



AUGI Design Academy 2008

Autodesk Inventor Prof.

Motion Simulation and FEA

Wasim Younis

S3-5

Course Summary:

At first glance, the world of motion simulation can appear to be highly technical. You'll be pleasantly surprised during this course to learn just how easy it is to set up and use! The unit will cover an introduction and overview followed by several examples that demonstrate how to set up a variety of simulations, helping you work more productively with Autodesk Inventor.

Instructor:

Wasim has been involved with Autodesk Inventor Simulation from when it was introduced and works as a consultant for Autodesk Inc. Some of the recent work include producing, AOTC Training Courseware on Inventor Simulation, white papers and marketing collateral for Autodesk.

Wasim is director of VDS Solutions which offers Autodesk Inventor Simulation training, coaching and consultancy to the manufacturing sector within UK.





Autodesk User Group International

www.AUGIuk.com

Motion Simulation and FEA.

Over the past couple of decades there have been several dedicated FEA and Motion Analysis Software available to use by Engineers. These technologies have been very comprehensive and are predominantly used by analysis and simulation engineering experts. Some of the other major drawbacks of such dedicated software include the advanced knowledge required and their lack of seamless integration between 3D CAD software. As a result of these drawbacks designers and experts still continue today to work in isolation from one another resulting in some duplication of work, increased design change time and cost and hence increased time to market.

These major drawbacks of past technologies have come one step closer with the advent of Autodesk Inventor 2008 Simulation. Here the designer has seamless links with his CAD models and the knowledge required is minimal to drive the analysis and simulation technologies available within Autodesk Inventor 2008 Simulation. In brief this new technology allows designers, themselves, to take their CAD designs one step further in allowing them to create a working virtual prototype of their designs. This will thus help designers to analyze and modify their designs efficiently and cost effectively.

Autodesk Inventor 2008 Simulation is a must if you are looking for an engineering tool that will resolve design problems, reduce failures and warranty costs, turn around designs faster and work seamlessly with your CAD models. Its for engineers and designers whom are involved in the design and manufacture of;

- Industrial machinery
- Medical devices
- Automotive
- Aerospace
- Equipment for construction
- Consumer products

FEA Analysis - Basic theory.

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. Most of the FEA Analysis software packages are based on the finite element method this is where a component is broken down into many small simple segments or elements. This process of creating elements in FEA analysis is called meshing.

There are mainly two types of FEA Analysis and are;

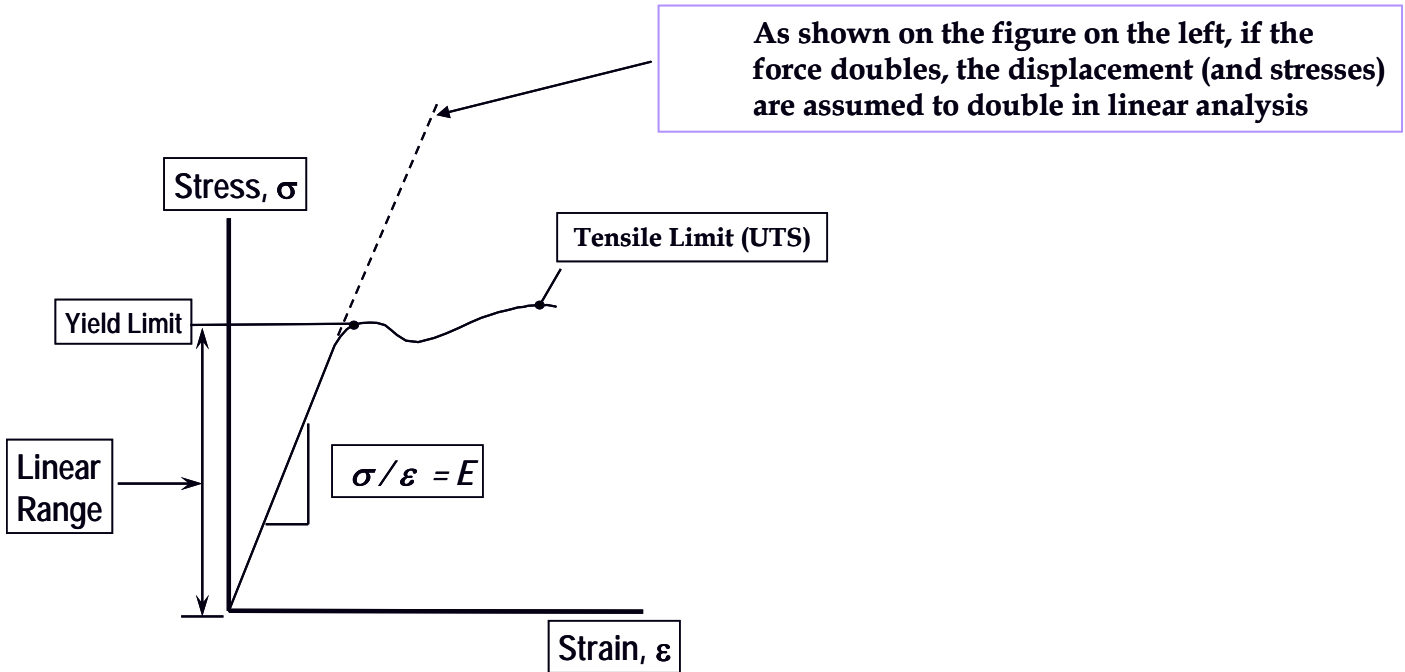
1. Linear Analysis – Currently supported
2. Non-Linear Analysis – Will be supported in future releases

