}$)	u	Instantaneous Velocity (ms^{-1})
x	Horizontal Distance along stream wise direction (m)	y	Vertical Distance normal to wall direction (m)
z	Distance parallel to the span wise direction (m)	ω	Specific dissipation rate
ε	Dissipation rate	y+	Dimensionless distance to the wall
ν_{air}	Kinematic viscosity of air ($m^2 s^{-1}$)		

INTRODUCTION

Constant area ducts have many applications in engineering. These are used in small aircrafts, combustors, internal cooling of gas turbines, HAVC ducting system, wind tunnels, heat exchangers in food processing refrigeration and hydrocarbon industries etc. Performance of a curved constant area duct depends on the geometrical

page-43

The 4th International Meeting of Advances in Thermofluids
Melaka, Malaysia - 3rd & 4th October, 2011

Numerical Investigation of flow through a Curved Annular Diffuser

P. K. Sinha[a], A.N. Mullick[b*], B. Halder[c] and B. Majumdar[d]

[a]*Department of Mechanical Engineering, DIATM, Durgapur-713212, India*
[b, c] *Department of Mechanical Engineering, NIT, Durgapur – 713 209, India*
[d]*Department of Power Engineering, Jadavpur University, Kolkata, India*
[*]*Corresponding author: e-mail: anmullick@gmail.com*

Abstract. In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.28. The centerline length is taken as three times of inlet diameter. The experimental results are then numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with varying area ratios ranging from 1.15 to 3.75. All the measurements are taken at Reynolds number 2.15 x 10^5 based on the inlet diameter and mass average inlet velocity. Predicted results of mass averaged coefficient of static pressure recovery (32%) and total pressure loss (21%) are in good agreement with the experimental results of mass averaged coefficient of static pressure recovery (28%) and total pressure loss (17%) respectively. Standard k-ε model in Fluent solver is chosen for validation. From the parametric investigation it is observed that static pressure recovery increases up to an area ratio of 2.85 and beyond the area ratios 2.85 and up to 3.75, pressure recovery decreases steadily. The coefficient of total pressure loss almost remains constant with the change in area ratio for similar inlet conditions.

Keywords: Annular curved diffuser, k-ε model, Fluent solver, Five-hole probe.
PACS: 47.11.-j

NOMENCLATURE

A_r	Area ratio	D	Inlet diameter of the Diffuser
A_s	Aspect ratio	L	Centerline length of the Diffuser
CC	Concave or outward wall	Re	Reynolds number
C_{PR}	Coefficient of pressure recovery	$\Delta\beta$	Angle of turn of the center line
CV	Convex or inward wall		

INTRODUCTION

Diffusers are used in many engineering application to decelerate the flow or to convert the dynamic pressure into static pressure. Depending on application, they have been designed in many different shapes and sizes. The annular curved diffuser is one of such design and is an essential component in many fluid-handling systems. Annular diffusers are an integral component of the open cycle gas turbine engines of high-speed aircraft. It facilitates effective operation of the combustor by increasing the static pressure at uniform flow distribution at the exit. The performance characteristics of these diffusers depend on their geometry and the inlet conditions. Part turn or curved diffusers are used in wind tunnels, compressor crossover, air conditioning and ventilation ducting systems, flumes, draft tubes, etc.

page -91

Proc. of the 5th International Conference on Advances in Mechanical Engineering (ICAME-2011), June 06-08, 2011
S V. National Institute of Technology, Surat – 395 007, Gujarat, India

A Numerical Analysis of Flow Development Through a Constant Area Duct

Asoke Raman, A. K. Biswas A. N. Mullick*

National Institute of Technology, Durgapur 713 209, West Bengal, India
*Corresponding Author (e-mail: amarnath.mullick@me.nitdgp.ac.in)

This paper represents the results of an experimental work with measurement of mean velocity along with total pressure contours in 2-D form and validation of the same with numerical results based on the wall y+ approach for various turbulent models like, Spalart Alamras, k-ε model, k-ω model and RSM models are used to solve the closure problem. The turbulence models are investigated in the commercial CFD code of Fluent using y+ as guidance in selecting the appropriate grid configuration and turbulence model. The experiment is carried out at mass averaged mean velocity of 40m/s and the geometry of the duct is chosen as rectangular cross-section of 90° curved constant area duct .In the present paper the computational results obtained from different turbulence models are compared with the experimental result along with the near-wall treatments are investigated for wall y+<30 in the region where both viscous and turbulent shear dominates and y+>30 in the fully turbulent region. It is concluded in the present study that the mesh resolving the fully turbulent region is sufficiently accurate in terms of qualitative features. Here RSM turbulence model predicts the best results while comparing with the experimental results.

Simulations of Mixed Convection with Internal Heat Generation Source in Two Sided Lid Driven Square Cavity

Sumer Dirbude*

Department of Mechanical Engineering, Indian Institute of Technology Kanpur, Kanpur-208 016, Uttar Pradesh, India
*Corresponding author (sumer@iitk.ac.in, subhadi.2011@gmail.com)

A laminar mixed convection in two-sided lid driven square cavity with internal heat generation is solved using finite volume method (FVM) for three representative Richardson numbers (Ri) of 0.1, 1 and 10.The external Rayleigh number is kept fixed at 10^4. Anti-parallel motions of top and bottom lids are considered. Top and bottom moving walls are maintained respectively at hot and cold temperature, whereas side walls are insulated. It is found that the velocity and temperature field is a function of wall motions and Ri. The heat transfer rate is an increasing function of decrease in Ri. With the internal heat generation ($Ra_I=10^4$) the average heat transfer rate from top (hot) and bottom (cold) moving wall is found to be same although they are maintained at different temperatures.

Page - 45

Proc. of the 5th International Conference on Advances in Mechanical Engineering (ICAME-2011), June 06-08 2011
S.V. National Institute of Technology, Surat – 395 007, Gujarat, India

Numerical Investigation of flow through Annular Curved Diffusing duct

P. K. Sinha[1], A.N. Mullick[2*], B. Halder[3], B. Majumdar[4]

[1]DIATM, Durgapur-713212, India
[2, 3]NIT, Durgapur – 713 209, India
[4]Jadavpur University, Kolkata, India
*Corresponding author (e-mail:anmullick@gmail.com)

In the present investigation of the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.273 and centerline length was chosen as three times of inlet diameter. The experimental results then were numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with different area ratios varying from 1.15 to 3.5. The measurements were taken at Reynolds number 2.0 x 105 based on inlet diameter and mass average inlet velocity. Predicted results of coefficient of mass averaged static pressure recovery (30%) and coefficient of mass averaged total pressure loss (21%) are in good agreement with the experimental results of coefficient of mass averaged static pressure recovery (26%) and coefficient of mass averaged total pressure loss (17%) respectively. Standard k-ε model in Fluent solver was chosen for validation. From the parametric investigation it is observed that static pressure recovery increases up to an area ratio of 2.86 and between the area ration 2.86 to 3.5, pressure recovery decreases steadily. The coefficient of total pressure loss almost remains constant with the change in area ratio for similar inlet conditions.

Keywords: *Annular curved diffuser, k-ε model, Fluent solver, Five-hole probe.*

Nomeclature

A_r	Area ratio	D	Inlet diameter of the Diffuser
A_s	Aspect ratio	L	Centerline length of the Diffuser
CC	Concave or outward wall	Re	Reynolds number
C_{PR}	Coefficient of pressure recovery	$\Delta\beta$	Angle of turn of the center line
CV	Convex or inward wall		

1. Introduction

The objective of the present study is to investigate the flow characteristics within a circular annular curved diffuser. The performance of an annular curved diffuser is characterizes by static pressure recovery and total pressure loss coefficients.

The first systematic studies on 2-D curved subsonic diffusers were carried out by Fox & Kline [1]. The centerline of the diffuser was taken as circular with a linearly varying area distribution normal to the centerline. They established a complete map of flow over a range of the L/D ratio and at different values of $\Delta\beta$. Seddon [2] has made extensive experimental investigations to explain the self-generated swirl within the S-shaped diffuser of rectangular to circular cross-section having A_r =1.338. He observed a significant improvement in the performance and exit flow distribution by introducing fences of 10 different configurations within the first bend of the diffuser.

