International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

Modelling and Vibration Analysis of Car Spoiler at Different Speeds

P.Akhil[1], C Lakshmaiah[2], A Vidya Sagar[3]
[1]P.G. Scholar, [2]Guide Assistant Professor, [3]Head of the Department
[1,2,3] BRANCH : *MACHINE DESIGN*
[1,2,3] Geethanjali College Of Engineering & Technology
EMAIL ID : [1]akheelahmed229@gmail.com, [2]lakshmaiahcherukupalle@gmail.com

Abstract

A car spoiler is a wing like accessory that is usually attached to the rear end of the cars, or normally mounted on top of a car's trunk or positioned under the front bumper. While the rear spoiler is sometimes called 'wing', the frontal car spoilers are also called 'air dam'. Car spoiler dynamically improves the external beauty of the car making the car stand out in a crowd, making it more trendy and sporty.

The main aim of the present document is to vibration characteristics of car spoiler obtained using harmonic analysis. The Design (UNIGRAPHICS) software provides different shapes of car spoil. The spoiler is modelled using UNIGRAPHICS software and is analyzed for the static deflection as well as harmonic analysis has been done by using ANSYS.

INTRODUCTION

A spoiler is an automotive aerodynamic device whose intended design function is to 'spoil' unfavorable air movement across a body of a vehicle in motion, usually described as turbulence or drag. Spoilers on the front of a vehicle are often called air dams. Spoilers are often fitted to race and high-performance sports cars, although they have become common on passenger vehicles as well. Some spoilers are added to cars primarily for styling purposes and have either little aerodynamic benefit or even make the aerodynamics worse.

The term "spoiler" is often mistakenly used interchangeably with "wing". An automotive wing is a device whose intended design is to generate down force as air passes around it, not simply disrupt existing airflow patterns.[1][2] As such, rather than decreasing drag, automotive wings actually increase drag.

Since spoiler is a term describing an application, the operation of a spoiler varies depending on the particular effect it's trying to spoil. Most common spoiler functions include disrupting some type of airflow passing over and around a moving vehicle. A common spoiler diffuses air by increasing amounts of turbulence flowing over the shape, "spoiling" the laminar flow and providing a cushion for the laminar boundary layer. However, other types of airflow may require the spoiler to operate differently and take on vastly different physical characteristics.

SPOILERS IN RACING CARS

While a mass is travelling at increasing speeds, the air of the environment affects its movement. Spoilers in racing are used in combination with other features on the body or chassis of race cars to change the handling characteristics that are affected by the air of the environment.

Often, these devices are designed to be highly adjustable to suit the needs of racing on a given track or to suit the talents of a particular driver, with the overall goal of reaching faster times.

TRAVELLING CARS

The goal of many spoilers used in passenger vehicles is to reduce drag and increase fuel efficiency. Passenger vehicles can be equipped with front and rear spoilers. Front spoilers, found beneath the bumper, are mainly used to decrease the amount of air going underneath the vehicle to reduce the drag coefficient and lift.

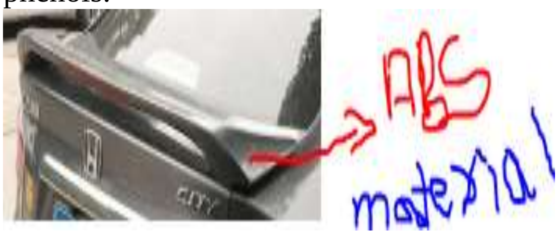
Sports cars are most commonly seen with front and rear spoilers. Even though these vehicles typically have a more rigid chassis and a stiffer suspension to aid in high speed manoeuvrability, a spoiler can still be beneficial. This is because many vehicles have a fairly steep downward angle going from the rear edge of the roof down to the trunk or tail of the car which may cause air flow separation. The flow of air becomes turbulent and a low-pressure zone is created, increasing drag and instability. Adding a rear spoiler could be considered to make the air "see" a longer, gentler slope from the roof to the spoiler, which helps to delay flow separation and the higher pressure in front of the spoiler can help reduce the lift on the car by creating down force. This may reduce drag in certain instances and will generally increase high speed stability due to the reduced rear lift.

Due to their association with racing, spoilers are often viewed as "sporty" by consumers. However, "The spoilers that feature on more upmarket models rarely provide further aerodynamic benefit.

MATERIALS USED FOR DEVELOPMENT OF SPOILERS:

Spoilers are usually made of lightweight polymer-based materials, including:

ABS plastic: Most original equipment manufacturers produce spoilers by casting ABS plastic with various admixtures, typically granular fillers, which introduce stiffness to this inexpensive material. Frailness is a main disadvantage of plastic, which increases with product age and is caused by the evaporation of volatile phenols.



e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

Fiberglass: Used in car parts production due to the low cost of the materials. Fiberglass spoilers consist of fibreglass cloth infiltrated with a thermosetting resin, such as epoxy. Fiberglass is sufficiently durable and workable, but has become unprofitable for large-scale production because of the labour involved.

Silicone: More recently, many auto accessory manufacturers are using silicon-organic polymers. The main benefit of this material is its phenomenal plasticity. Silicone possesses extra high thermal characteristics and provides a longer product lifetime.

Carbon fiber: Carbon fiber is lightweight and durable but also expensive. Due to the large amount of manual labor, large-scale production cannot widely use carbon fiber in automobile parts currently.

