KG REDDY
College of Engineering
& Technology

Certificate Course in Mechanical Engineering with Specialization "HYPERMESH"

Held On

06th August to 10th August 2018

Department of Mechanical Engineering,

KG Reddy College of Engineering & Technology

Chilkur (Village), Moinabad (Mandal), Hyderabad RR Dist-501504

Course coordinator

Principal

Principal
KG Reddy College of Engineering & Technology
Chilkur (V), Moinabad (M).
R.R.Dist., Telangana.

KG REDDY
College of Engineering
& Technology

SUMMARY REPORT ON HYPERMESH

About the Course

The certificate course on Hypermesh is concluded its work successfully by department of Mechanical Engineering (ME) in KG Reddy college of Engineering and technology (KGRCET), Hyderabad, Telangana. This course is a forum to bring together students to discuss innovative ideas and diverse topics of this course on next generation of information technologies. Department has taken a new step for students to improve the quality of study through this course and become most wide scale , extensive, spectacular event in ME. The course was held in two locations of the department (a) Department E-learning room and (b) Department class.

HyperMesh is the market-leading, multi-disciplinary finite element pre-processor which manages the generation of the largest, most complex models, starting with the import of a CAD geometry to exporting a ready-to-run solver file. Generating high-quality mesh quickly is just one of HyperMesh'score competencies.

This course is absolutely practical oriented course which is helped to student for making their carrier through database in any industry. The students of 3rd year 1st semester have been benefited in many ways from this course. More than 80 students have joined in this course as their own interest and completed this course. The trainer taught to students very nice with real time example and sharing his knowledge to develop technical skill in industry.

Objectives of the course

Design optimization seeks a design that minimizes the objective function while satisfying design constraints. Real-world design problems are usually characterized by the presence of many conflicting objectives. Therefore, it is natural to look at the engineering design problems as multi-objective optimization problems. For example, we may want to maximize range and payload mass while trying to minimize manufacturing costs for an airplane design.

Scope of the Course

Creo 2.0 has got vast scope. Many companies are now switching to Creo because of its Parametric, Easy UI, More enhanced tools, CAE features, In built simulate and Etc. features(like sheetmetal, Free Style). So get itself mastered in any one module of this software and get ready to rule in market. Also TVS, VECV, Eicher Motors works solely on Creo.

OUTPUT:

This course was not only shared the knowledge among students but also tied up with expert for upcoming course.

- Capable of producing meshes that are high quality made with no complexity.
- With comprehensive state of art technologies, evaluation and manufacturing, it reduces the development cost to manifolds.
- Virtual testing and superb designing makes it safe and secure with minimized risk factor.
- The test results obtained through virtual testing are valuable enough to facilitate collaborations and change decisions for further processes.

- Attain accurate and high quality meshes in fractions of seconds, since it supports batch meshing which makes it possible to develop several meshes at once.
- Flexible enough to accelerate your business abilities as per the changing conditions.
- Get a full access and level of control with the meshing algorithms.
- Comes with a built in automated supports which adds to its flexibility.
- Precisely design and edit meshes within a small time frame to suit the user's needs and match up or exceed their expectations.
- It provides several capabilities and tools that help you with sustainability issues.

Summary of Participants

(a) Number of students attended this course: 53
(b) Number of student attend the exam: 49
(c) Number of certificate issued: 49

Day-1
06/08/18

Time: 09:00 AM to 10:00 AM

Inauguration of certificate course

The first day of certificate course started with welcoming and opening ceremony at the KGRCET conference Hall. The following dignitaries were representatives of the certificate course who were addressed and pointed out the importance on course with short welcoming speeches.

Welcome addressed by Dr. P. Pravuraj, HOD, H&S, KGRCET
About the certificate course by Principal Dr. R. S. Jahagirdar, KGRCET.
Importance of this course by expert trainer Mr.V. Karthik, Hyderabad
Interaction with 3rd year 1st semester students

Time: 10:00 AM to 04:15 PM

Introduction to HyperMesh

The HyperMesh window has four main
menu areas: graphics, the header bar, the main menu, and the permanent menu. The header bar divides the screen into two areas. The graphics area of the screen is above the bar and the menu area is below the bar. The menu area is further divided into the main menu and the permanent menu. A secondary menu can be accessed by using keyboard keys. The secondary menu allows you to use panels that add information necessary to complete the currently active menu panel.

The Main Menu

Due to the addition of functions and options, the main menu is expanded from five to seven pages. The page names, *Geom*, *1-D*, *2-D*, *3-D*, *BCs*, *Tool*, and *Post*, are given by the functionality of the page panels. Some of the panels on each page contain functionality that applies to multiple pages. These panels appear on all pages applicable to that panel's functionality.