Vakili *et al.* [3] reported the experimental studies in an S-shaped diffusing duct of $\Delta\beta$ = 30°/30° having circular cross-section and A_r =1.5. Yaras [4] experimentally investigated the flow characteristics of 90° curved diffuser with strong curvature having A_r =3.42 for different values of inlet boundary layer thickness and turbulence intensity. Reichert and Wendt [5] experimentally studied the effect of vortex on the flow field of a diffusing S-duct with $\Delta\beta$ = 30°

Page -46

12.8.12

CFD Analysis of Solid-Liquid Suspension Flow in a Horizontal Pipe

Arti tiwari and Triloki Nath

Department of Applied Mechanics, MNNIT Allhabad, 211004 , UP Email.Add:

Abstract: The present work focuses on the three-dimensional modeling of the solid –liquid suspension flow based on the Eulerian-Eularian multiphase flow model. The k-ε turbulence model and particle- induced turbulence model were used to simulate the solid-liquid suspension flow. The momentum transfer between the solid and liquid phases including drag force, shear induced lift force and virtual mass force, together with buoyancy force were considered in the model. The simulation geometry focuses on a horizontal pipe. The simulation results showed that the suspension status of solid particles depends on the input flow rate.

Keywords: solid-liquid suspension, flow dynamics in horizontal pipe, CFD modeling

Characteristics of Fluid Flow through Microchannels

Meenakshi Mour, Debarun Das and Amar N. Mullick

Department of Mechanical Engineering, National Institute of Technology, Durgapur-713209, India

Abstract. The microfluidics devices have recently attracted tremendous interest due to its potential of bringing novel applications into reality in various fields. However, the challenges in the design of microfluidics devices still remain since all aspects of fluid flow in microchannels have not been yet fully understood. Microfluidics is a vast and rapidly evolving research field. This report presents major findings in the literature on fundamentals of flow physics in microchannels. The review is intended to provide an extensive overview on the available knowledge base as well as the areas that require intensive investigation. It includes an extensive parametric study on effect of the wall roughness, entrance conditions, friction factor and pressure drop on flows in microchannels.

Keywords: microfluidics, microchannels, friction factor

Flow Investigation through Annular Curved Diffusing Duct

P. K. Sinha[a], A.N. Mullick[b], B. Halder[b] and B. Majumdar[c]

[a] *Department of Mechanical Engineering, DIATM, Durgapur – 713 212, India*

[b] *Department of Mechanical Engineering, NIT, Durgapur – 713 209, India*

[c] *Department of Power Engineering, Jadavpur University, Kolkata – 700 09, India*

Abstract. In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.25 and centerline length was chosen as three times of inlet diameter. The experimental results then were numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with different area ratios varying from 1.25 to 2 with change in angle of turn 30° to 75°. The measurements were taken at Reynolds number 2×10^5 based on inlet diameter and mass average inlet velocity. Predicted results of coefficient of mass averaged static pressure recovery (30%) and coefficient of mass averaged total pressure loss (21%) are in good agreement with the experimental results of coefficient of

page-47

Proceedings of the 4th International & 37th National Conference on Fluid Mechanics and Fluid Power

FMFP2010

December 16-18, 2010, IIT Madras, Chennai, India

FMFP2010-273

FLOW INVESTIGATION THROUGH ANNULAR CURVED DIFFUSER

Prasanta K Sinha
Department of Mechanical Engineering
Durgapur Institute of Advanced Technology & Management, Durgapur, WB, India
pksinha@yahoo.com

Biswajit Haldar
Department of Mechanical Engineering
National Institute of Technology
Durgapur, WB, India
jeetarkaanik@yahoo.co.in

Amar N Mullick
Department of Mechanical Engineering
National Institute of Technology
Durgapur, WB, India
anmullick@gmail.com

Bireswar Majumdar
Department of Power Engineering
Jadavpur university, WB, India
bmaj3255@gmail.com

ABTRACT

In the present investigation the distribution of mean velocity, static pressure and total pressure are experimentally studied on an annular curved diffuser of 30° angle of turn with an area ratio of 1.27 at Reynolds number 2 x 10^5 based on inlet diameter and mass average inlet velocity. The experimental results then were numerically validated with the help of Fluent and then a series of parametric investigations are conducted with same centre line length and inlet diameter but with different area ratios varying from 1.15 to 3.5.

The maximum values of the mass average static pressure recovery and loss total pressure loss are 26% and 17% compared to the predicted results of 30% and 20% respectively, which shows a good agreement between the experimental and predicted results.

From the parametric investigation it is observed that static pressure recovery increases up to an area ratio of 2.86 and beyond this the pressure recovery decreases steadily. The coefficient of total pressure loss almost remains constant with the change in area ratio for similar inlet conditions.

Keywords: *Annular curved diffuser, k-ε model, Fluent solver, Five-hole probe.*

INTRODUCTION

Diffusers are used in many engineering application to decelerate the flow or to convert the dynamic pressure into static pressure. Depending on application, they have been designed in many different shapes and sizes. The annular curved diffuser is one of such design and is an essential component in many fluid handling systems. Annular diffusers are an integral component of the gas turbine engines of high-speed aircraft. It facilitates effective operation of the combustor by reducing the total pressure loss. The performance

Page -48

Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power

FMFP2010

December 16-18, 2010, IIT Madras, Chennai, India

FMFP2010 - 257

FLOW DEVELOPMENT IN A CONSTANT AREA CURVED DUCT

A. K. Biswas
National Institute of Technology
Durgapur, W.B., India
arupkumar.biswas@me.nitdgp.ac.in

A. N. Mullick
National Institute of Technology
Durgapur, W.B., India
amarnath.mullick@me.nitdgp.ac.in

B. Majumdar
Jadavpur University
Kolkata, W.B., India

ABSTRACT

The present study dealt with the wall static pressure of a rectangular curved duct (with $\Delta\beta$=90°) at all four faces with various locations. The normalized wall static pressure distributions at the top, bottom, inside and outside surfaces of the C-duct are drawn in the form of contours at three different air velocities U_{av} = 20m/s, 40m/s and 60m/s respectively. These iso-bar contours indicate towards the presence of secondary flow in the duct in the form of a pair of contra-rotating vortices for all the three air velocities.

Keywords: *Curved Duct, Secondary Motion, Wall pressure, Wind Tunnel.*

INTRODUCTION

C-ducts are used in aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. Heat exchangers in the form of curved ducts are used widely in food processing, refrigeration and hydrocarbon industries. Gas turbine engine components such as turbine compressors, nozzle etc. utilise several complex duct configuration. Performance of duct flow depends upon the geometrical and dynamical parameters of the duct. So it is very much essential to design the duct with proper geometry to improve the performance.

Study of flow characteristics through constant area ducts is a fundamental research area of basic fluid mechanics since the concepts of potential flow and frictional losses in conduit flow were established. Duct is a part and parcel of any fluid-mechanical system. It is a passageway made of sheet metal or other suitable material used for conveying air or other gases or liquids at different pressures. Depending on its application the shape the duct may be either of straight, curved, annular, polar, sector, trapezoidal, rhombic etc. Flow through curved ducts has practical importance in chemical and mechanical industries in particular. Obviously, compared to a straight duct, flow in a curved duct is more complex due to curvature of the duct axis. It induces centrifugal forces on the flowing fluid resulting in the development of a secondary motion (normal to primary direction of flow) which is manifested in the form of a pair of counter-rotating vortices. Depending on the objective, fluid mechanical systems often demands for the design of ducts with complex

page-49

Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power

FMFP2010

December 16-18, 2010, IIT Madras, Chennai, India

FMFP2010 - 256

AN EXPERIMENTAL INVESTIGATION ON FLOW THROUGH A CURVED DIFFUSER

Nirmal K. Das
Bengal College of Engg & Technology
Durgapur, West Bengal, India
nkdas_dgp@yahoo.co.in

A.N. Mullick
NIT Durgapur
West Bengal, India
amarnath.mullick@me.nitdgp.ac.in

B. Halder
NIT Durgapur
West Bengal, India
jeetarkaanik@yahoo.co.in

P. Ray
NIT Durgapur
West Bengal, India
pramnitd@yahoo.com

B. Majumdar
Jadavpur University
Kolkata, West Bengal, India
b_maj3255@yahoo.com

Keywords: *C-shaped diffuser; aspect ratio; 2-D isobars; wall pressure; secondary motion.*

ABSTRACT

In the present paper static pressure was measured on the walls of a C-shaped diffuser of rectangular cross-section with inlet aspect ratio and area ratio of 2.0, at three different inlet velocities, at incompressible flow regime. 2-D isobars were plotted with the normalized wall-pressure. The wall pressure distribution depicts the development of secondary motion, as a result of the centrifugal effect due to centerline curvature and the subsequently developed favorable transverse pressure gradient. The pattern of flow development was more or less similar for the different inlet velocities and a continuous rise in wall-pressure along flow direction was observed due to diffusion.