Both these discussed further in the following sections.





Linear Analysis.

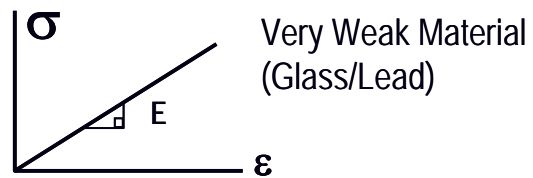
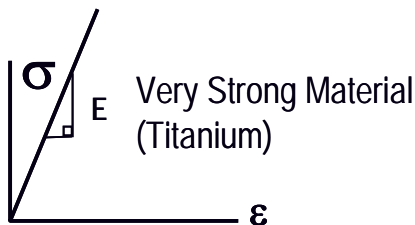


Some definitions:

- Stress = Force / Area
A large force or a small area will produce high stresses within the component.

TIP: When applying loads it is advisable not to apply loads at point or small edges as this will produce very high localized stresses.

- Youngs Modulus = Stress / Strain
The Youngs Modulus provides the stiffness of the material, for example a high youngs modulus will produce a stronger material and lower youngs modulus will produce a weaker material as illustrated below.



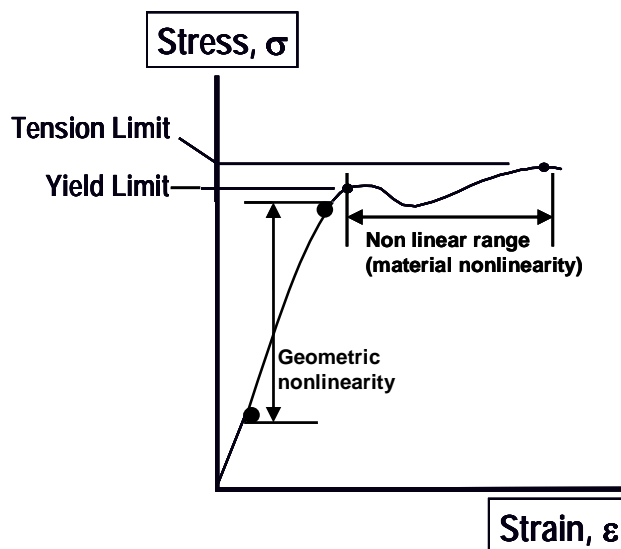


- Strain = Change in length / Original length
For linear analysis it is assumed the change in length is very small compared to the original length
- Factor of Safety = Calculated Stress / Yield Limit Stress

The following are some of the assumptions made to conduct a linear analysis

1. The material properties of the component remain linear after the yield limit. Hence results beyond this limit are not valid using a linear analysis software
2. The deflections of components are small compared to overall component size
3. Only interested in determining a factor of safety above 1 for the component in question
4. The components are rigid such as metal components. (Not rubber)
5. The components deform equally in all three directions ie. Material properties are isotropic

Non-Linear Analysis – Currently not supported in Inventor 2008 Simulation Suite.