Day-2
07/08/18

The Modeling Subpanel
This panel contains most of the entities previously located in the *options* panel as well as new options including the *cleanup* and *geometry tolerance* fields, *fixed points*, and *coincident node picking*.

Used when finding line intersections and determining duplicate nodes. The node tolerance also affects the generation of elements in the automesher. When quads are created and the side of a quad is less than the node tolerance, HyperMesh tries to create a tria element instead of a quad. If it create a model with characteristic dimensions less than the node tolerance, reduce the default node tolerance. The geometry tolerance specifies the mathematical accuracy of lines and surfaces in the model. Lines and surfaces are guaranteed geometrically accurate to within the distance specified by *geom tol*. The geometry tolerance influences the speed of file i/o and geometric operations in HyperMesh. Very small geometry tolerances can increase file read times and increase the length of time required to perform geometric operations.

The graphics Subpanel
This is the new location for the *graphics* panel. New options are included. In addition, the *hidden lines* field and *resize* box were moved here from the original *option* panel. Lighting tools for shaded elements are also included in this subpanel. The performance graphic engine now contains bitmap animation tools, view acceleration tools, and the result color type options.

Day-3
08/08/18

Cleanup Surfaces and Add Fixed Points
This exercise demonstrates the usage of the *cleanup*, *add points*, and *remove points* subpanels while automeshing a model. The addition of these functions in *automesh* panel allows it to clean up surfaces without leaving the *automesh* panel. The mesh is done by setting the *reset meshing prarmeters to:* to *element size* with specified element type.

Using Springs
The *springs* panel allows you to create spring elements. A spring element is an element created in a space between two nodes of a model where a spring connection is desired. Spring elements store a property and a degree of freedom (dof). Spring elements are displayed as a line between two nodes with the letter K written at the centroid of the element. Springs can translate to CELAS2 in NASTRAN or *spring in ABAQUS. Springs require a property definition.

Calculating Beam Cross Section - HM-220

The *beam cross section* panel calculates the cross sectional plane for a beam element and creates a beam element. It allows it to create a summary file with the results of the calculations performed. The *beam cross section* panel has two subpanels, *offset lines* and *pick geom.* After you use the *pick geom.* subpanel to calculate the cross-sectional plane, a secondary panel is displayed. The secondary panel allows you to apply the results to the previously created HyperMesh property and create the beam element and summary file. The *offset lines* subpanel allows you to create welds on the cross section of the element. The *beam cross-section post-processing* subpanel allows you to apply the results to the previously created HyperMesh property solver and create the beam element and a summary file. The *Center of Gravity (purple +)* and *Shear Center (yellow +)* are displayed in the graphics area.

Day-4
09/08/18

Combining Shell Elements using the edit element panel

The *combine* subpanel on the *edit element* panel allows you to combine an arbitrary number of shell elements simultaneously or a set number of shell elements automatically. Both methods are controlled by the *tolerance* = and *angle* = functions. When elements are being combined, HyperMesh requires the nodes attached to the elements to be planar within a user-specified tolerance. The tolerance may be changed with the menu item *tolerance* =.

When elements are being combined, HyperMesh performs node condensation on mid-side nodes. Nodes are considered to be mid-side nodes if the angle between any three nodes in the set of nodes being condensed is greater than a user-specified angle.

Analysis of a Plate with a Hole

To create finite elements on a given geometry of a plate with a hole, apply boundary conditions, and perform a finite element analysis of the problem. Post-processing tools will be used in HyperMesh to determine deformation and stress characteristics of the loaded plate.

Defining Contact Surfaces and Surface Interactions

HyperMesh supports definition of the *SURFACE DEFINITION card using sets, components, or individual element IDs with faces. In this example, you use individual element faces to define the slave contact surface and sets to define the master contact surface. This model is made from solid elements, so you must first skin the surface with face elements, and then use those face elements to define the contact surface.

Defining Geometry Properties for 3-D Solid Elements

HyperMesh supports geometry properties for shells, solids and beams from the component collector. In this example, create the GEOMETRY model definition card and tie them to the already existing component collectors.

Day-5
10/08/18

Creating Loads and Boundary Conditions

In HyperMesh, every load collector can be used to define a set of loads and boundary conditions. The load collectors can be added to a loadstep, which defines either a model

definition data card block (before END OPTION card), or a history definition data card block (between two CONTINUE cards) in the MARC input deck. In this example, define three HISTORY blocks with different load magnitudes.

Checking for Penetration

The *penetration* panel is used to check contact interfaces for nodal penetrations. It allows you to determine how much penetration is occurring and to correct the penetration by moving any penetrating nodes. The *penetration* panel supports all of the solver interfaces that contain card images and interface elements. Before you use the *penetration* panel, element thicknesses and contact interfaces must be defined for the current template loaded in the *global* panel. For more information on defining thickness on collector cards, refer to the *collectors* panel in the *Panels* section of the online help. For more information about creating contact interfaces, see the *interfaces* panel in the on-line help.