INTRODUCTION

Diffuser is a mechanical device usually made in the form of a gradual conical expander intended to raise the static pressure of the fluid flowing through it. Specific use of a diffuser with a proper area ratio should also ensure an adequate amount of uniform flow at the outlet together with a considerable static pressure. Diffusers play an important role in many engineering and industrial applications to accomplish the objectives under geometric constraints. In aircraft gas turbines, high velocity air from the wing or fuselage first flows through a diffusing intake where it is decelerated for increasing the pressure, and then fed to the engine compressor. In the design of ventilation and air conditioning systems, diffusers are commonly used for discharging the conditioned air into the space to be cooled in order to reduce the air velocity and increase the static pressure. In these applications, on one hand there is restriction on space as well as design compatibility to match with the shortest possible duct length, while on the other hand, given cross-sectional

Ist International Conference on "New Frontiers in Biofuels, DTU"
January 18-19, 2010, New Delhi

ID: 2010/NFIB/048
Biofuels- A Healthier and Wealthier Future for India

Shankaransh Srivastava*, Nirupam Kar and A. N. Mullick
Department of Mechanical Engineering, National Institute of Technology, Durgapur
*Email ID: shankaransh_dps@yahoo.com

ABSTRACT

The global thirst for energy is constantly increasing. Amongst rising energy prices and increasing concern over global environmental issues, searching for alternative and cleaner energy sources has become the prime objective for the energy industry and the governments of various countries. At the moment fossil fuels dominate the world energy scenario may it be liquid or solid. India currently imports 75% of their total petroleum consumption, with imports set to rise to as much as 94% in the next two decades. With such alarming figures India desperately needs a new lease of life, which can be provided only by biofuels.

India as a market has tremendous potential when it comes to biodiesel, considering the percentage of diesel fuel consumed in agriculture, transport or power generation. Proper channelizing and usage of resources can help in tapping these markets. This study shall contain an overview of the present resources of India in terms of land, man power and government policies and how all these can be brought in tandem with each other in order to make India a major player in the biofuel sector.

Keywords: Biodiesel, Hemp, Jatropha, Transesterification, Emissions, Wastelands.

1. INTRODUCTION

The last one and a half decades has escalated the concerns of developed as well as developing powers of the world over the increase in global warming and environmental meltdown occurring due to massive carbon footprints left by them due to exponentially amplified demand in transportation and agro purposes [1]. Alternatives ranging from hydrogen based vehicles to natural gas vehicles have been developed and it has been found that none of them are as effective and efficient as biofuels especially biodiesels both from commercial as well as environmental perspective. Considering the potential in agro based industries and transport sector, in India the scope for replacement of a substantial amount of conventional fuel (diesel oil) by biodiesel is huge and a clear trend in that direction has already begun. In last couple of years, the production and consumption of biofuels have entered a new era of global growth, experiencing expansion both in the scale of the industry and the number of countries involved.

Shooting up of oil prices across the globe is sizeable due to the rise in demand in comparison to the meagre 7% appreciation in world oil production has lead to development of more efficient conversion technologies and introduction of stronger government policies that have resulted in huge investments in biofuel production [2]. The production of fuel ethanol has more than doubled since 2000, while production of biodiesel has expanded nearly four-folds [3]. Compared to petroleum, the use of biofuels for transport is still quite low in all the countries. By far the largest production and use of ethanol is in Brazil and the United States, with almost similar volumes, but much higher than any other country [4]. Currently, ethanol is blended with gasoline and biodiesel is blended with petroleum-based diesel for use in conventional diesel-fuelled vehicles.

However in India, considering the market potential in diesel usage as well as plantation potential on wastelands and hilly areas, biodiesel obtained from non-edible oils like Hemp, Jatropha, etc. which are low maintenance plants need to be given more prominence hence biodiesel has the potential to leap-frog the developments for the alternative fuel. Fuels derived from renewable biological resources for use in diesel engines are known as biodiesel. Biodiesel is an environment friendly liquid fuel similar to petrol-diesel in combustion properties. It is an oxygenated fuel containing 10%-15% oxygen by weight. Also it can be said a sulphur-free fuel [5]. These facts

Page - 51

Proceedings of the ASME 2009 International Mechanical Engineering Congress & Exposition
IMECE2009
November 13-19, Lake Buena Vista, Florida, USA

IMECE2009-12456

FLOW INVESTIGATION THROUGH A 30° TURN DIFFUSING DUCT IN SUBSONIC FLOW REGIME

Prasanta K Sinha
Durgapur Institute of Advanced Technology & M anagement, Durgapur, WB, India

Biswajit Haldar
Department of Mechanical Engineering
National Institute of Technology
Durgapur, WB, India

Amar N Mullick
Department of Mechanical Engineering
National Institute of Technology
Durgapur, WB, India
amarnath.mullick@me.nitdgp.ac.in

Bireswar Majumdar
Department of Power Engineering
Jadavpur University, WB, India

ABSTRACT

Curved diffusers are an integral component of the gas turbine engines of high-speed aircraft. These facilitate effective operation of the combustor by reducing the total pressure loss. The performance characteristics of these diffusers depend on their geometry and the inlet conditions. In the present investigation the distribution of axial velocity, transverse velocity, mean velocity, static and total pressures are experimentally studied on a curved diffuser of 30° angle of turn with an area ratio of 1.27. The centreline length was chosen as three times of inlet diameter. The experimental results then were numerically validated with the help of Fluent, the commercial CFD software

The measurements of axial velocity, transverse velocity, mean velocity, static pressure and total pressure distribution were taken at Reynolds number 1.9×10^5 based on inlet diameter and mass average inlet velocity. The mean velocity and all the three components of mean velocity were measured with the help of a pre-calibrated five-hole pressure probe. The velocity distribution shows that the flow is symmetrical and uniform at the inlet and exit sections and high velocity cores are accumulated at the top concave surface due to the combined effect of velocity diffusion and centrifugal action. It also indicates the possible development of secondary motions between the concave and convex walls of the test diffuser. The mass average static pressure recovery and total pressure loss within the curved diffuser increases continuously from inlet to exit and they attained maximum values of 35% and 14% respectively. A comparison between the experimental and predicated results shows a good qualitative agreement between the two. Standard k-ε model in Fluent solver was chosen for validation. It has been observed that coefficient of pressure recovery C_{pr} for the computational investigation was obtained as 38% compared to the experimental investigation which was 35% and the coefficient of pressure loss is obtained as 13% in computation investigation compared to the 14% in experimental study, which indicates a very good qualitative matching.

INTRODUCTION

Flow development within a diffuser, especially curved diffuser is complicated in nature and depends on several geometrical and dynamic parameters, which also

Page -52

20

Proceedings of ICTACEM 2007
International Conference on Theoretical, Applied and Experimental Mechanics
December 27-29, IIT, Kharagpur, India
ICTACEM-2007/048

A computational approach to analyze the secondary motions in a Twin Intake Duct

Amar N Mullick[a] and Bireswar Majumdar[b]

[a] National Institute of Technology, Durgapur 713209, India
[b] Jadavpur University, Kolkata 700098, India

ABSTRACT

Twin Intakes are mainly used in modern fighter aircraft with single engine embedded in the fuselage typically consists of a pair of twin limbs, which merge into a single duct. In the present paper, two limbs of the duct are taken as S-shaped and are symmetrical about their central plane. The intake of the aircraft is an important component, which affects the propulsion as well the aerodynamic characteristics of the aircraft. The task of the air intake is to decelerate the different flow conditions ranging from subsonic to supersonic speed of the external flow to a speed acceptable to the compressor inlet. In the present paper, an experimental study was made of an incompressible subsonic flow through a model intake duct of rectangular cross-section. The operating Reynolds number based on the inlet hydraulic mean depth is maintained at 3.17 x 105. The geometry of the model twin intake is taken as two S-shaped diffusing passage of angle of turn as 30°/30° and centerline length as 300 mm each. A pre-calibrated multi hole pressure probe took measurements at the different section of the intake duct. The investigation involved comparing the predicted velocity distribution and experimental results at the various planes of the intake duct. For the predicted results, a commercial CFD code Fluent[1] was used. Different turbulence models available within the code were tested and it has been observed that the Reynolds Stress Model (RSM) of turbulence provided the best results. The comparison was made on the results of longitudinal velocity distributions in the form of iso-lines for experimental and computational work. Coefficient of pressure recovery and coefficient of pressure loss also compared between the experimental and computational work. It has been observed that the patterns of the iso-lines of longitudinal velocity distributions are very close to each other in both the cases. Moreover, the coefficient of pressure recovery and coefficient of pressure loss are within 6% of the experimental results. This agreement confirms that the CFD code using Reynolds Stress Model can predict the flow characteristics reasonably well for similar geometries with identical boundary and operating conditions used during the simulation. The primary objective of the present study is to acquire the knowledge and understanding of the flow development through a twin intake duct. Moreover, the experimental data are used as a benchmarking for the validation of CFD code. Based on these agreements the predicted transverse velocity distribution at the merger plane of the intake duct is discussed in the present paper. From the predicted results, it has been observed that the two pairs of contra-rotating vortices are generated at this section and the same corroborate the experimental results.

