HyperMesh for FEA Plastic & Sheet Metal Applications

SKILL⬭LYNC

Who are we?

Skill-Lync is an engineering e-learning platform
based in Chennai. The brainchild of two engineers, we are
at the forefront of re-shaping engineering education in India.
In the winter of 2019, we became the first start-up from Chennai to
be funded by Y-Combinator.

Today, close to 8000+ students from over 58 countries study at
Skill-Lync. Over three years we have helped 280 engineers achieve
their dream careers.

58+	8000+	300+	30000+
Countries	Students	Placements	Projects

About the program

Within Computer Aided Engineering (CAE) workflow there are three phases - pre-processing, solving, and post-processing. This course focuses on the pre-processing phase. If you are someone who aspires to work in the CAE domain this course is for you. As a fresher completing this course opens up opportunities for you to be recruited as a CAE modeler/mesher in any industry. You will learn the geometry cleanup, shell meshing, solid meshing, 1D meshing, connectors and morphing. Moreover, you will also get an overview of the HyperMesh tool which will come in handy in the FEA, MBD, and CFD domains as well.

By the end of this course, you will be skilled in 1D, 2D (shell meshing), 3D (solid meshing) and in connectors. There are several complex models in the course work that will help you in practicing the techniques taught in this course.

Reasons to select this course

- Designed by Skill-Lync with inputs from academicians and industry experts to ensure that students are trained in the skills expected in the industry.

- The finite element method is a widely used method for numerically solving differential equations arising in engineering and mathematical modeling.

- This course is the gateway program to the world of FEA meshing techniques and cleaning geometry before sending it for analysis.

- Completing a Skill-Lync's course on HyperMesh for FEA Plastic and Sheet Metal Applications is among the few tools in your resume that can instantly boost your chances of getting employed.

Modules

Basics of FEA

In this module, you will study the fundamentals of Finite Element Analysis (FEA). FEA is a generic technique used to solve boundary value problems. FEA finds a lot of applications in structural analysis. In this module, we will discuss the math behind FEA, along with the following topics

- 5Ws of Finite Element Analysis
- Types of engineering problem-solving methods
- Types of CAE process
- Model setup

Introduction to HyperMesh GUI

In this module, you will study the meshing software - HyperMesh. HyperMesh is a highly capable simulation software and in this module you will learn the various tools and features that HyperMesh has to offer. In this module, you will learn of the following topics

- Geometric tool and tools menu introduction
- Basic geometric tools
- Geometry cleanups (includes auto cleanup)
- Midsurface creation

Key Geometric Tools

While dealing with complex 3D components, it is important to remove small features and holes that are inevitable when designing a CAD model. You will be introduced to the batch mesher macro that will be instrumental in accomplishing this. In this module, you will study the following topics

- Criteria files for geometry cleanup
- MNode's creation & node edit
- Point edit
- Edge edit

1D Meshing

In this module, you will study the meshing software - HyperMesh. HyperMesh is a highly capable simulation software and in this module you will learn the various tools and features that HyperMesh has to offer. In this module, you will learn of the following topics

- In this module, you will learn the following
- 5Masses, Bar, Rod, Rigid, RBE3, Springs, Gaps
- Connectors, HYPER BEAM, Linemesh, Linear 1D
- Edit, Element, Split, Replace, Detach
- Order change, configuration edit, element type

Geometry Cleanup and Midsurface

In this module you will learn to perform surface clean-up on a given geometry and to extract the mid-surface using an in-built tool - Auto-mid-surface. You will also learn how to clean up the surface manually because relying only on the Auto-mid-surface option may give rise to a distorted surface while dealing with a complex geometry. Here is a list of topics that you will learn in this module

- Surface clean up
- Midsurface generation
- Handling distorted surfaces

2D Meshing Introduction and Examples

In this module, you will learn the basics of 2D meshing, how to mesh a surface, how to re-mesh a meshed surface using Elements, and how to attain mesh flow in the geometry. You will also learn how to handle the Tria elements in the model.

- 2D meshing introduction
- 2D basic tools
- Mesh flow
- Handling Tria Elements

Quality Criteria

In this module, you learn what a quality check is and understand the different quality-checks that hyperMesh offers. You will also learn how to fix areas that failed the quality check.