KG REDDY
College of Engineering
& Technology

DEPARTMENT OF MECHANICAL ENGINEERING

Ref No: KGRCET/ME/2018-19/08 Date: 01/08/2018

CIRCULAR

All the III-Year I-semester B.Tech Mechanical Engineering students are here by informed to enroll for the certification course on "HYPERMESH", which is offering by KG Reddy college of Engineering and Technology from 06/08/2018 to 10/08/2018. Interested students are instructed to complete their registration before 04/08/2018.

HOD

Principal

Principal
KG Reddy College of Engineering & Technology
Chilkur (V), Moinabad (M).
R.R.Dist., Telangana.

Copy to:

1. Exam Section
2. Notice Boards
3. Library

KG REDDY
College of Engineering
& Technology

KG REDDY COLLEGE OF ENGINEERING & TECHNOLOGY
Chilkur (Vill) Moinabad (Mdl) R R Dist

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE COURSE ON HYPERMESH
SCHUDULE

Day	Date	Timings	Topic name
1	06/08/18	09:00 to 11:00	**Introduction to HyperMesh** Introduction to CAD & CAE, Application of CAE Software, Introduction to FEM, Introduction to HyperMesh, Menu bars, tool bars.
		11:10 to 01:00	Create Node , Node edit, Temp Nodes
		01:45 to 02:50	Lines, Line edit, Length, Delete Mas
		02:50 to 04:15	Surface & Surface edit, Normals, Translate, Rotate
2	07/08/18	09:00 to 11:00	**2D Meshing**
		11:10 to 01:00	Introduction to Meshing, Types of collectors, Auto-Meshing (Size & Biasing)
		01:45 to 02:50	Mesh Connectivity, Replace & Remeshing, Current and surface comps
		02:50 to 04:15	Density and mesh style
3	08/08/18	09:00 to 11:00	**Mesh Edit** Edit Elements, Find entities, Organize Entities
		11:10 to 01:00	Project, Position, Normals, Scale
		01:45 to 02:50	Color, Rename, Detach Order Change
		02:50 to 04:15	Number and Mass Calculation
4	09/08/18	09:00 to 11:00	**3D Meshin**
		11:10 to 01:00	3D Hex Mesh
		01:45 to 02:50	3D Solid Mesh
		02:50 to 04:15	3D Tetra Meshing
5	10/08/18	09:00 to 11:00	**Linear Meshing**
		11:10 to 01:00	Introduction to Analysis, Create collectors
		01:45 to 02:50	Card edit, loads Constraints
		02:50 to 04:15	Loadsteps

Signature of Coordinator

KG REDDY College of Engineering & Technology

KG REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

Chilkur (Vill) Moinabad (Mdl) R R Dist

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE COURSE ON HYPERMESH

ATTENDANCE SHEET

S.No	Roll No.	Student name	Date									
			06.08.2018		07.08.2018		08.08.2018		09.08.2018		10.08.2018	
			FN	AN	FN	AN	FN	AN	FN	AN	FN	AN
1	14QM1A0384	Y Shashank Reddy										
2	14QM1A0386	Mateti Akshay Kumar										
3	15QM1A0343	Mohammed Umer										
4	16QM1A0303	Amirgudem Mahesh										
5	16QM1A0304	Anupati Sai Ram Reddy										
6	16QM1A0305	Atheli Saketh Reddy										
7	16QM1A0306	Atla Goutham Reddy										
8	16QM1A0311	Chenna Saiteja										
9	16QM1A0312	Chevula Naveen										
10	16QM1A0314	Danda Ravi Teja										
11	16QM1A0317	Dumpala Chakra Babu										
12	16QM1A0318	G Sai Prashanth Reddy										
13	16QM1A0320	Gaddam Vishal										
14	16QM1A0321	Gajulagudem Arunkumar										
15	16QM1A0326	Gongupalli Rajinikanth										
16	16QM1A0330	Jaithwala Karthik										
17	16QM1A0331	Kalakuntla Manoj Kumar										
18	16QM1A0335	Keshaboina Rahul Yadav										
19	16QM1A0336	Khan Adil A Rahman										
20	16QM1A0338	Kosari Manoj										
21	16QM1A0339	Kunapalli Preeteesh Raja										
22	16QM1A0342	Landeri Venkata Saikrishna										
23	16QM1A0344	Maniyar Safwan Sayeed										