Keywords: Twin Intake Duct, Merger Plane, Fluent Code, Transverse Velocity, Multi-hole probe.

1. INTRODUCTION

Modern high performance single engine aircrafts consists of in air intake duct to convey air to the compressor face of the engines. Air intakes are large aircraft component, which requires synchronized integration in to the overall design of the aircraft so that the efficiency of the aircraft does not deteriorate. The main problem of the air intake design is to ensure that the

---

* Further author information: (Send correspondence to [a])
[a]: E-mail: anmullick@gmail.com, Telephone: +919830385342(M), Applied Mechanics Department, NIT, Durgapur, Mahatma Gandhi Avenue, Durgapur 713209, India.
[b]: E-mail: b_maj3255@yahoo.com, Telephone: +919433213671

page - 53

71

A Computational Approach to Analyze the Performance of a Twin Intake Duct

A. N. Mullick[1], B. Majumdar[2] and S. Chakravorty[3]

Abstract

Single engine aircrafts consists of a pair of twin intake, which is symmetrical about their central plane and embedded in the fuselage of the aircraft. The two limbs of the twin intake duct merged at a section beyond which the duct has a single outlet symmetric about its axis in vertical plane. In the present paper a computational approach has been adopted to validate the numerical results and also the prediction of other flow conditions through a parametric study. The validation has been done by using Fluent CFD code. A twin intake duct of rectangular cross-sectional area of aspect ratio=2.0, area ratio=1.5, centerline length=300 mm and angle of turn=30°/30° has been used for the present analysis. The operating Reynolds number based on the inlet hydraulic mean depth and mass averaged mean velocity is maintained at 3.17×10^5. The geometry of the intake duct is taken as two S-shaped diffusing passage merged to form a single duct. The investigation involved comparing the experimental results and the predicted results of the distribution of flow parameters at the merger and exit sections. In the present paper Reynolds Stress Model (RSM) of turbulence is used for numeric analysis, with ε value is taken as 0.8. The inlet condition of the numerical analysis is made exactly same with the experimental study. It has been found that the numeric results match both qualitatively and quantitatively with the experimental results. The longitudinal velocity distributions in the form of iso-lines for both experimental and numerical work are presented in this paper. The agreement between the results confirms that the CFD code using RSM model can predict the flow and performance characteristics reasonably well for intake ducts of similar geometries and same boundary conditions.

The comparison of the performance parameters like coefficient of pressure recovery and coefficient of pressure loss for both the experimental and numerical work are also presented in this paper. It has been observed that coefficient of pressure recovery for the numerical investigation is obtained as 44% compared to the experimental work which is 41%. Similarly the

[1] Assitant Professor, Applied Mechanics Dept., NIT, Durgapur, Durgapur-703209, India.
[2] Professor, Power Engg. Dept., Jadavpur University, Kolkata -700098, India.
[3] Professor, Electrical Engg. Dept., Jadavpur University, Kolkata-700032, India.

A Study of Wall Pressure Distribution of an Axisymmetrical Y-shaped Air Intake

Amar Nath Mullick
Mechanical Engineering Department,
FIEM, Kolkata,
anmullick@vsnl.net

Bireswar Majumdar
Power engineering Department,
Jadavpur University, Kolkata,
b_maj3255@yahoo.com

ABSTRACT

An experimental study was made of an incompressible subsonic flow through a model Y-intake duct of rectangular cross-sectional area of Aspect ratio=2.0 and Area ratio=1.5. The operating Reynolds number based on the inlet hydraulic mean depth and mass averaged mean velocity is 3.17×10^5. The geometry of the model Y-intake was taken as two S-shaped diffusing passages having angle of turn 30°/30° with centerline length of 300 mm of each. The measurements indicate that there is a continuous increase in static pressure on both the straight parallel walls. The high pressure zones are located at the exit of the diffusing passages. In the present study, Y-intake duct flow was studied in order to document the detailed flow data for validation and comparison with numerical code and prediction of various flow parameters. Experimental results indicate that the generation of secondary motion in the form of contra-rotating vortices. Experimental results also suggest that the details of the near wall flow incipient vortical motion are likely to provide a vigorous test for various computational methods and turbulence models.

KEYWORDS

Y-intake duct, Turbulent flow, S-shaped diffuser, Contra-rotating vortex.

INTRODUCTION

The understanding of flow development in Y-shaped diffusing ducts is relevant to a vast range of aeronautical and industrial applications. In such applications they may be integrated to intake ducts to convey air to the compressor face of aircraft engines with dorsal or wing-rooted intakes and are critical to the performance of the entire propulsion system. It is a matter of great importance to make an appropriate design of this type of air intake ducts for the modern high performance aircraft engines. Air intakes are fairly large aircraft components, which require synchronized integration in to the overall design of the aircraft so that the efficiency of the engine does not impaired. The main problem of the air intake design is to ensure that the turbine aero engine is properly supplied with air under all conditions of the aircraft operation. The task of the air intake is thus to decelerate the high speed external flow to a speed acceptable to the compressor. Depending on the designed speed of the aircraft the different intake types are employed. The geometrical variations along with diffusion of flow make the flow characteristics of these ducts very complicated in nature.

The intake duct divides the oncoming airstreams into an internal flow and an external flow. The internal flow has the positive duty of feeding the engine whereas the external flow preserves the proper aerodynamics of the airframe.

The characteristic of the design point of the intake is that the cross-section of the captured area is a maximum, corresponding to the maximum airflow requirement of the engine. The condition applies to one particular flight Mach number, the engine thrust setting is changed by altering engine RPM and thus the pressure at the compressor inlet will change accordingly. In the S-shaped diffusing passage, centrifugal forces originating as a result of curvature of the limbs of the diffusing passage have a tendency to nullify the adverse pressure gradient resulting from the diffusing action. The rapidly moving fluid located at the central part of the flow passage has a tendency to move in axial direction but forced towards the outer wall. In order to satisfy the continuity of flow the secondary motion generated within the flow passage. Thus the intricacy of the design of an air intake duct is to guide the fluid stream with pressure and flow uniformity at the downstream of the intake exit.

The Y-intakes are generally referred to a pair of intakes on the two sides of a fuselage, feeding a single engine via a common duct or plenum chamber. These types of intake are widely used for ingestion of atmospheric air to single engine aircraft. Again, the design of such intakes generally aims to provide the shortest possible duct length for a given cross-sectional area in order to minimize the hydraulic loses and flow distortion at the exit by avoiding separation. Moreover, due to the space constraints in the downstream assembly units, the shape of the flow passage generally taken as curved to maintain the overall diffusion and pressure recovery. The centerline offset of the intakes with respect to the engine is a real challenge in designing the intake conforming to the constraints imposed by other aspects of the aircraft design.