Non-linear Analysis falls into the following 3 categories.

1. Geometric nonlinearity – This is where a component experiences large deformations and as a result can cause the component to experience non-linear behaviour as for example in a fishing Rod.
2. Material nonlinearity – When the component goes beyond the Yield limit the Stress/Strain relationship becomes non-linear as the material starts to deform permanently
3. Contact – Includes the effect of when two components come into contact where they can experience an abrupt change in stiffness resulting in localized material deformation, at region of contact.





Mesh and Convergence.

As explained earlier the process of breaking down the component into smaller elements is called meshing within the field of Finite Element Analysis. Currently there are three types of elements within Inventor 2008 and are;

1. Hex solid
2. Tetrahedron solids
3. Shell elements (for sheet metal components)

Hex elements are simpler and more accurate hence less of them are required in the component. This means the computer time to calculate the results will be less. The downside of this element is that it can only be applied to simple solid components. Tet elements on the other hand are the default element type and can be used to mesh any component simple or complex.

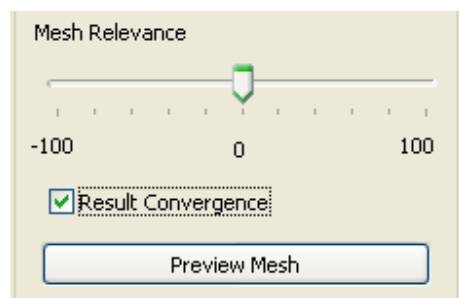
NB: The more tet elements there are in the component the longer it takes to run the analysis.

TIP: Always run the analysis on the default mesh as this will give you an indication of whether the behaviour of the component under the applied loading is correct.

TIP: The deflection results have no effect on the number of elements. Therefore if its deflection you are interested in there is no need to rerun the analysis again with a different mesh control and also no need to converge the results.

TIP: For Sheet metal components always use Shell elements.

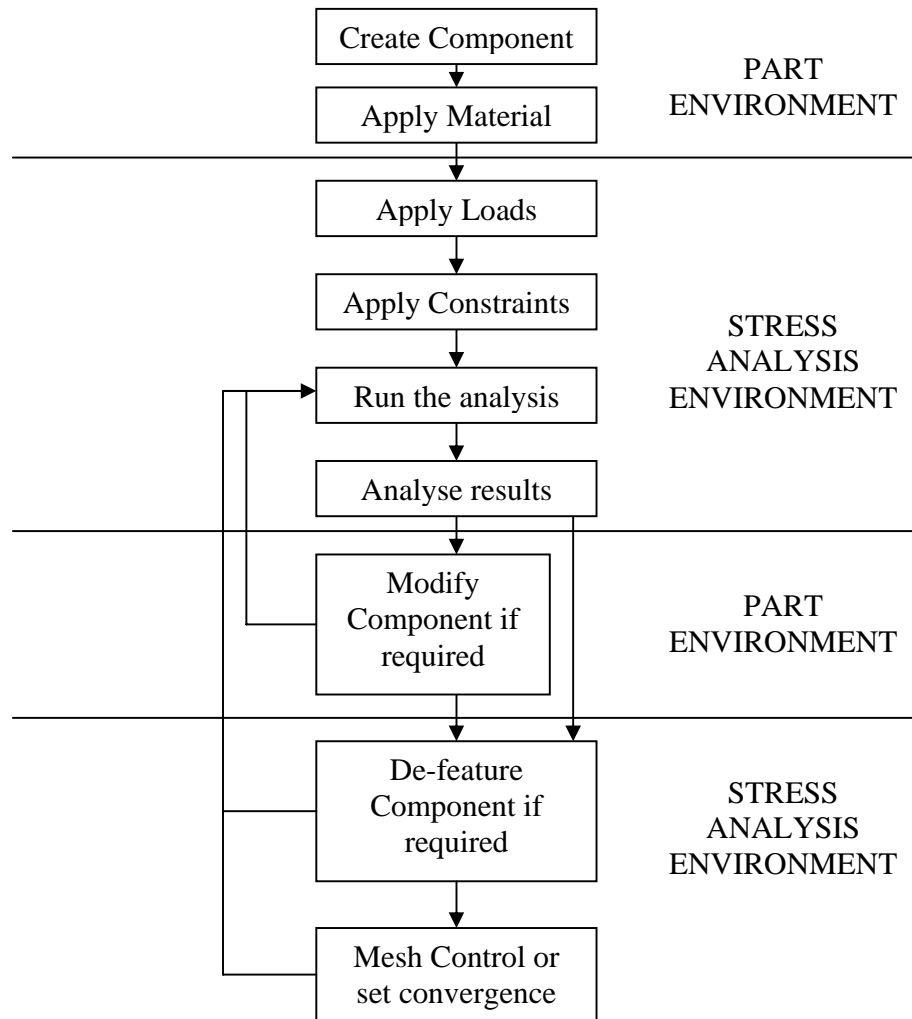
In order to check whether the stress results have converged, you will need to rerun the analysis again with more elements to check whether the stress has changed within 10% of the first value. If the results are within 10% there is no need to run the analysis again. However if it is above 10% you will need to run the analysis again or/and click the convergence button as detailed below.