- Here is a list of key concepts that you will learn in this module.
- 1D Element quality check
- 2D Element quality check
- Understanding the quality parameters for 2D meshing

Advance 2D Meshing

In this module you will learn how to do manual mid surfacing when the auto-mid surfacing option fails. You will also learn how to handle geometry if they are symmetrical, normals check in the model and its significance, and free edges along with their significance

- The following topics are covered in this
- Manual mid surface
- Handling symmetric geometry
- Free edge checks
- Normals check

3D Meshing

In this module, you will learn the different tools and techniques to generate a hexamesh for a given geometry. You will also learn a variety of techniques to generate tetra meshes. You will also understand the quality-checks for tetra elements and how to rectify areas that have failed these checks. The following topics are covered in this module.

- Hexa meshing
- Tetra meshing
- Different methods of tetra mesh generation.
- Quality check 3D element

Connectors

Sometimes, meshes are made with both 2D and 3D meshes that are interspersed. Connectors help the computer understand that the interface is between two different kinds of meshes. Connectors are time-saving features that are a must in large models. In addition to this, connectors offer a way to let the computer know that two components have been seam welded.

- Seam weld
- Spot weld
- Bolt connection
- Spring element
- Adhesive modelling - HEXA

Batch Meshing

In this module, you will understand the uses of batch meshing and its applications in industries. You learn about the parameter file and criteria file, how they can be optimized for the requirement. Here is a list of key concepts that you will learn in this module.

- Understand quality parameter and criteria files
- Optimizing these files according to the requirement

Morphing Techniques and Macro Generation

In this module, you will learn about morphing which can be done using constraints, volumes, and geometry. Also you will be working with manipulation of TCL commands, running a TCL command using the inbuilt run options.

- Morph constraints
- Morph volumes
- Map to geometry
- Basic macro generation using TCL commands

Industry Specific Projects

Meshing of Inner Panel

Here you will be provided with the door's plastic inner panel. The inner panel must be cleaned, should create mid-surface, and mesh the whole component for the given quality criteria.

Meshing of Car Spare Tyre Plastic Component

Here you will be provided with the car's spare tyres under panel. The under panel must be cleaned, should create a mid surface and mesh the whole component for the given quality criteria.

Should you take this course?

- For students who are pursuing a BE/B Tech in Aeronautical, Automotive or Mechanical Engineering.
- Freshers who passed out of the college recently.
- Professionals who are working in the quality, production, or manufacturing domain, looking to climb up the ladder in their career.
- If you are looking for a deeper understanding into pre-processing a sheet metal or plastic component.
  To gain confidence to clean and mesh any kind of model by the end of the course.

Software covered

Altair **HyperMesh**™

Altair's HyperMesh is the market-leading, multi-disciplinary finite element preprocessor 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.

Basic

2 Months Access

₹7000

Enroll Now

Per month for 3 months

- Access Duration : 2 months
- Mode of Delivery : Online
- Project Portfolio : Available
- Certification : Available
- Email Support : Available
- Whatsapp Support : Available

Pro

4 Months Access

₹10000

Enroll Now

Per month for 3 months

- Access Duration : 2 months
- Mode of Delivery : Online
- Project Portfolio : Available
- Certification : Available
- Indivdual Video Support : 4 month
- Group Video Support : Available
- Email Support : Available
- Whatsapp Support : Available
- Telephone Support : Available
- Add-ons Induatry Projects : 1

Premium

Lifetime Access

₹15000

Enroll Now

Per month for 3 months

- Access Duration : 2 months
- Mode of Delivery : Online
- Project Portfolio : Available
- Certification : Available
- Indivdual Video Support : 12/month
- Group Video Support : 12/month
- Email Support : Available
- Whatsapp Support : Available
- Telephone Support : Available
- Add-ons Induatry Projects : 2
- Dedicated Support Engineer : Available

Contact Details

✉ info@skill-lync.com

📱 +91 8939850851

🏠 BAID Hi-Tech Park 129B,
2nd & 3rd Floor, Valmiki Nagar,
East Coast Road,
Thiruvanmiyur, Chennai - 600041.

▶ https://www.youtube.com/user/edxengine

📷 skill_lync

in https://www.linkedin.com/school/skill-lync/

SKILL LYNC