ACTIVE SPOILERS

An active spoiler is one which dynamically adjusts while the vehicle is in operation based on conditions presented, changing the spoiling effect, intensity or other performance attribute. Found most often on sports cars and other passenger cars, the most common form is a rear spoiler which retracts and hides partially or entirely into the rear of the vehicle, then extends upwards when the vehicle exceeds a specific speed. Active front spoilers have been implemented on certain models as well, in which the front spoiler or air dam extends further towards the road below to reduce drag at high speed. In most cases the deployment of the spoiler is achieved with an electric motor controlled automatically by the onboard computer or other electronics, usually based on vehicle speed, driver setting or other inputs. Often the driver can manually deploy the spoiler if desired, but may not be able to retract the spoiler above a certain speed because doing so could dangerously diminish the high-speed handling qualities of the vehicle.

Active rear spoiler

Active spoilers can offer additional benefits over fixed spoilers. Cosmetically, they can allow a cleaner or less cluttered appearance when the vehicle is parked or travelling at low speeds, when it is most likely to be observed. A spoiler which hides may be appealing to vehicle designers who are seeking to improve the high-speed aerodynamics of an iconic or recognizable model (for example the Porsche 911 or Audi TT), without drastically changing its appearance. Hiding a spoiler at low speeds can improve aerodynamics as well. At low speeds a fixed spoiler may actually increase drag, but does little to improve the handling of the vehicle due to having little airflow over it. A retractable front spoiler can reduce scraping of the car on curbs or other road imperfections, while still improving drag at high speeds.

LITERATURE REVIEW

[1] Investigation of Drag and Lift Forces over the Profile of Car with Rear spoiler Using CFD Now a day's demand of a high speed car is increasing in which vehicle stability is of major concern. Forces like drag & lift, weight ,side forces and thrust acts on a vehicle when moving on road which significantly affect the fuel consumption The drag force is produced by relative motion between air and vehicle and about 60% of total drag is produced at the rear end. Reduction of drag force at the rear end improves the fuel utilization. This work aims to reduce the drag force which improves fuel utilization and protects environment as well. In the stage of work a sedan car with different types of spoilers are used to reduce the aerodynamic drag force. The design of sedan car has been done on CATIA-2010 and the same is used for analysis in ANSYS-(fluent). The analysis is done for finding out drag and lift forces at different velocities, and spoilers. This study proposes an effective numerical model based on the computational fluid dynamics (CFD) approach to obtain the flow structure around a passenger car with a rear spoiler.

[2] Environmental issues and increased fuel are driving forces for the automotive manufactures to develop more fuel efficient vehicles with lower emissions. The need for fuel efficiency is a rapidly increasing trend in automotive industries in the recent years. Therefore, extensive research is undergoing for development of aerodynamically optimized vehicle designs. One of the design goals of the spoiler is to reduce drag and increase fuel efficiency. The drag coefficient is an important factor that determines the fuel efficiency of a vehicle in close proximity to the ground. The primary objective of the project is to study the effects of fluid flow and the effective drag of the vehicle over a 3D standard car (BOLERO) with attached Rear Spoiler by using Computational Fluid Dynamics (CFD) simulation. A 1:1 scale model of the actual vehicle was designed in CAD package SOLIDWORKS and CATIA V5 R20. CFD analysis was done over the scaled model keeping conditions as close as possible to the actual road conditions.

[3] The research deal with Computational Fluid Dynamic analysis and simulation to maximize down force and minimize drag during high speed of the car. Using ANSYS FLUNT software and mentoring provided by ANSYS , the results employs efficient Descritization techniques and real loading conditions to study down force on rear wing of the car with drag generated by all active mounted surfaces. Wing and external surface under high velocity runs of the car are presented. Optimization of wing direct angle of attack and geometry modifications on surfaces of the car are performed to enhance down force and reduce drag for higher degree of stability and to control during operation. Moreover to ensure more stability active aerodynamic spoiler which can adjust its height and its angle of attack according to speed. So the aim of research to find best height at which maximum down force and air flow separate ,so that car body can get high level of stability . And another factor is angle of attack at which highest value of down force we get.

Investigation of Drag and Lift Forces over the Profile of Car with Rear spoiler Using CFD:

Now a day's demand of a high speed car is increasing in which vehicle stability is of major concern. Forces like drag& lift, weight ,side forces and thrust acts on a vehicle when moving on road which significantly affect the fuel consumption

The drag force is produced by relative motion between air and vehicle and about 60% of total drag is produced at the rear end. Reduction of drag force at the rear end improves the fuel utilization. This work aims to reduce the drag force which improves fuel utilization and protects environment as well. In the stage of work a sedan car with different types of spoilers are used to reduce the aerodynamic drag force. The design of sedan car has been done on CATIA-2010 and the same is used for analysis in ANSYS-(fluent). The analysis is done for finding out drag and lift forces at different velocities, and spoilers. This study proposes an effective numerical model based on the computational fluid dynamics (CFD) approach to obtain the flow structure around a passenger car with a rear spoiler.