By applying the convergence button the software will calculate a maximum of three results to check whether the stress value has converged. However there may be cases that the stress results may not converge at all in this case you will need to modify how your loading is applied on the component.



FEA Analysis Workflow – Explained



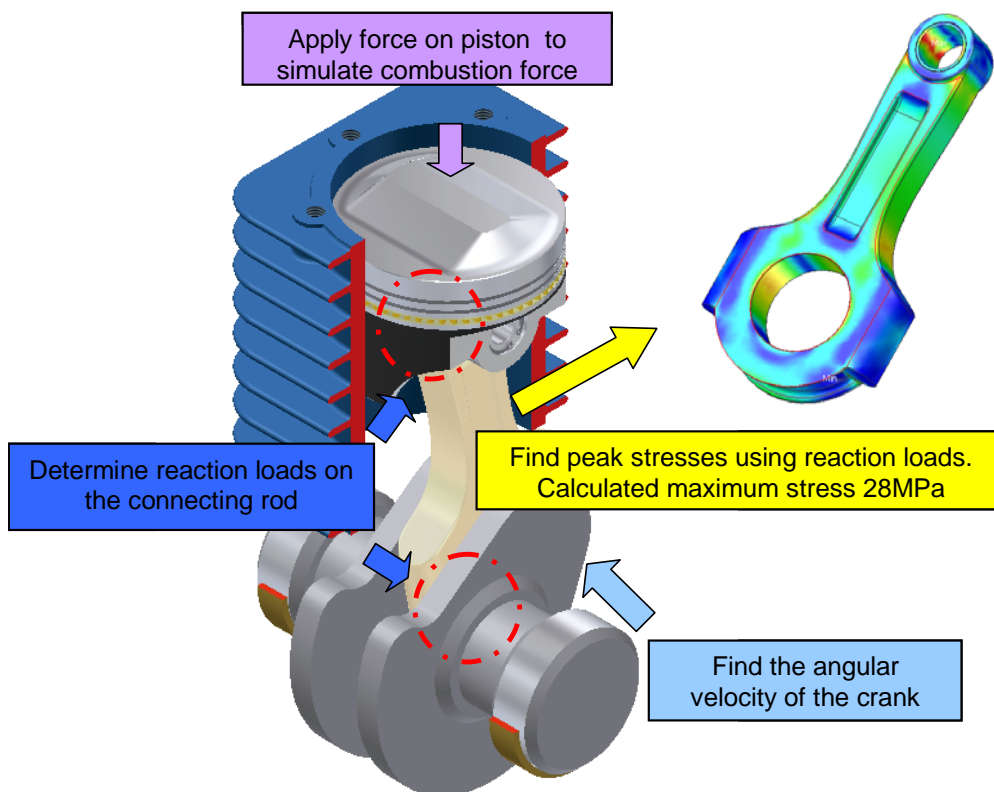


Motion Simulation – Explained in English!

Motion Simulation provides not only animation but information on kinematics and dynamics of the moving assembly including positions, velocities, accelerations, reaction forces and torques of joints.