KG REDDY
College of Engineering
& Technology

24	16QM1A0347	Mohammad Jameel									
25	16QM1A0348	Mohammed Talha Hussain Khan									
26	16QM1A0349	Mulli Ashok									
27	16QM1A0350	Oruganti Prudviraj									
28	16QM1A0352	Pampana Mukesh									
29	16QM1A0356	Sabnakar Sachin									
30	16QM1A0362	Yellu Pranath Reddy									
31	16QM5A0307	Chintakuntala Jaisurya									
32	16UR1AO302	K Rajendra									
33	17QM5A0301	Arigela Ranjith									
34	17QM5A0302	Barmali Karthik									
35	17QM5A0303	Bunne Naveen									
36	17QM5A0304	Chandrakani Mahesh									
37	17QM5A0306	Gajjela Vinod									
38	17QM5A0307	Golkonda Raj Kumar									
39	17QM5A0308	Javoji Hari Krishna									
40	17QM5A0309	K Rajashekhar									
41	17QM5A0310	Kanakala Arun Kumar									
42	17QM5A0311	Kokkula Uday									
43	17QM5A0312	Maloth Santhoshkumar									
44	17QM5A0313	Mangali Ganesh									
45	17QM5A0314	Md Saqlain									
46	17QM5A0315	Mula Ramakanth									
47	17QM5A0316	Myagani Krishna Kanth									
48	17QM5A0318	Nalapuram Kumar									
49	17QM5A0320	Panthangi Nikhil									
50	17QM5A0322	Rayudu Nagababu									
51	17QM5A0323	Sara Vishnuvardhan									
52	17QM5A0324	Shaik Luqman									
53	17QM5A0326	Vulli Shiva Krishna									

Signature of the Coordinator

KG REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

Chilkur (Vill) Moinabad (Mdl) R R Dist

B.TECH III Year I SEM, Aug - 2018

HYPERMESH

OBJECTIVE EXAM

9/10

NAME UMAR **HALL TICKET NO** | 1 | 4 | Q | M | 1 | A | 0 | 3 | 8 | 6 |

Answer all the questions. All questions carry equal marks. Time: 30min. 10 marks.

I choose correct alternative:

1. In Finite element method formulation of problem results in a system of
   a) Algebraic equations b) Logical equations
   c) Arithmetic equations c) Flow equations — a ✓
2. Finite element method is also called
   a) Infinite element analysis b) Frequency element analysis
   c) Finite element analysis c) Partial element analysis — c ✓
3. To solve FEM problem, it subdivides a large problem into smaller, simpler parts that are called
   a) Finite elements b) Infinite elements
   c) Dynamic elements d) Static elements — a ✓
4. The finite element method is mostly used in the field of
   a) Structural mechanics b) Classical mechanics
   c) Applied mechanics d) Engineering mechanics — d ✓
5. FEM can't produce exact result as those of.....methods
   a) Analytical b) Logical
   c) Theoretical d) all the above — d ✓
6. FEM also operates the parameters like
   a) Heat transfer b) Temperature
   c) Both A&B d) None — d ✓
7. Range of Poisson's ratio for metals is
   a) 0.25-0.33 b) 0.25-0.50
   c) 0.22-0.25 d) 0.22-0.45 — b ✓
8. A bar is modelled as 1-D element only if its
   a) Area of cross section is small b) M.I is small
   c) Length is very large compared to cross sectional area d) all of the above — d ✓
9. Stiffness matrix contains information on
   a) Geometry b) Material properties
   c) Both d) None — d ✓
10. A 1-D structural element is a
    a) Truss element b) Beam element
    c) Bar element d) all of them — d ✓

KG REDDY

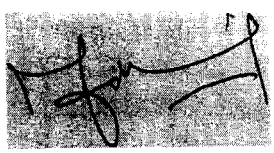
CERTIFICATE

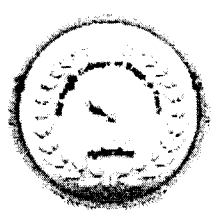
Name: YEDULA SHASHANK REDDY

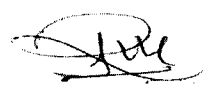
Registration No: 14QM1A0384

has successfully completed the prescribed requirements for the award of certificate course on " **HYPERMESH** " conducted by Mechanical Engineering held in month of February from 06/08/18 to 10/08/18 in the academic year 2018-2019.

Date: 10/08/18







Course Coordinator

Principal

KG REDDY

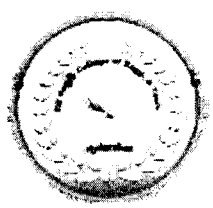
CERTIFICATE

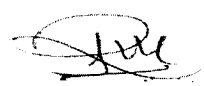
Name: Mateti Akshay Kumar

Registration No: 14QM1A0386

has successfully completed the prescribed requirements for the award of certificate course on " **HYPERMESH** " conducted by Mechanical Engineering held in month of February from 06/08/18 to 10/08/18 in the academic year 2018-2019.

Date: 10/08/18





Course Coordinator

Principal