Design and analysis of a new rear spoiler for SU vehicle Mahindra bolero using CFD

Environmental issues and increased fuel are driving forces for the automotive manufactures to develop more fuel efficient Vehicles with lower emissions. The need for fuel efficiency is a rapidly increasing trend in automotive industries in the recent years. Therefore, extensive research is undergoing for development of aerodynamically optimized vehicle designs. One of the design goals of the spoiler is to reduce drag and increase fuel efficiency. The drag coefficient is an important factor that determines the fuel efficiency of a vehicle in close proximity to the ground. The primary objective of the project is to study the effects of fluid flow and the effective drag of the vehicle over a 3D standard car (BOLERO) with attached Rear Spoiler by using Computational Fluid Dynamics (CFD) simulation. A 1:1 scale model of the actual vehicle was designed in CAD package SOLIDWORKS and CATIA V5 R20. CFD Analysis was done over the scaled model keeping conditions as close as possible to the actual road conditions. For evaluation, optimization, the Reynolds-Averaged NavierStokes (RANS) equations with Reliable k-ε turbulence model was used over commercial package ANSYS 14, FLUENT CFD Solver. The effect of aerodynamic drag is significant only at higher velocities. Therefore, the simulation was done for vehicle speed at 80 kmph and the results were compared with scaled base vehicle various velocity, pressure, streamline contours and velocity plots were examined and analyzed at rear part of the vehicle. It was concluded that, the Co-efficient of drag (Cd) of the vehicle with attached Rear Spoiler went down by 4.8%.

Design and analysis a new spoiler for ump rev using CFD

This thesis presents the design and develops new rear spoilers to overcome a drag force that is created because of low pressure zone at the rear hack. With the new design of rear spoiler for the Proton iswara Hybrid Electrical Vehicle (BV) UMP, the performance of the car had increased from the aspect of acceleration and also from the aspect of handling of the car, the controlled speed of this research is from 80 km/hr to 110 km/hr. The design of rear spoiler is based on the type of the car used, therefore aerodynamic shape of the body and the point of the rear spoiler is important in this research. In this research 2 spoilers have been choosing that are square back and fastback rear spoilers. According to that reason, low pressure zone will be annihilated slowly if one of the rear spoilers is put at the rear back of the car. Refer to data testing obtained, using square back spoiler will reduce more low pressure zone than using fastback spoiler. Based on the analysis, the suitable place to mount the rear spoiler is at the point where square back spoiler is mounted and it synchronized with the shape of the Proton Iswara HEy. It will give additional information to the performance car researcher to continue on this research thoroughly about the effect of spoiler on any HEY car and eventually to Malaysia's first HEV race car. In the future, this factor will give benefit to the Malaysia car developer especially HEY car developer.

Analysis of Effects of Rear Spoiler in Automobile Using ANSYS

Aerodynamic characteristics of a Honda civic car 2009 are of significant interest in reducing car-racing accidents due to wind loading and in reducing the fuel consumption. Even though these vehicles typically have a more rigid chassis and a stiffer suspension to aid in high speed manoeuvrability, a spoiler can still be beneficial. One of the design goals of a spoiler is to reduce drag and increase fuel efficiency. Many vehicles have a fairly steep downward angle going from the rear edge of the roof down to the trunk or tail of the car. Reducing flow separation decreases drag, which increases fuel economy; it also helps keep the rear window clear because the air flows smoothly through the rear window. This thesis will present a numerical simulation of flow around racing car with spoiler positioned at the rear end using commercial Autodesk flow design software. The thesis will focus on CFD-based lift and drag prediction on the car body after the spoiler is mounted at the rear edge of the vehicle. A 3D computer model of 4-door sedan car (which will be designed with commercial software Solidworks) will be used as the base model. Different spoilers, in different locations will be positioned at the rear end of vehicle and the simulation will be run in order to determine the aerodynamic effects of spoiler.

UNI GRAPHICS INTRODUCTION

INTRODUCTION TO UNI-GRAPHICS

Overview of Solid Modelling

The UNIGRAPHICS NX Modelling application provides a solid modelling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modelling feature and constraint based solid modeller. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modelling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Advantages of Solid Modelling

Solid Modelling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position.

Features

- Powerful built-in engineering-oriented form features-slots, holes, pads, bosses, pockets-capture design intent and increase productivity
- Patterns of feature instances-rectangular and circular arrays-with displacement of individual features; all features in the pattern are associated with the master feature

Blending and Chamfering

- Zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modelling Operations

- Profiles can be swept, extruded or revolved to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary

- Fixed and variable radius blends may overlap surrounding faces and extend to a Tapering for modelling manufactured near-net shape parts
- User-defined features for common design elements (UNIGRAPHICS NX/User-Defined Features is required to define them in advance.

General Operation

Start with a Sketch

Use the Sketcher to freehand a sketch, and dimension an "outline" of Curves. You can then sweep the sketch using Extruded Body or Revolved Body to create a solid or sheet body. You can later refine the sketch to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects. Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

Creating and Editing Features:

Feature Modelling lets you create features such as holes, slots and grooves on a model. You can then directly edit the dimensions of the feature and locate the feature by dimensions. For example, a Hole is defined by its diameter and length. You can directly edit all of these parameters by entering new values. You can create solid bodies of any desired design that can later be defined as a form feature using User Defined Features. This lets you create your own custom library of form features.

- **Associativity**

Associatively is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces. See Introduction to Feature Modelling for additional details.

- **Positioning a Feature**

Within Modelling, you can position a feature relative to the geometry on your model using Positioning Methods, where you position dimensions. The feature is then associated with that geometry and will maintain those associations whenever you edit the model. You can also edit the position of the feature by changing the values of the positioning dimensions.