It has been observed through an intensive survey of literatures that the reported results are very few in open journals because the studies on Y-intakes are mainly restricted in aircraft industries and defence sectors. These limitations and zeal for

Page - 55

Annexure III

Details of Ph D Students (Guided and Ongoing)

S.No	Name of Student	Year	Title of the Dissertation	Status(Completed /Ongoing)
1	Prasanta Sinha	2006	Flow Investigation through Annular Curved Diffuser	Degree Awarded in 2011
2	Arup Kumar Biswas	2011	Development of Flow through Constant area Duct in Turbulent Regimes	Degree Awarded in 2015
3	Vidyasagar Bhattacharjee	2011	Numerical Simulation of Bifurcated Air Intake	Ongoing
4	Deepak Verma	2012	Numerical Simulation on the Performance of Continuous	Ongoing
5	Bibhuti Bhusan Nayak	2013	Numerical Study on the Thermo-Fluidic Transport of Fly Ash-Water Slurry in Horizontal Pipeline	Thesis submitted
6	Rahul Mishra	2014	Hydrodynamic and Erosion Studies for Flow of Particulate Slurries	Ongoing
7	Sudhanshu Mishra	2014	Hydrodynamic and Thermal Studies for Flow of Particulate Slurries	Ongoing

Annexure IV

18 Administrative/Institute Support Work:

S.No.	Section/office/Institute level committee	From	To	Position held	Responsibilities
1.	Served as Hostel Warden Hall No 6 and A1 & A7 of the Institute	August 07	July 08	Warden Hall 6 & Hall A1 & A7	To look after the day to day work of the Hostel and also to maintained the discipline of the Hall
2.	Served as joint chairman of Anti-ragging Committee for Hall VI	August 07	July 08	Joint Chairman	
3.	Served as Hostel Warden for Extended PG Hall	Sept 07	July 08	Additional responsibility	To look after the day to day work of the Hostel and also to maintained
4.	Served as Lab-in-Charge Fluid Mechanics & Hydraulic Machine Lab and Aerodynamics Lab	July 08	June 15	In-Charge	To look after the day to day work and procured new Lab Set up and developed Aerodynamics Lab
5.	Served as Warden Hall No. 2 of the Institute	Aug 08	July 10	Warden Hall No. 2	Same as above-
6.	Anti-Ragging Committee of the Institute	August 09	April 14	Member	
7.	Served as Associate Dean, Student Welfare of the Institute	August 09	Apr 14	Associate Dean, Student Welfare	To assist the day to day work of Dean, Student Welfare, To look after the Scholarships for all the students.
8.	Served as Professor-in-Charge Post Graduate Examination	Nov 09	Apr 14	In-Charge	To look after all the activities including conduct of Examinations and publication of results of various examination of the Institute.
9.	Convener of Institute Scholarship Committee	Nov 09	Apr 14	Convenor	
10.	Served as Chairman Departmental Purchase Committee	Nov 09	July 15	Chairman	
11.	Served as Committee Member of Hostel Mess Committee	Feb 10	Apr 14	Member	
12.	Serving as Centre-in-Charge of West Bengal Joint Entrance Examination	Feb 12	Till date	Centre-in-Charge	
13.	Served as Joint Centre-in-charge of CCMT 2012 & 2013	2012	2013	Joint Centre-in-Charge	
14.	Serving as Head of Department of Mechanical Engineering Department	2016	Till now	Head	

15.	Serving as Member of Academic and Professional ethic committee	Jan 17	Till now	Member	
16.	Preparation of Draft Concept Paper on Roles and Structures of Schools, Department, Centres (Services or Research).	Apr 17	Till now	Member	
17.	For finalisation, the terms, conditions, and contract value of AMC of the Air-conditioning System at S N Roy Memorial Building.	June 17	Till now	Member	

Page - 58

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
(DEEMED UNIVERSITY)

No.:NITD/Admn/2701 Dated: July 31, 2007

OFFICE ORDER

As approved by Director, the following faculty members are hereby appointed as Wardens of different Halls of Residence of the Institute w.e.f. 01.08.2007 (F/N). They are entitled to receive remuneration as per Institute rules. This office order supersedes earlier office order in this respect.

The services rendered by relieved Wardens are highly appreciated and gratefully acknowledged. Their cooperation and advice is solicited for any future activities of the Hostels.

List of Wardens:

Hall No.	Name of Wardens
I	1. Prof. A. K. Bhattacharya, ECE Dept
	2. Dr. A. K. Patra, Chemistry Dept.
II	1. Dr. S. Banerjee, EE Dept
	2. Dr. A. B. Puri, ME Dept.
III	1. Prof. S. S. Thakur, EE Dept.
	2. Dr. J. Banerjee, Humanities Dept.
IV	1. Dr. A. K. Meikap, Physics Dept.
	2. Dr. D. Sukul, Chemistry Dept.
V	1. Prof. K. C. Ghanta, ChE Dept.
	2. Shri A. Layek, ME Dept.
VI	1. Dr. A. N. Mullick, A. M. & Drwg. Dept.
	2. Shri M. K. Mondal, Physics Dept.
IX	1. Prof. M. C. Majumdar, ME Dept.
	2. Dr. B. P. Mukhopadhyay, Chemistry Dept.
	3. Dr. S. C. Moi, Chemistry Dept.
	4. Shri S. Sahoo, Physics Dept.
LH (Nivedita)	1. Mrs. T. Pal (Debroy), CSE Dept.
	2. Dr. (Ms.) Jayati De, EE Dept.
LH (Preetilata)	1. Shri G. Banerjee, DMS
	2. Dr. (Mrs.) D. Dutta, Bio-Tech. Dept.

(A. K. Mitra) 31.07.07
Dean (Admn)

Copy to:

1. Secretary Wardens' Council
2. Prof. - In-Charge (Hostels)
3. Prof. -In-Charge (Estate)
4. Relived Wardens
5. Reliving Wardens
6. Dy. Registrar (Admn)
7. Dy. Registrar (A/Cs)
8. Mess Managers / Matron LH
9. Medical Officer
10. PA to Director
11. File Copy

Page -59

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
(DEEMED UNIVERSITY)
MAHATMA GANDHI AVENUE – 713 209, WEST BENGAL

OFFICE ORDER

NITD/59/Hall- VI/ Dated: Aug 01, 2007

Sub: Anti-ragging committee for Hall - VI

An Anti-Ragging Committee for Hall – VI is formed for prevention of ragging in the premises of Hall – VI with immediate effect as here under:

1. Warden – I, Faculty - Joint Chairman
2. Warden – II, Faculty - Joint Chairman
3. Shri Rajdeep Kar, Roll No. 06/CSE/4108, Room No. 131 - Member
4. Shri Ram Prasad Panda, Roll No. 06/MBA/56, Room No. 239 - Member
5. Shri Rajib Dey, Roll No. 06/MBA/59, Room No. 335 - Member
6. Shri Saswat, Roll No. 06/MBA/05, Room No. 105 - Member
7. Shri Samnanya Ray, Roll NO. 06/CSE/4103, Room No. 124 - Member
8. Shri Sourave Neogi, Roll No. 06/MBA/55, Room No. 315 - Member
9. Shri Tapan Samadder, Mess Manager, Hall - VI, - Convenor

(S. Bhattacharya)
Director

Copy to:

6. Warden – I, Joint Chairman, Anti Ragging Committee, Hall – VI
7. Warden – II, Joint Chairman, Anti Ragging Committee, Hall - VI
8. Convenor , Anti Ragging committee, Hall - VI
9. All Concerned members
10. Hall – VI Notice Board

Page - 60

NATIONAL INSTITUTE OF TECHNOLOGY – DURGAPUR
(DEEMED UNIVERSITY)

No.: NITD/59

Dated: September 14, 2007

OFFICE ORDER

In suppression of earlier order in this regard the following Wardens of Hall – VI are assigned additional responsibility to act as Warden for extended PG Hall of residence in Building No. A-7 & other Annex PG Buildings with immediate effect.

i. Dr. A. N. Mullick, Assistant Professor, A. M. & Drg. Dept.
ii Shri M. K. Mondal, Lecturer, Physics Dept.

(Prof. A. K. Mitra) 14.09.07
Prof. – In – Charge)

Copy to:

i. Dr. A. N. Mullick, Assistant Professor, A. M. & Drg. Dept.
ii. Shri M. K. Mondal, Lecturer, Physics Dept.
iii. Dr. A. K. Bhattacharya, Secretary Wardens' Council
iv. Dr. P. Ray, Professor – In – Charge Hostels'
v. Dy. Registrar (A/Cs)
vi. Mess Manager – Building No. A-7 & Annex PG Building.
vii. File Copy

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR

MAHATMA GANDHI AVENUE, DURGPAUR – 713 209 (W.B.) INDIA

No.: NITD/59

Dated: July 31, 2008

OFFICE ORDER

In supersession of all previous orders in this regard and as approved by the Director the following Faculty Members are appointed as Wardens of different Halls of Residence as per the list as under, w.e.f. August 01, 2008 and until further orders, under terms & conditions as per rules.

