



Ansys Training Outline

POLYTRON

Day	Ansys SpaceClaim	
Day-0	Ansys SpaceClaim Introduction	What is Ansys SpaceClaim?
		Discovering SpaceClaim Interface
		Discovering Sketching Tools
		Discovering Modeling Tools
		Common Issues While Importing Geometry
		Discover Tools for Repair
		Creating Mid-Surfaces in SpaceClaim
		Discovering Beam Extraction in SpaceClaim

Day	Ansys Mechanical	
Day-1	Getting Started with Ansys Mechanical	Problem Statement
		Modeling Approach
		Geometry Preparation
		Defining Material Properties
		Applying Loads, and Supports
		Analysis Settings
		Results and Validation
Day-2	Getting Advanced with Ansys Mechanical	Improved Modeling Approach
		Further Geometry Considerations
		More Realistic Connections
		Enhanced Mesh Techniques
		Additional Analysis Settings, Loads, and Supports
		Expanded Results and Validation
		Parameters and Associativity

Day-3	Workshop	Problem Statement
		Modeling Approach
		Geometry Preparation
		Defining Material Properties
		Applying Loads, and Supports
		Analysis Settings
		Results and Validation
Day	Ansys LS-Dyna	
Day-1	Workshop-1: Simulasi Taylor Impact	Membuat geometry dengan spaceclaim
		Material definition: Elastic & plastic region (bilinear isotrphic hardening)
		Meshing: Global sizing, automatic hexahedral meshing
		Mendefinisikan kontak: frictionless & frictional
		Menentukan boundary conditions: Initial velocity, ground constraint
		Analysis settings: Mass scaling, time step, save time step
		Energy conservation and verification
	Workshop-2: Simulasi Bumper Impact	Mengolah hasil simulasi: Stress, deformation, strain, eps
		Import geometry dan cleanup dengan spaceclaim
		Menentukan rigid dan flexible body
		Material definition: Elastic & plastic region (bilinear isotrphic hardening)
		Meshing: Global sizing, local sizing, & automatic hexahedral meshing
		Mendefinisikan kontak: frictional
		Menentukan boundary conditions: Initial velocity vs constant velocity
Day-2	Workshop-3: Simulasi Impact pada box Frame	Analysis settings: Mass scaling, time step, save time step
		Mengolah hasil simulasi: Stress, deformation, strain, eps
		Import geometry, midsurfacing, extend, share topology.
		Menentukan rigid dan flexible body
		Material definition: Elastic & plastic region (multilinear isotrophic hardening)
		Meshing: Batch connection, local sizing, & automatic hexahedral meshing
		Mendefinisikan kontak: bonded & frictional
Menentukan boundary conditions: Constant velocity		
Analysis settings: Mass scaling, time step, save time step		
Mengolah hasil simulasi: Stress, deformation, strain, eps		

Day-3	Workshop-4: Simulasi Drop Test	Cleanup geometry dengan spaceclaim (small faces, rounded, midsurfacing)
		Material definition: Elastic & plastic region (bilinear isotrphic hardening)
		Meshing: Global sizing, local sizing, & automatic hexahedral meshing
		Mendefinisikan kontak: bonded, remote points, frictional
		Menentukan boundary conditions: height and gravity
		Analysis settings: Mass scaling, time step, save time step
Mengolah hasil simulasi: Stress, deformation, strain, eps		

Day	Ansys CFD	
Day-1	Workshop-1: Flow in pipe	Membuat 3D computational domain menggunakan Space Claim Design Modeller
		Membuat naming selection menggunakan Space Claim Design Modeller
		Membuat hexahedral meshing menggunakan ANSYS Meshing
		Membuat setup menggunakan ANSYS FLUENT (memberikan kecepatan masuk fluida)
		Mengambil kontur kecepatan, kontur vektor kecepatan dan grafik kecepatan menggunakan ANSYS FLUENT
	Workshop-2: Simulasi fan	Membuat 3D computational domain menggunakan Space Claim Design Modeller (interior boundary conditions)
		Membuat naming selection menggunakan Space Claim Design Modeller
		Membuat hexahedral meshing menggunakan ANSYS Meshing
		Membuat setup menggunakan ANSYS FLUENT (menginput fan curve)
		mengambil kontur kecepatan, tekanan, vektor, dan streamline menggunakan ANSYS FLUENT
Day-2	Workshop-3: Concentric Heat Exchanger	Membuat lebih dari satu 3D computational domain menggunakan Space Claim Design Modeller
		Membuat naming selection menggunakan Space Claim Design Modeller
		Membuat hexahedral meshing menggunakan ANSYS Meshing
		Membuat setup perpindahan panas menggunakan ANSYS FLUENT (memberikan temperature masuk fluida)
		melihat fenomena perpindahan panas konveksi-konduksi dengan melihat distribusi temperature pada udara, air dan pipa alumunium

Day-3	Workshop-4: Film kondensasi	Membuat 3D computational domain menggunakan Space Claim Design Modeller
		Membuat naming selection menggunakan Space Claim Design Modeller
		Membuat hexahedral meshing menggunakan ANSYS Meshing
		membuat setup film kondensasi menggunakan ANSYS FLUENT (setting fluida mana yang mengalami kondensasi)
		mengambil kontur species mass fraction, relative humidity, film thickness dan film mass menggunakan ANSYS FLUENT