- **Reference Features**

You can create reference features, such as Datum Planes, Datum Axes and Datum CSYS, which you can use as reference geometry when needed, or as construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. You can use a datum plane as a reference plane in constructing sketches, creating features, and positioning features. You can use a datum axis to create datum planes, to place items concentrically, or to create radial patterns.

- **Expressions**

The Expressions tool lets you incorporate your requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, you can define the height of a boss as three times its diameter, so that when the diameter changes, the height changes also.

- **Boolean Operations**

Modelling provides the following Boolean Operations: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

- **Undo**

  You can return a design to a previous state any number of times using the Undo function. You do not have to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets you cease worrying about getting it wrong, and frees you to explore more possibilities to get it right.

- **Additional Capabilities**

Other UNIGRAPHICS NX applications can operate directly on solid objects created within Modelling without any translation of the solid body. For example, you can perform drafting, engineering analysis, and NC machining functions by accessing the appropriate application. Using Modelling, you can design a complete, unambiguous, three dimensional model to describe an object. You can extract a wide range of physical properties from the solid bodies, including mass properties. Shading and hidden line capabilities help you visualize complex assemblies. You can identify interferences automatically, eliminating the need to attempt to do so manually. Hidden edge views can later be generated and placed on drawings. Fully associative dimensioned drawings can be created from solid models using the appropriate options of the Drafting application. If the solid model is edited later, the drawing and dimensions are updated automatically.

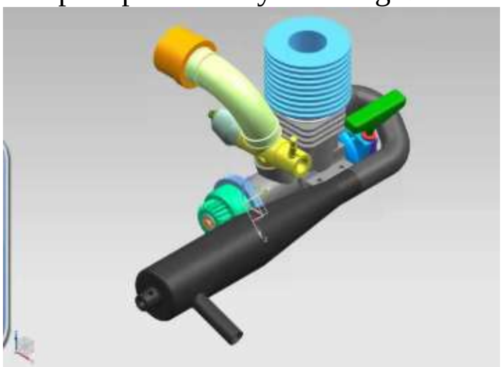
- **Parent/Child Relationships**

If a feature depends on another object for its existence, it is a child or dependent of that object. The object, in turn, is a parent of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child.A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in your work part, open the Part Navigator.

Creating a Solid Model

Modelling provides the design engineer with intuitive and comfortable modelling techniques such as sketching, feature based modelling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.

The strategy you use to create and edit your model to form the desired object depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, you can create a sketch "outline" of curves. Then you can sweep or rotate these curves to create a complex portion of your design.

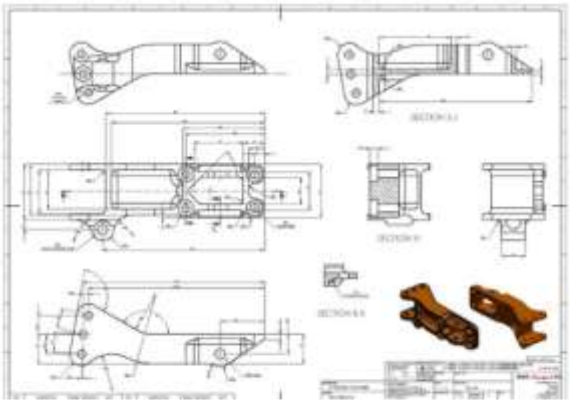


INTRODUCTION TO DRAFTING

The Drafting application is designed to allow you to create and maintain a variety of drawings made from models generated

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

from within the Modelling application. Drawings created in the Drafting application are fully associative to the model. Any changes made to the model are automatically reflected in the drawing. This associativity allows you to make as many model changes as you wish. Besides the powerful associativity functionality, Drafting contains many other useful features including the following:



- An intuitive, easy to use, graphical user interface. This allows you to create drawings quickly and easily.
- A drawing board paradigm in which you work "on a drawing." This approach is similar to the way a drafter would work on a drawing board. This method greatly increases productivity.
- Support of new assembly architecture and concurrent engineering. This allows the drafter to make drawings at the same time as the designer works on the model.
- The capability to create fully associative cross-sectional views with automatic hidden line rendering and crosshatching.
- Automatic orthographic view alignment. This allows you to quickly place views on a drawing, without having to consider their alignment.
- Automatic hidden line rendering of drawing views.
- The ability to edit most drafting objects (e.g., dimensions, symbols, etc.) from the graphics window. This allows you to create drafting objects and make changes to them immediately.
- On-screen feedback during the drafting process to reduce rework and editing.
- User controls for drawing updates, which enhance user productivity.

Finally, you can add form features, such as chamfers, holes, slots, or even user defined features to complete the object.

- **Updating Models**

A model can be updated either automatically or manually. Automatic updates are performed only on those features affected by an appropriate change (an edit operation or the creation of certain types of features). If you wish, you can delay the automatic update for edit operations by using the Delayed Update option. You can manually trigger an update of the entire model. You might, for example, want to use a net null update to check whether an existing model will successfully update in a new version of UNIGRAPHICS NX before you put a lot of additional work into modifying the model. (A net null update mechanism forces a complete update of a model, without changing it.)