List of Wardens:

Hall No.	Name of Warden
I	1. Dr. A. K. Patra, Chemistry Deptt. 2. Dr. M. K. Mondal, Physics Deptt.
II	1. Dr. A. B. Puri, ME Deptt. 2. Dr A. N. Mullick, AM &Drg. Deptt.
III	1. Dr. S. Banerjee, EE Deptt. 2. Dr. J. Banerjee, Humanities Deptt.
IV	1. Sri A. K. Mal, ECE Deptt. 2. Sri R. Kar, ECE Deptt.
V	1. Prof. S. S. Thakur, EE Deptt. 2. Dr. A. Layek, ME Deptt.
VI	1. Prof. A. K. Bhattacharya, ECE Deptt. 2. Sri D. Mondal, ECE Deptt.
IX	1. Dr. A. K. Meikap, Physics Deptt. 2. Dr. S. Sahoo, Physics Deptt. 3. Dr. S. C. Moi, Chemistry Deptt. 4. Sri A. Modak, Humanities Deptt
Nivedita Hall	1. Prof. A. K. Saha, ME Deptt. 2. Dr. (Mrs.) Jayati Dey, EE Deptt.
Preetilata Hall	1. Dr. D. Sukul, Chemistry Deptt. 2. Dr. (Mrs.) D. Dutta, Bio-Technology Deptt.

The Wardens of Hall VI will look after the Hall Annex in A-7. The Wardens of Preetilata Hall will look after the Hall Annex in B-9.

The services rendered by the outgoing Wardens Prof. V. P. Shukla, Dr. B. P. Mukhopadhyay and Smt. T. Pal (Debroy) are gratefully appreciated and acknowledged.

Prof-In-Charge
31.07.08

Copy to:

1. All Deans
2. All HODs
3. Secretary Wardens' Council
4. Prof. – In – Charge (Hostels)
5. Prof. – In – Charge (Estate)
6. Dy. Registrar (Admn)
7. Dy. Registrar (A/Cs)
8. Audit Officer
9. Wardens, Halls of Residence & Outgoing Wardens

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR – 713 209 (W.B.), INDIA

No.: NITD/59

Dated: July 19, 2009

OFFICE ORDER

The Anti Ragging Committee of the Institute is hereby re-constituted as follows:

1. Prof. Goutam Sanyal, Dean (Student Welfare) - Chairman
2. Prof. A. K. Mitra, Dean (Admin) - Member
3. Prof. S. B. Das, Dean (Academic) - Member
4. Prof. S. Haldar, Dean (P&D) - Member
5. Prof. P. P. Gupta, Dean (R&C) - Member
6. Prof. A. K. Bhattacharjee, Secretary Wardens' Council - Member
7. Prof. S. N. Sarkar, Prof. TPSW - Member
8. Prof. N. K. Roy, Associate Dean (Academic) - Member
9. Prof. K. C. Ghanta, Associate Dean (R&C) - Member
10. Dr. A. N. Mullick, Associate Dean (Student Welfare) - Member
11. Dr. Surabhi Choudhury, Associate Dean (Student Welfare) - Member
12. All HODs - Members
13. Registrar (I/C) - Member
14. All Wardens - Members
15. Deputy Registrar (Admin) - Member
16. Deputy Registrar (A/Cs) - Member
17. Manager (HES) - Member
18. Shri P. Chatterjee, Security Inspector - Member
19. All elected office bearer of Students Gymkhana - Members
20. Dr. J. Banerjee, Warden Hall – III - Convenor

The Committee will start functioning with immediate effect and will be valid till further order.

This order supersedes all the previous orders in this regard.

(S. Bhattacharya)
Director

Copy to: All concerned.

Page-63

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR - 713 209 (W.B.) INDIA

No.: NIT/D/60 [illegible]

Dated: August 20, 2009

OFFICE ORDER

Dr. A. N. Mullick, Assistant Professer, Dept. of Mechanical Engineering and Dr. Surabhi Chaudhury, Assistant Professor, Dept. of Bio Technology are hereby nominated as Associate Dean (Students Welfare). Dr. Chaudhury will also act as the Member of Disciplinary Committee.

This nomination is with immediate effect and will continue till further order.

(S. Bhattacharya) 20/08/09
Director

Copy to: All concerned.

Page - 64

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR – 713 209 (W.B.), INDIA

No.: NITD/60

Dated: February 17, 2010

OFFICE ORDER

An examination cell consisting of the following members is being constituted to look after all the activities including conduct of examinations and publication of results for various examinations of our Institute.

1. Prof. S. Ghosh, Dept. of Electrical Engineering
2. Dr. A. N. Mullick, Dept of Mechanical Engineering

This examination cell will start functioning immediately with an active support from Prof. I. Basak, Prof.-In-Charge Examinations.

(S. Bhattacharya)
Director

Copy to:

1. Prof. S. Ghosh, Dept. of Electrical Engineering.
2. Dr. A. N. Mullick, Dept. of Mechanical Engineering
3. Prof. I. Basak, Prof.-In-Charge, Examination
4. All Deans
5. All Associate Deans
6. All HODs / Sections
7. File copy.

Office : (0343)2546397
Fax: (0343) 2547375/2546753
E-mail: director@admin.nitdgp.ac.in
Website: www.nitdgp.ac.in

NATIONAL INSTITUTE OF TECHNOLOGY,DURGAPUR

MAHATMA GANDHI AVENUE,
DURGAPUR-713209 ,West Bengal, INDIA

No. NITD/3712 Dated: February 06, 2012

OFFICE ORDER

As approved by the Director, the Scholarship Committee of the Institute is hereby reconstituted with the following members .The committee will start functioning with immediate effect.

Sl. No.	Name of the committee member	
1.	Prof. G. Sanyal, Deptt. of CSE & Dean (Students' Welfare)	Chairman
2.	Prof. D. K. Mondal, Deptt. of MME & Dean (Academic)	Member
3.	Prof. P.P. Gupta, Deptt. of Che. E. & Dean (R & C)	Member
4.	Sri R.N. Ray, Associate Prof., Deptt. of MME	Member
5.	Dr. A.N. Mullick, Associate Prof. Deptt. of M.E. & Associate Dean(Students' Welfare)	Convener

Registrar (I/C)

Copy to:-

1. Concerned members.
2. Concerned HOD.
3. Dy. Registrar(A/Cs).
4. Manager (HES)
5. Audit Section.
6. Director Office.
7. File copy.

Audit Officer
06/02/12

Page - 66

As Associate Dean

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR
MAHATMA GANDHI AVENUE, DURGAPUR – 713 209 (W.B.), INDIA

No.: NITD/59

Dated: February 10, 2010

OFFICE ORDER

As approved by the Director, the Tender Committee for procurement of food materials for the hostels is reconstituted as under:

1. Prof. G. Sanyal, Dean (Students' Welfare) - Joint Chairman
2. Prof. A. K. Bhattacharjee, Secy. Wardens' Council - Joint Chairman
3. Dr. A. Lavek. Warden. Hall – V - Joint Convenor
4. Dr. J. Banerjee, Warden, Hall – III - Joint Convenor
5. Dr. Surabhi Choudhuri, Associate Dean (Students' Welfare)- Member
6. Dr. A. N. Mullick, Associate Dean (Students' Welfare) - Member
7. All Wardens of all Halls of Residence - Members
8. All Mess Secretaries of all Halls of Residence - Members
9. Executive Council of Students' Gymkhana - Members
10. All Mess Managers and Matrons - Members

The committee will start functioning with immediate effect.

(A. K. Mitra) 10.02.10
Prof. – In – Charge

Copy to:

1. All Committee members.
2. Registrar (I/C)
3. Deputy Registrar (Admin)
4. Deputy Registrar (A/Cs)
5. Audit Officer

Page - 67

☎: Office : (0343) 2546406 / 2752007
: Fax : (0343) 2547375
: E-mail : registrar@admin.nitdgp.ac.in
: Website : www.nitdgp.ac.in

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR

MAHATMA GANDHI AVENUE, DURGAPUR – 713209 (WEST BENGAL), INDIA

No. NITD/Estt./4P-917/Headship –M.E/2704/ 2016 Date : November 16th , 2016

OFFICE ORDER

Prof., Amar Nath Mullick, Prof., Department of Mechanical Engineering of the Institute is hereby appointed as Head, of the Department of Mechanical Engineering with immediate effect for a period of two years.

The services rendered by Prof. Nilotpal Banerjee, as head of the Department of Mechanical Engineering is appreciated and acknowledged.

This issues with the approval of the competent authority.