Why use Motion Simulation

During a typical design process, designers go through a series of typical questions like do the parts fit together? Do the parts move well together? Is there interference? And do the parts follow the right path? Even though most of these questions can be catered for by 3D CAD and Rendering Software there may be other questions which cannot? For example designers may want to know the machinery time cycle, Is the actuator powerful enough? Is the link robust enough? And can we reduce weight? All these questions can only be answered by building a working prototype or a series of prototypes. The major issue with this method is that it's timely and costly. An alternative cost effective method is to create a working virtual prototype by using motion simulation. In all this information will help designers to build an optimum product as illustrated by the following example.





Motion Simulation Workflow.

In order to carry out a motion simulation on an assembly there are three core steps you need to follow and are;




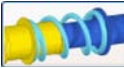



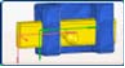

1. Create joints
2. Apply environmental constraints
3. Run and analyse results

Create Joints

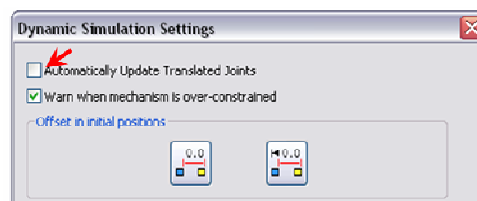
In Motion Simulation there are four categories for joint types including.

1. Standard
2. Rolling and Sliding
3. 2D Contact
4. Force

These categories are illustrated below

Examples			
Standard	Rolling & sliding	2D Contact	Force
 Pivot	 Cylinder in cylinder		 Spring
 Spherical	 Screw		 3D Contact
 Prismatic	 Worm gear		
and much more.	and much more.		

TIP: Assembly constraints can be automatically converted into standard joints by selecting the following setting.



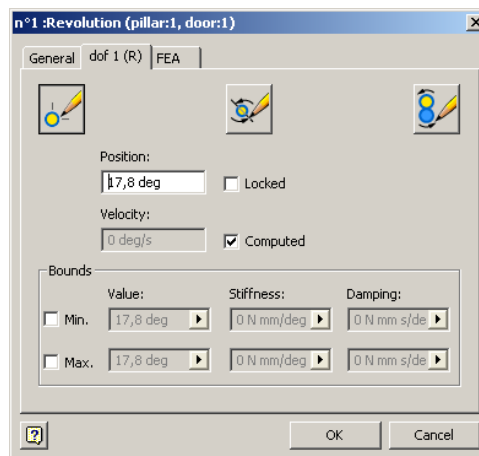


NB: The only joints which can be automatically converted from Assembly constraints are Standard Joints. All the other joints including 2D contact and Spring have to be created within the Motion Simulation environment.

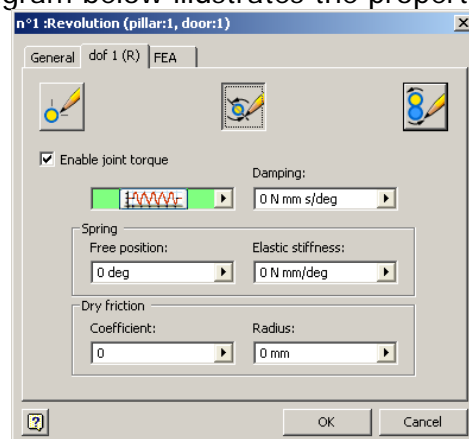
TIP: Create Subassemblies within Assembly environment to enable grouping of individual components with similar motion. This will help to simplify joint creation process and help towards creating an efficient simulation model.

Apply Environmental Constraints.

At this stage of the workflow you set the initial conditions of the joints including position and velocities as illustrated below

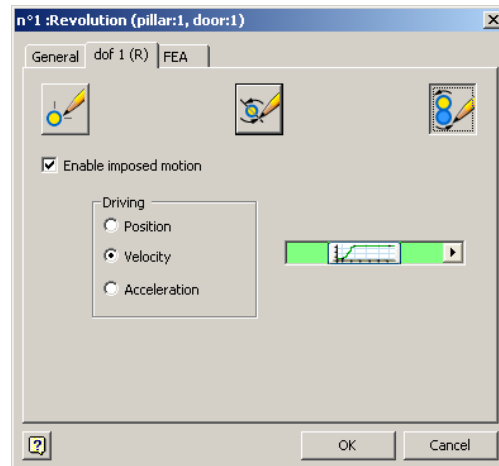


Secondly, but optional, the joint torque settings can be enabled to mimic real life joints that include joint frictional properties. The diagram below illustrates the properties that can be applied.