The manual methods include:

- The Unigraphics NX Open C and C++ Runtime function, UF_MODL_update_all_features, which logs all the features in the current work part to the Unigraphics NX update list, and then performs an update. See the Unigraphics NX Open C and C++ Runtime Reference Help for more information.
- The Playback option on the Edit Feature dialog, which recreates the model, starting at its first feature. You can step through the model as it is created one feature at a time, move forward or backward to any feature, or trigger an update that continues until a failure occurs or the model is complete.

- The Edit during Update dialog, which appears when you choose Playback, also includes options for analyzing and editing features of the model as it is recreated (especially useful for fixing problems that caused update failures).Methods that users have tried in the past that has led to some problems or is tricky to use:

- One method uses the Edit Feature dialog to change the value of a parameter in each root feature of a part, and then change it back before leaving the Edit Feature dialog. This method produces a genuine net null update if used correctly, but you should ensure that you changed a parameter in every root feature (and that you returned all the parameters to their original values) before you trigger the update.
- Another method, attempting to suppress all of the features in a part and then unsuppressed them, can cause updates that are not net null and that will fail.The failures occur because not all features are suppressible; they are left in the model when you try to suppress all features. As the update advances, when it reaches the point where most features were suppressed, it will try to update the features that remain (this is like updating a modified version of the model). Some of the "modifications" may cause the remaining features to fail. For these reasons, we highly recommend that you do not attempt to update models by suppressing all or unsurprising all features. Use the other options described here, instead.

ASSEMBLIES CONCEPTS

Components

Assembly part files point to geometry and features in the subordinate parts rather than creating duplicate copies of those objects at each level in the assembly. This technique not only minimizes the size of assembly parts files, but also provides high levels of associativity. For example, modifying the geometry of one component causes all assemblies that use that component in the session to automatically reflect that change. Some properties, such as translucency and partial shading (on the Edit Object Display dialog), can be changed directly on a selected component. Other properties are changed on selected solids or geometry within a component. Within an assembly, a particular part may be used in many places. Each usage is referred to as a component and the file containing the actual geometry for the component is called the component part.

- **Top-down or Bottom-up Modelling**

You are not limited to any one particular approach to building the assembly. You can create individual models in isolation, and then later add them to assemblies (bottom-up), or you can create them directly at the assembly level (top-down). For example, you can initially work in a top-down fashion, and then switch back and forth between bottom-up and top-down modelling.

- **Design in Context**

When the displayed part is an assembly, it is possible to change the work part to any of the components within that assembly (except for unloaded parts and parts of different units). Geometry features, and components can then be added to or edited within the work part. Geometry outside of the work part can be referenced in many modelling operations. For example, control points on geometry outside of the work part can be used to position a feature within the work part. When an object is designed in context, it is added to the reference set used to represent the work part.

- **Associativity Maintained**

Geometric changes made at any level within an assembly result in the update of associated data at all other levels of affected assemblies. An edit to an individual piece part causes all assembly drawings that use that part to be updated appropriately. Conversely, an edit made to a component in the context of an assembly results in the update of drawings and other associated objects (such as tool paths) within the component part. See the next two figures for examples of top-down and bottom-up updates.

- **Mating Conditions**

Mating conditions let you position components in an assembly. This mating is accomplished by specifying constraint relationships between two components in the assembly. For example, you can specify that a cylindrical face on one component is to be coaxial with a conical face on another component. You can use combinations of different constraints to completely specify a component's position in the assembly. The system considers one of the components as fixed in a constant location, and then calculates a position for the other component which satisfies the specified constraints. The relationship between the two components is associative. If you move the fixed component's location, the component that is mated to it also moves when you update. For example, if you mate a bolt to a hole, if the hole is moved, the bolt moves with it.

- **Using Reference Sets to Reduce the Graphic Display**

Large, complex assemblies can be simplified graphically by filtering the amount of data that is used to represent a given component or subassembly by using reference sets. Reference sets can be used to drastically reduce (or even totally eliminate) the graphical representation of portions of the assembly without modifying the actual assembly structure or underlying geometric models. Each component can use a different reference set, thus allowing different representations of the same part within a single assembly. The figure below shows an example of a bushing component used twice in an assembly, each displayed with a different reference set.

When you open an assembly, it is automatically updated to reflect the latest versions of all components it uses. Load Options lets you control the extent to which changes made by other users affect your assemblies. Drawings of assemblies are created in much the same way as piece part drawings. You can attach dimensions, ID symbols and other drafting objects to component geometry. A parts list is a table summarizing the quantities and attributes of components used in the current assembly. You can add a parts list to the assembly drawing along with associated callout symbols, all of which are updated as the assembly structure is modified. See the following figure.

Machining of Assemblies

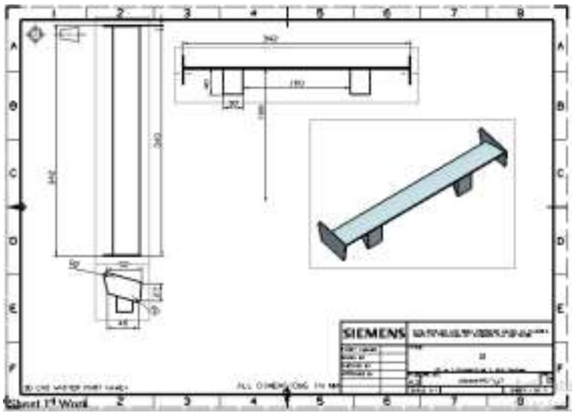
Assembly parts may be machined using the Manufacturing applications. An assembly can be created containing all of the setup, such as fixtures, necessary to machine a particular part. This approach has several advantages over traditional methods:

- It avoids having to merge the fixture geometry into the part to be machined.
- It lets the NC programmer generate fully associative tool paths for models for which the programmer may not have write access privilege.
- It enables multiple NC programmers to develop NC data in separate files simultaneously.