Registrar 16/11/16

Copy to:-

1. Director.
2. Registrar .
3. All Deans.
4. All Head of the Departments/ Sections.
5. Prof., Nilotpal Banerjee, Prof., Department of M.E - With a request to hand over the charges of HOD, as per format attached (five copies).
6. Prof. A. N. Mullick, Prof, Department of M.E - With a request to take over the charges of HOD, as per format submitted to Prof. Nilotpal Banerjee.
7. Director / Registrar Secretariat
8. File copy.

Dy. Regr.(Estt.) 16/11/16

JKR 16/11/16

Page - 68

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR

MAHATMA GANDHI AVENUE, DURGAPUR – 713 209, INDIA

Ref. No. NITD/Regis/752/OR

Date: 5/4/2017

OFFICE ORDER

As approved by the Senate of the Institute in its 44th meeting held on 09.01.2017 vide item no. 44.13, the Academic and Professional ethic committee of the Institute is hereby constituted with the following faculty members.

Sl. No.	Name	Remarks
01.	Prof. K.L. Chopra	Chairman
02.	Prof. P. Gupta (Dean- AIRO)	Member
03.	Prof. A Gongopadhyay (Dean-FW)	Member
04	Prof. K. Bhattacharyya (Dean -P&D)	Member
05.	Prof. S. Chottopadhyay (Dean- R&C)	Member
06.	Prof. S. Ghosh (Dean-Academics)	Member
07.	Prof. N. Banerjee (Dean-SW)	Member
08.	Prof. A.K. Meikap (HoD, Physics)	Member
09	Prof. A.N. Mullick (HoD, ME)	Member

This issues with the approval of the competent authority.

05/4/17

Registrar

Copy to:

1. Director
2. All Members
3. All HoDs
4. File Copy

National Institute of Technology, Durgapur

Committee for Preparation of Draft Concept Paper on Roles and Structures of Schools, Departments, Centres (Services or Research), Cells, Facilities etc.

Date: May 1, 2016

Notice

A meeting of the committee will be held on May 5 (Thursday), 2016 at 11:00 a.m. in the Academic Board Room for the following agenda items. All committee members are hereby requested to kindly make it convenient to attend the meeting.

Agenda items:

1. Discussion on preparation of the Concept Paper on Roles and Structures of Schools, Departments, Centres (Services or Research), Cells, Facilities etc.
2. Any other matter.

(Dr. Subhankar Roy-Barman)
Convenor of the Committee

Copy to:

1. Prof. S. Ghosh, Dean (Academic), Chairman
2. Prof. Soumya Bhattacharya, Dept. of C. E., Member
3. Prof. A. N. Mullick, Dept. of M.E., Member
4. Dr. Seema Sarkar (Mondal), Dept. of Maths, Member
5. Dr. Suchismita Roy, Dept. of C.S., Member
6. Dr. Koushik Mandal, Dept. of M.S., Member

Page-70

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR
M.G. AVENUE, DURGAPUR – 713 209

Ref No. NITD/Regis/OR/852

Date: 23rd June, 2017

OFFICE ORDER

In accordance with the approval from the competent authority to the resolution taken in a meeting dated 23/06/2017, a committee has been formed to negotiate with **Blue Star** for finalising the terms and conditions and contract value of AMC of the Air-conditioning system that was installed by **Blue-Star** at S.N. Ray Memorial Building under the renovation work done by NBCC.

1.	Dr. Kamal Bhattacharya, Dean (P & D)	Chairman
2.	Dr. G. Sanyal, Head (CSE)	Member
3.	Dr. S. Banerjee, Head (EE)	Member
4.	Dr. A. N. Mallick, Head (ME)	Member
5.	Dr. R. Ghatak, Head (ECE)	Member
6.	Dr. N. K. Roy, Professor (EE)	Member
7.	Mr. T. Haldar, EE-NITDGP	Member, convener.

The committee is empowered to induct external member(s) from CMERI for the expertise if needed.

The committee is requested to submit its recommendations to the competent authority by 1st week of July 2017.

23/6/17

Registrar

1. Director.
2. Registrar
3. All concerned members.
4. File copy.

Page - 71

CONFIDENTIAL

No.F.10 (51)-RECT/TPSC/2017
TRIPURA PUBLIC SERVICE COMMISSION

Agartala, 20th October, 2017

To
Dr. Amarnath Mullick, Professor,
Department of Mechanical Engineering, NIT Durgapur,
Dist : Bardhaman, West Bengal,
PIN- 713209
Ph. No.09830385342,

Subject: - Appointment of Advisor.

Sir,

I am directed to inform that the Tripura Public Service Commission will hold Interview in the Commission's office at Agartala for recruitment to the post of Lecturer, Diploma Level Technical Institutions in Tripura, Group-A Gazetted under the Education(Higher) Department, Government of Tripura as per programme below.

Sl. No.	Name of Subject/Discipline.	Date	Session		Total
			Forenoon (10.30 AM)	Afternoon (2.30 PM)	
1.	Mechanical Engineering	14-11-2017	08	07	15
		15-11-2017	08	07	15
2.	Automobile Engineering	15-11-2017	-----	01	01

The Commission has decided to invite you as an Advisor for the Interview.
In this connection, I am to inform you that the Expert- Adviser being invited from outside Tripura will avail the facilities as mentioned below :-

a) Arrangement for Reception & See- off at Agartala airport with provision of one vehicle for his movement at Agartala.

b) Reimbursement to and fro travelling expenses.
- Residence to airport Taxi/ Train fare (AC II tier)
- Cost of air ticket (actual paid)
- Air incidental charge.

c) Honorarium @Rs.1500/- (Rupees fifteen hundred) only per day of Personality Test/Interview.

Contd. Next Page

EXP-3

Page - 72

NETAJI SUBHAS OPEN UNIVERSITY

H.O.: DD-26, Salt Lake, Sector -I, Kolkata - 700 064.
Phone : 4066 3220, TELEFAX : 4066 3225
Website : www.wbnsou.ac.in

Memo No. Reg/734 Date-7.06.2017

To,
Prof. (Dr.) Amar Nath Mullick,
HOD,
Deptt of Mechanical Engineering,
NIT, Durgapur

Subject:- Member of Standing Committee for Selection in Officer Posts(s)

Sir,

I am directed to bring to your kind notice that you have been designated as member of the Standing Committee by the Hon'ble Vice Chancellor for Selection of **System Analyst,** in Netaji Subhas Open University.

Now, may I request you to kindly spare some of your valuable time for the university in connection with the aforesaid selection.

The interview in this regard would be held on 23rd & 24th June, 2017 from 11A.M onwards at the Headquarters of Netaji Subhas Open University at DD-26, (3rd floor) Salt Lake, Sector-1, Kolkata-700 064. We shall be obliged if you kindly make yourself available on this date to be present at the university for the selection process. For any further communication the undersigned maybe contacted.

With best Regards.

Registrar

Mail Id-"registrar@wbnsou.ac.in"

Page-73

DURGAPUR INSTITUTE OF ADVANCED TECHNOLOGY & MANAGEMENT

Registered & Administrative Office : G. T. Road, Rajbandh, Durgapur - 713212, Dist. : Burdwan, (W.B.)
Ph. : (0343) 2520712, 2520713
Fax : (0343) 2520881
City Office : 21, Princep Street, 1st Floor
Kolkata - 700072
Tele Fax : (033) 22258326, 22128062

No. DIATM/DIR/44/26
Date : 27.03.2017.

To.
Prof. A. N. Mullick,
Professor & HOD of Mechanical Engg. Dept., of
National Institute of Technology,
Gandhi More,
Durgapur.

Dear Sir,

Sub : **Request to attend All India Seminar organized by our Mechanical Engineering Dept. as Invited Speaker on 04.04.2017.**

We feel happy to inform you that our Mechanical Engineering Department will organize 3 days' all India seminar from 03.04.2017 to 05.04.2017 on "**Enabling Sustainable Development in Mechanical Engineering in the context of Make-in-India**" in association with The Institution of Engineers(India), Durgapur Chapter. We shall feel immensely proud to have you as our Invited Speaker on 04.04.2017.

You are also requested to give us a message to incorporate in our Souvenir.

Your august presence will definitely grace this auspicious occasion and make the Seminar a grand success.

Thanking you,

Yours faithfully,

27. 3. 2017

(Prof. (Dr) P. K. Sinha)
Principal, DIATM & Chairman(Organizing Committee)

Principal
Durgapur Institute of Advanced
Te...

