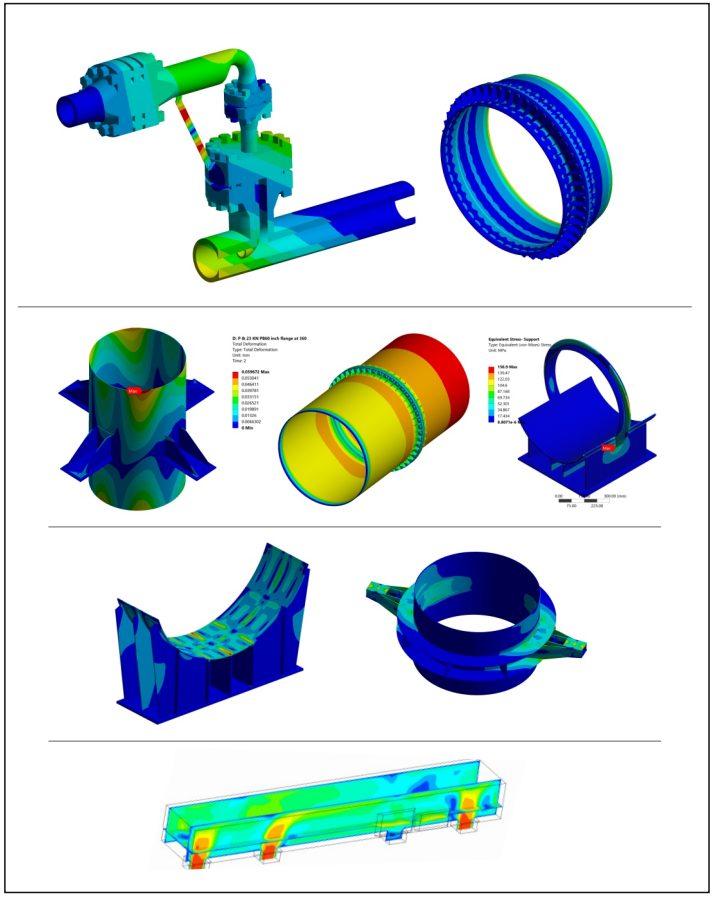
protton
synergy
+vely charged



: CONTACT US :
A 204, Rajdarshan, Dada Patil Compound,
Opp. Platform No. 1, Thane (W), Maharashtra - 400 602
P : + 91 22 4131 8404
E : gaurav.bhende@protton.co.in, info@protton.co.in
W : www.protton.co.in

protton
synergy
+vely charged

CORPORATE AND INDIVIDUAL TRAINING

PROTTON SYNERGY (formerly known as Protton Engineering) is a specialized engineering consultancy firm providing total engineering and design solutions in the field of Process & Petrochemical Industry. We value our clients by providing Ethical, Economical, Effective and Efficient engineering solutions.

'We make your HY' clear

Application of

FINITE ELEMENT ANALYSIS (FEA)

in Oil and Gas / Process industry

The Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM).

Computational Fluid Dynamics (CFD) is the process of mathematically modeling a physical phenomenon involving fluid flow and solving it numerically using the computational process.

Engineers use FEA software to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster while saving on expenses.

In a CFD software analysis, the examination of fluid flow in accordance with its physical properties such as velocity, pressure, temperature, density and viscosity is conducted. To virtually generate an accurate solution for a physical phenomenon associated with fluid flow, those properties have to be considered simultaneously.

MODULE 1

INTRODUCTION TO COURSE

This module briefs about overview of the course. Topics that will be covered during the course are elaborated.

- Modelling
- Geometry preparation
- Analysis overview
- Meshing
- Linear static analysis

MODULE 2

FINITE ELEMENT ANALYSIS

Briefing about Finite Element method and introduction to Finite element analysis.

MODULE 3

MODELLING

This module briefs about sketching and modelling operations.

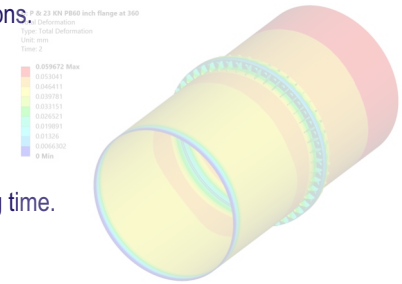
- Briefing user interface to modeler (spaceclaim).
- 2d sketching operations such as polygon, arc and modify operations.
- 3d operations such as extrude, revolve, sweep, etc

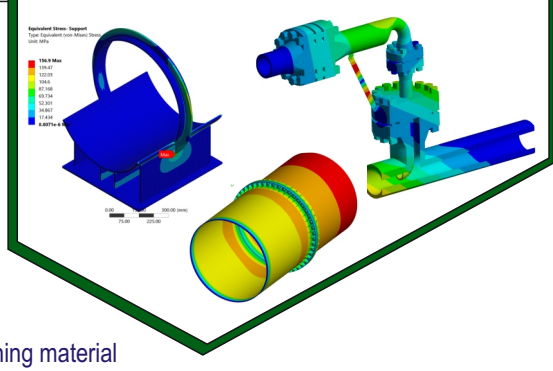
MODULE 4

GEOMETRY PROCESSING

In certain models, there are few modifications required before meshing the model. These operations aid in reducing the processing time.

- Operations such as combine, subtract, etc.
- Operations such as projections, slice, etc.





MODULE 5

ANALYSIS OVERVIEW

Complete processes from preprocessing, analysis and post processing will be performed to understand the FEA project flow. This module will give us an overview of complete analysis procedure.

MODULE 6

ASSIGN MATERIAL PROPERTIES

This module briefs us about assigning material and defining material properties as per analysis module

MODULE 7

MESHING OVERVIEW

Discretization of the model will be studied in this section. Different meshing options will be examined as per type of analysis to be performed.

- Type of element 1d, 2d, 3d elements
- Type of meshing
- Element quality

MODULE 8

LINEAR STATIC ANALYSIS – SIF CALCULATIONS

Performing linear static analysis in static structural module. Modelling and analyzing piping joints. Validation of stress intensification factor using FEA tool.

- Miter bend
- Elbow modelling and analysis

MODULE 9

DUCT PIPING ISSUES

In case of a duct component, the diameter to thickness ratio is high as compared to pipe. Thus FEA tool is used to check the component for buckling load

- External pressure vacuum check
- Euler bucking and circumferential buckling

MODULE 10

ANALYZING PIPING SUPPORTS

In a power or process plant, different type of pipe supports are used based on the location and loads of the piping system

- Basic support system
- Lug support
- Saddle support

MODULE 11

OPTIMIZATION OF SUPPORTS

If the standard thickness and support arrangement does not qualify with the allowable stress values additional reinforcement or membrane and bending stresses are checked.

- Adding stiffness rings
- Linearization

MODULE 12

OPTIMIZATION OF SUPPORTS

Process of extraction results and formatting report will be studied in this module. General report format, mesh acceptance values, etc. will be discussed