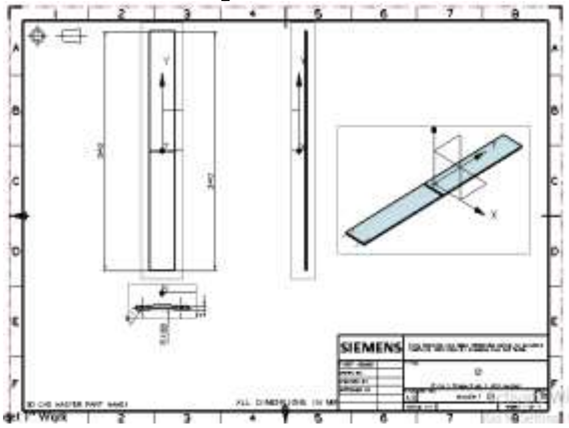
DEVELOPMENT OF SPOILER 3D MODEL

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019



Final model of spoiler 2D sketch



Spoiler wing of 2D dimensions

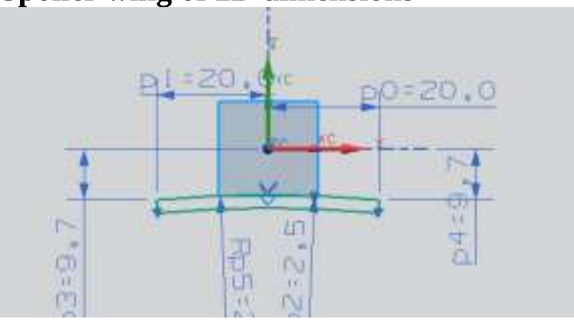


Fig: shows the 2D sketch of spoiler wing

Fig: shows the 3D extrusion of the spoiler wing

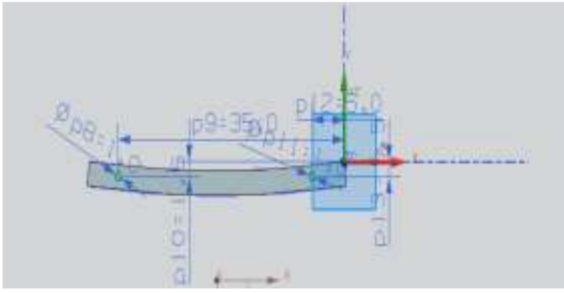


Fig: shows the 2d sketch on spoiler wing

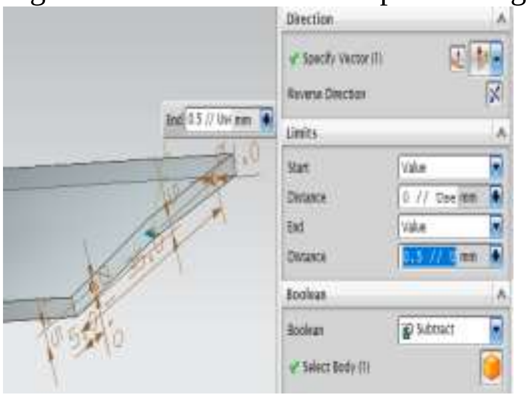


Fig: shows the material subtraction

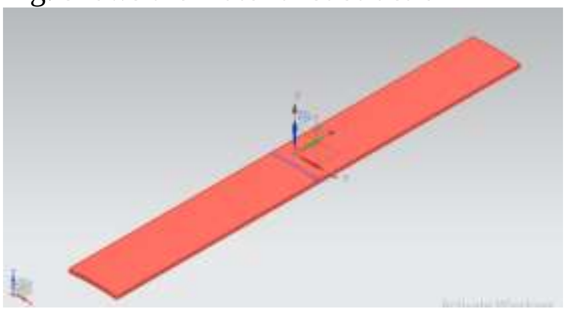


Fig: shows the final model of the spoiler wing

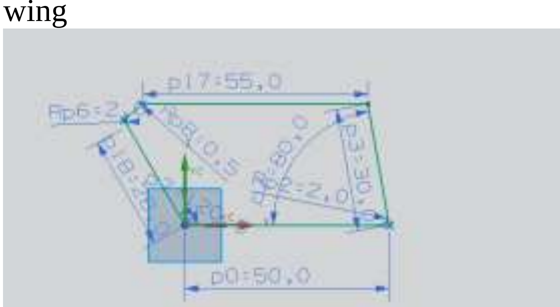


Fig: shows the sketch of side end support for spoiler

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

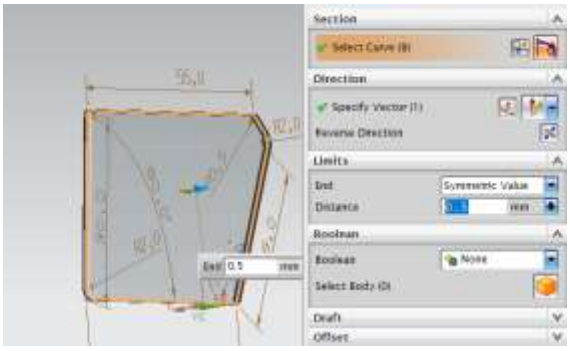


Fig: shows the 3d extrusion of the spoiler end support

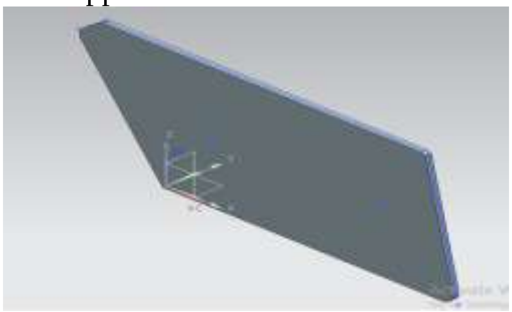


Fig: shows the isometric model of side end support

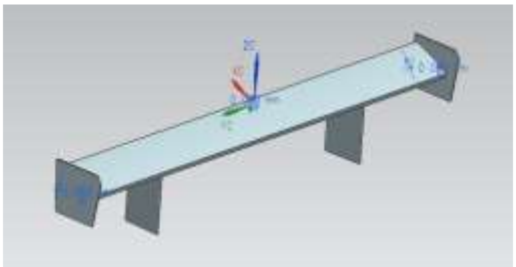


Fig: shows the assembled part the final spoiler

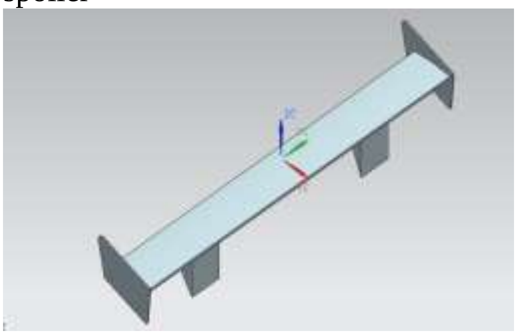


Fig: shows the final isometric model of the car spoiler

Axisymmetric Elements with Nonanisymmetric Loads:

The use of ax symmetric model greatly reduces the modeling and analysis time compared to that of 3-D model. A special class of ANSYS ax symmetric elements (called harmonic elements) allows a nonaxisymmetric load. For these elements (PLANE25, SHELL61, PLANE75, PLANE78, PLANE83 and FLUID81), the load is defined as a series of harmonic functions.

Shear Deflection:

Shear deflection effects are often significant in the lateral deflection of short beams. The significance decreases as the ratio of the radius of gyration of the beam cross section to the beam length becomes small compared to unity. Shear deflection effects are activated in the stiffness matrices of ANSYS beam element by including a nonzero shear deflection constant (SHEAR) in the real constant list for the element type.

The shear deflection constant is defined as the ratio of the actual beam cross sectional area to the effective area resisting shear deformation. The shear constant should be equal to or greater than zero. The element shear stiffness decreases with increasing the value of shear deflection constant. A zero shear deflection constant may be used to neglect shear deflection.

Element Characteristics

List of Element Types:

The ANSYS element library consists of more than 100 different element formulations. An element type is identified by a name (8 characters maximum), such as BEAM3, consisting a group label (BEAM) and a unique identification number (3)