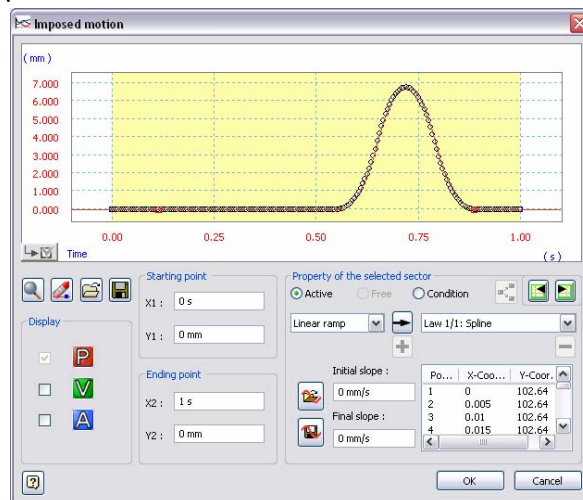




Finally you can impose position, velocity or acceleration on the joints to create motion of the assembly by specifying a constant value as illustrated below.



Or you can create complex motion equations using the input grapher to get the desired motion of the assembly as illustrated below.



The input grapher has standard equations including ramp, sine, harmonic and much more. The standard equations can be combined to create complex input motions.

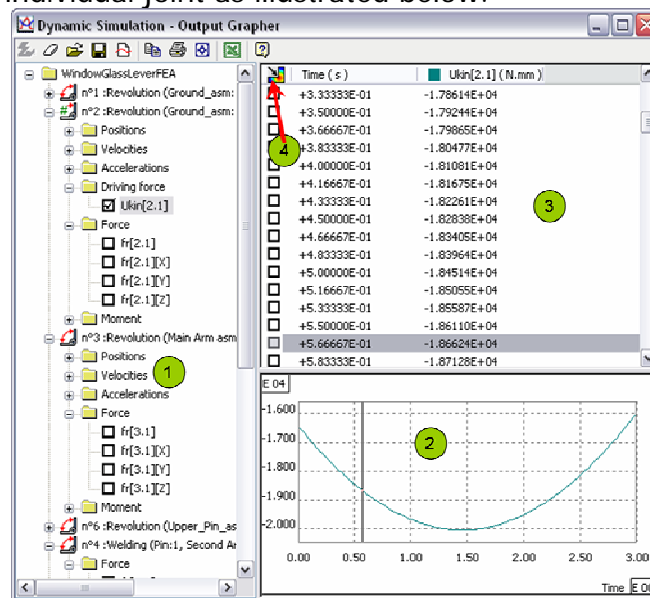
TIP: Data can be imported into the input graphed from a predefined data within a text file.

TIP: Force can also be applied in addition, or as alternative, to imposed motion.



Run and Analyse Results

At this final stage of the workflow you can check whether the assembly is correctly mimicking reality. Apart from visually looking at the animation of the assembly you can use the powerful output grapher to better understand each individual joint as illustrated below.