Page -74

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR
MAHATMA GANDHI AVENUE
DURGAPUR-713 209(WEST BENGAL), INDIA

Office: (0343) 2545290
Fax: (0343)2547375/ 2546406
E-mail: registrar@admin.nitdgp.ac.in
Website: www.nitdgp.ac.in

NITD/Estab_Sec/4P-845,143,917

Date- 03.04.2017

OFFICE ORDER

As approved by the Competent Authority following Faculty Members have been permitted to attend the following programme as mentioned below against their names –

Sl. No.	Name, Emp. ID, Designation and Dept. of the Faculty Member	Purpose	Place	Duration	Nature of Leave	Financial support
1	**Prof. N. K. Roy, Professor, Dept. of Electrical Engg., ID NO. 4P-845**	To act as an Assessor in the Performance Management of Faculty members	Dept. of Electrical Engg. Asansol Engineering College, Asansol	27th March,2017	Not Applied	Not Required
2	**Prof. P. Kumbhakar, Professor, Dept. of Physics, ID No. 4P-143**	To act as an Assessor in the Performance Management of Faculty members	Dept. of Physics, Asansol Engineering College, Asansol	30th March,2017	On duty leave	Not Applied
3	**Prof. A. N. Mullick Professor, Dept. of Mechanical Engg., ID No. 4P-917**	To act as an Assessor in the Performance Management of Faculty members	Dept. of Mechanical Engg. Asansol Engineering College, Asansol	29th to 30th March,2017	Not Applied	Not Applied

Registrar 4/4/17

Copy to:

1. Director
2. Registrar
3. Deputy Registrar (Finance & Accounts)
4. Assistant Registrar (Internal Audit)
5. Head, of the Department (i) E.E (i) Physics (ii) M.E.
6. Prof. N. K. Roy, Professor, Dept. of M.E.
7. Prof. P. Kumbhakar, Professor, Dept. of Physics
8. Prof. A. N. Mullick, Professor, Dept. of M.E.
9. Dealing Assistant (Leave)
10. Director Secretariat & Registrar Secretariat
11. File copy

Deputy Registrar (Establishment)

Page - 75

NATIONAL INSTITUTE OF TECHNOLOGY DURGAPUR

MAHATMA GANDHI AVENUE
DURGAPUR - 713209 (WEST BENGAL), INDIA

Office: (0343) 2545290
Fax: (0343)2547375/ 2546406
E-mail: registrar@admin.nitdgp.ac.in
Website: www.nitdgp.ac.in

NITD/Per_Sec/4P-917

Date- 14.03.2016

OFFICE ORDER

As approved by the Competent Authority, Professor Amar Nath Mullick, Department of Mechanical Engineering has been permitted to present and evaluate the viva-voce examination for awarding Ph.D (Engg.) of Sri Arindam Bit at Jadavpur University on 10th March 2016.

No financial support will be provided from the Institute for this purpose.

Registrar 14/3/16

Copy to,

1) Director
2) Registrar
4) Deputy Registrar(F&A)
5) Asstt. Registrar(Internal Audit)
6) Head, Department of Mechanical Engg.
7) Prof. Amar Nath Mullick, Department of Mechanical Engg.
8) File copy

Deputy Registrar Establishment

(Annexure - B page-04)

Office : (0343)2546397
Fax: (0343) 2547375 / 2546753
E-mail: registrar @admin.nitdgp.ac.in
Website www.nitdgp.ac.in

NATIONAL INSTITUTE OF TECHNOLOGY, DURGAPUR

MAHATMA GANDHI AVENUE, DURGAPUR-713209, West Bengal, INDIA

No. NITD/Estt/ HAG-faculty/2018 Dated:- April 24th, 2018

OFFICE ORDER

Based on the recommendation of Anomaly Committee constituted for the purpose and subsequent approval of the Board of Governors in its 49th Meeting held on 9th March, 2018, the AGP (Academic Grade Pay) of the following faculty members of the Institute are hereby mapped from Rs.10,000/ to Rs. 10,500/- as mentioned against each within the purview of prescribed norms and instructions of MHRD, Govt. of India with effect from the date of adoption by 49th BoG Meeting i.e. 09.03.2018.

Sl. No.	Name of Faculty	Designation	Department	Existing AGP	Mapped to AGP
1	Prof. Sudip Chattopadhyay	Professor	Biotechnology	₹10,000/-	₹.10,500/-
2	Prof. Apurba Dey	Professor	Biotechnology	₹10,000/-	₹.10,500/-
3	Prof. Anup Kumar Sadhukhan	Professor	Chemical Engg.	₹10,000/-	₹.10,500/-
4	Prof. Parimal Pal	Professor	Chemical Engg.	₹10,000/-	₹.10,500/-
5	Prof. Parthapratim Gupta	Professor	Chemical Engg.	₹10,000/-	₹.10,500/-
6	Prof. Tamal Mandal	Professor	Chemical Engg.	₹10,000/-	₹.10,500/-
7	Prof. K. C. Ghanta	Professor	Chemical Engg.	₹10,000/-	₹.10,500/-
8	Prof. B. P. Mukhopadhyay	Professor	Chemistry	₹10,000/-	₹.10,500/-
9	Dr. Vijay Kumar Dwivedi	Professor	Civil Engg.	₹10,000/-	₹.10,500/-
10	Dr. Showmen Saha	Professor	Civil Engg.	₹10,000/-	₹.10,500/-
11	Dr. S. Bhattacharyya	Professor	Civil Engg.	₹10,000/-	₹.10,500/-
12	Dr. Dilip Kr. Singha Roy	Professor	Civil Engg.	₹10,000/-	₹.10,500/-
13	Dr. Amlan Das	Professor	Civil Engg.	₹10,000/-	₹.10,500/-
14	Prof. Goutam Sanyal	Professor	CSE	₹10,000/-	₹.10,500/-
15	Prof. Aniruddha Gangopadhyay	Professor	EES	₹10,000/-	₹.10,500/-
16	Prof. S. Ghosh	Professor	Electrical Engg.	₹10,000/-	₹.10,500/-
17	Prof. N. K. Roy	Professor	Electrical Engg.	₹10,000/-	₹.10,500/-
18	Prof. Subrata Banerjee	Professor	Electrical Engg.	₹10,000/-	₹.10,500/-
19	Dr. Siddhartha Sankar Thakur	Professor	Electrical Engg.	₹10,000/-	₹.10,500/-
20	Prof. Sakti Prasad Ghosal	Professor	Electrical Engg.	₹10,000/-	₹.10,500/-
21	Prof. A. K. Bhattacharya	Professor	ECE	₹10,000/-	₹.10,500/-
22	Prof. Sumit Kundu	Professor	ECE	₹10,000/-	₹.10,500/-
23	Prof. B. Maji	Professor	ECE	₹10,000/-	₹.10,500/-
24	Prof. Rowdra Ghatak	Professor	ECE	₹10,000/-	₹.10,500/-
25	Dr. Goutam Kr. Mahanti	Professor	ECE	₹10,000/-	₹.10,500/-
26	Prof. P. P. Sengupta	Professor	HSS	₹10,000/-	₹.10,500/-

25/04/18

27	Prof. Mousumi Roy	Professor	Management Studies	₹10,000/-	₹.10,500/
28	Dr. Kajla Basu	Professor	Mathematics	₹10,000/-	..10,500/-
29	Prof. Nilotpal Banerjee	Professor	Mechanical Engg.	₹10,000/-	₹.10,500/-
30	Dr. Indrajit Basak	Professor	Mechanical Engg.	₹10,000/-	₹.10,500/-
31	Prof. A. K. Saha	Professor	Mechanical Engg.	₹10,000/-	₹.10,500/-
32	Prof. Amar Nath Mullick	Professor	Mechanical Engg.	₹10,000/-	₹.10,500/-
33	Prof. Manik Ch. Majumder	Professor	Mechanical Engg.	₹10,000/-	₹.10,500/-
34	Prof. K. S. Ghosh	Professor	MME	₹10,000/-	₹.10,500/-
35	Prof. A. K. Meikap	Professor	Physics	₹10,000/-	₹.10,500/-
36	Prof. P. Kumbhakar	Professor	Physics	₹10,000/-	₹.10,500/-

25/04/18

Registrar (I/C)

Copy forwarded for information to:-

1. Director
2. All Deans and HOD
3. Concerned Faculty Members
4. Dy. Reg. (F&A)
5. Asstt. Reg. (I.A.)
6. Director Secretariate
7. Registrar Secretariate
8. File Copy

25/04/18

Dy. Reg. (Estt.)

J.K. R