2-D versus 3-D Models:

ANSYS models may be 2-D or 3-D, depending on the element type used. The axisymmetric modes are also considered as 2-D models. Some elements (such as COMBIN14) may be 2-D or 3-D, depending upon the KEYOPT value selected.

Element Characteristic Shape

In general, four shapes are possible: Point, Line, Area or Volume. A point element typically dented by a node, e.g., a mass element. A line element is typically represented by a line or arc connecting two or three nodes. E.g., beams spars, pipes and axisymmetric shells. An area element has a triangular or quadrilateral shape and may be a 2-D solid element or shell element. A volume element has a tetrahedral or brick shape usually a 3-D solid element

Degrees of Freedom and Discipline

The DOF of the element determine the discipline for which the element is applicable: Structural, Thermal, Fluid, Electric, Magnetic or Coupled field. The element type should be chosen such that the DOF's are sufficient to characterize the model's response. Including the unnecessary DOF increases the solution memory requirements and run time.

THEORIES OF FAILURE:

Determining the expected mode of failure is an important first step in analysing a part design. The failure mode will be influenced by the nature of load, the expected response of the material and the geometry and constraints. In an engineering sense, failure may be defined as the occurrence of any event considered to be unacceptable on the basis of part performance. The modes of failure considered here are related to mechanical loads and structural analysis. A failure may include either an unacceptable response to a temporary load involving no permanent damage to the part or an acceptable response which does involve permanent, and sometimes catastrophic, damage. The purpose of theories of failure is to predict what combination of principal stresses will result in failure. There are number of theories to describe failure criteria, of them these are the widely accepted theories.

Maximum principal stress theory (rankine's) $\sigma 1$ or $\sigma 2$ or $\sigma 3$ (which ever is maximum) = σy. According to this theory failure of the material is assumed to have taken place under a state of complex stresses when the value of the maximum principal stress reaches a value equal to that of the elastic limit stress (yield stress) as found in a simple tensile test.