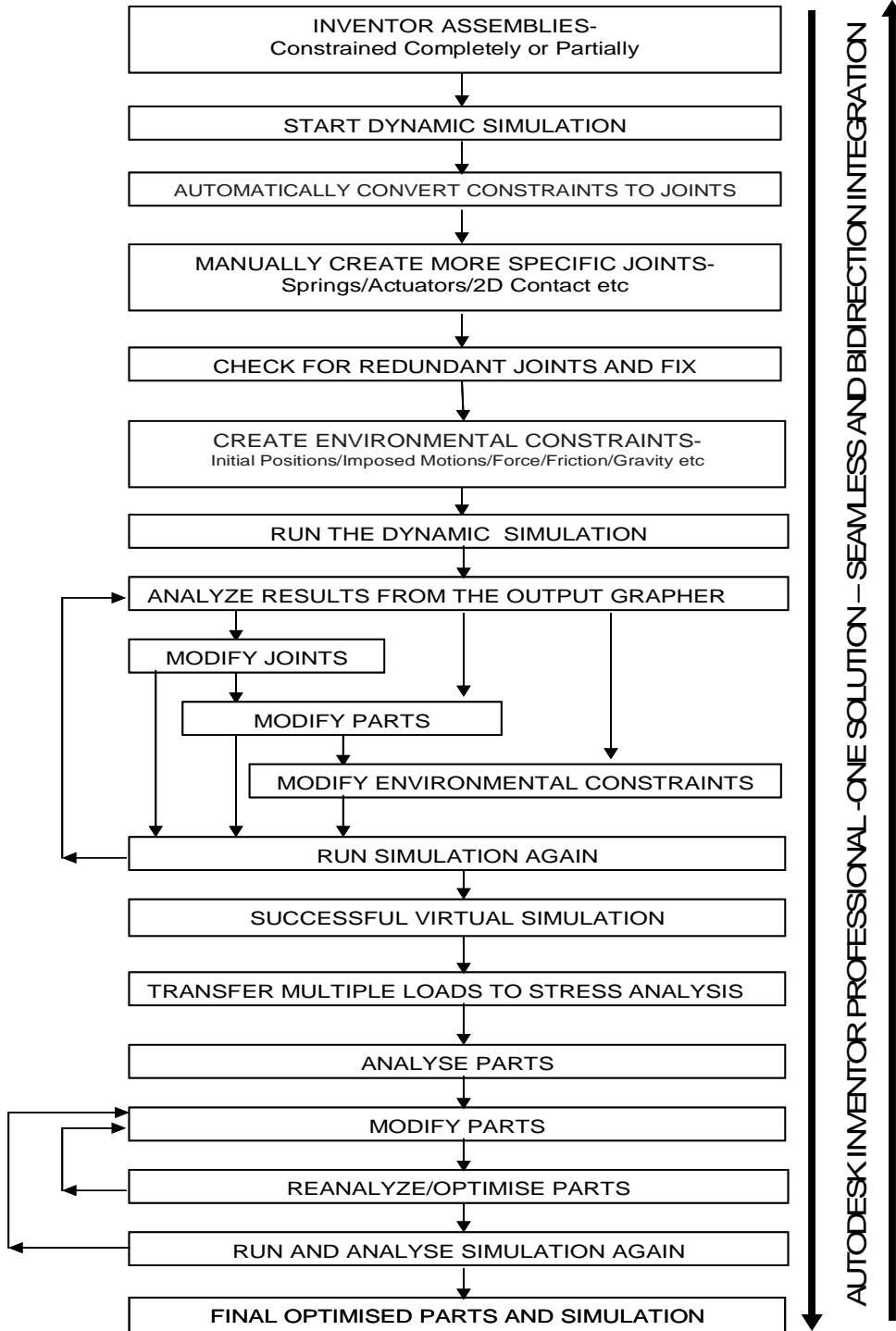


1. Analyse position, velocity, acceleration or forces for each individual joint.
2. Graphical plot of chosen joint results.
3. Tabular data of chosen results for each time frame.
4. The ability to export reaction forces to the Stress Analysis environment (as motion loads).

TIP: If you need to analyse a component within an assembly then it is best to conduct the simulation of the whole assembly and then export the reaction loads to the Stress Analysis Environment, as this will help to reduce errors as a result of poor boundary conditions.



Motion Simulation and FEA Total Workflow Diagram





Some of the design problems that will be discussed in the course are;

1. CAM Design
2. Stress Analysis of Bike Release lever
3. Sizing of a spring for a bike Suspension
4. Stress Analysis of a bracket
5. Simulation of a Piston Assembly
6. 3D Contact Simulation

Conclusion

Autodesk Inventor 2008 Simulation has enabled to remove the shortcomings of the traditional dedicated FEA and Motion solutions by making the software easy to use, providing seamless integration and most importantly it is a designer's tool. Another important point to note is that it provides seamless transfer of reaction forces to the stress analysis environment. It is hoped that Inventor 2008 simulation would aim to help reduce or eliminate the need to build physical prototypes, which are needed, for example, to produce reaction loads and most importantly to see if the product function's. Finally it is anticipated that with the aid of Inventor 2008 the designer will have the capability to;

- Get to market faster
- Decrease warranty costs
- Improve product quality