Maximum shear theory (guest's or coulomb's) $(\sigma 1-\sigma 2)$ or $(\sigma 2-\sigma 3)$ or $(\sigma 3-\sigma 1)$ = σ y (Which ever is maximum). According to this theory the failure of the material is deemed to have taken place when the maximum shear stress exceeds the maximum shear stress in a simple tension test.

Maximum principal strain theory (St.Venant's)

$\sigma 1-\nu(\sigma 2+\sigma 3)$ or $\sigma 2-\nu(\sigma 3+\sigma 1)$ or $\sigma 3-\nu(\sigma 1+\sigma 2)$ (which ever is maximum) = σ y. According to this theory, failure of the material is deemed to have taken place when the maximum principal strain reaches a value calculated from a simple tensile test.

Maximum strain energy theory (Beltrami-Haigh's)

$\sigma 1^2+\sigma 2^2\sigma 3^2-2\nu(\sigma 1\sigma 2+\sigma 2\sigma 3+\sigma 3\sigma 1) = \sigma\ y^2$. According to this theory failure is assumed to take place when the total strain energy exceeds the strain energy determined from a simple tensile test.

Octahedral or distortion energy theory (von mises-hencky)

$\sigma 1^2+\sigma 2^2+\sigma 3^2-\sigma 1\sigma 2-\sigma 2\sigma 3-\sigma 3\sigma 1 = \sigma y^2$. According to this theory failure is assumed to take place when the maximum shear strain energy exceeds the shear strain energy in a simple tensile test. This is very much valid for ductile material; in this the energy which is actually responsible for the distortion is taken into consideration.

Soderberg's equations (recommended for ductile materials only):

$1/n= \sigma m/\sigma y + Kf\ \sigma a/\sigma\text{-}1$

$1/n= tm/ty + Kf\ ta/t\text{-}1$

Where, σm =mean stress

σy = yield stress

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

σa = stress amplitude (σmax-σmin)/2
σ-1 = endurance limit stress
tm = mean shear stress
ty = yield shear stress
1/n = factor of safety

Goodman's equations (for brittle materials)

1/n=Kt [σm/σu +σa/σ-1]

1/n=Kt[tm/tu +ta/t-1]

Where, σu = ultimate stress
K = stress concentration factor

Choice of the theories of failure:

Well documented experimental results by various authors on the various theories of failure, indicate that the distortion energy theory predicts yielding with greatest accuracy. Compared to this maximum shear stress theory predicts results which are always on safer side. Maximum principal stress theory gives conservative results only if the sign of the two principal stresses is the same (2-D case). Therefore, the use of maximum principal stress theory for pure torsion is ruled out where the sign of the two principal stresses are opposite.

When the fracture of a tension specimen loaded up to rupture is examined, it shows that for ductile materials, failure occurs along lines at angles 45 degrees with the load axis. This indicates a shear failure. Brittle materials on the other hand, rupture on planes normal to the load axis, indicating that maximum normal stress determines failure. Because of the above mentioned observations, it is universally accepted that for a brittle materials, the maximum normal stress theory is the most suitable. For ductile materials, the maximum shear stress theory gives conservative results and it is simpler to use as compared to distortion energy theory, so it is universally accepted as the theory of failure for ductile materials. But, where low weight is desired, the distortion energy theory is recommended.

In brief: Ductile material

Under combined static loading, the machine parts made of ductile material will fail by yielding. The working or allowable stress is therefore, passed on the yield point stress. The maximum shear stress theory will be used for the design because it is conservative and easy to apply.

Brittle materials

Failure in brittle materials, takes place by fracture. Brittle materials do not have a distinct yield point and so, the ultimate strength is used as the basis for determining the allowable or design stress. Separate design equations should be used in tension and compression, since for materials like cast iron; the ultimate compressive strength is considerably greater than the ultimate tensile strength. The maximum principal stress theory will be used for the design. Due consideration will be given to the sign of principal stresses. If both the principal stresses (2-D case) are of the same sign, the effect of the smaller stress is neglected. If the two principal stresses are of opposite sign, then the maximum principal stress theory does not give conservative results. In that case another equation should be used.

ANALYSIS OPERATION ON THE CAR SPOILER

5.1 ANALYSIS OF CAR SPOILER AT 60KMPH SPEED

5.1.1 STATIC ANALYSIS OF SPOILER USING STEEL MATERIAL

Density: 7800 kg/m^3
Young's modulus : 200 GPa
Poisson ratio: 0.33
Yield strength : 250MPa

Boundary conditions

Ends of spoiler subjected to fixed condition in linear and rotation moments. Aerodynamic pressure 0.22MPa acting at the face of spoiler.

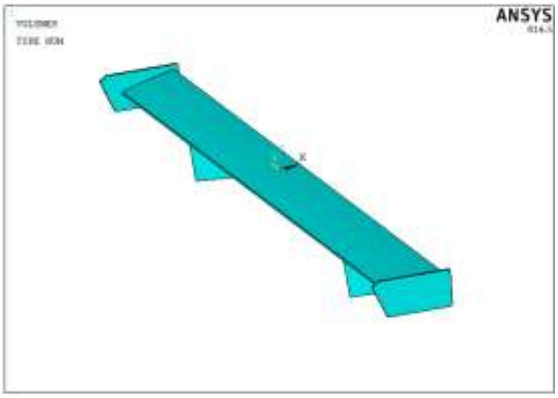


Fig. Imported spoiler in ansys

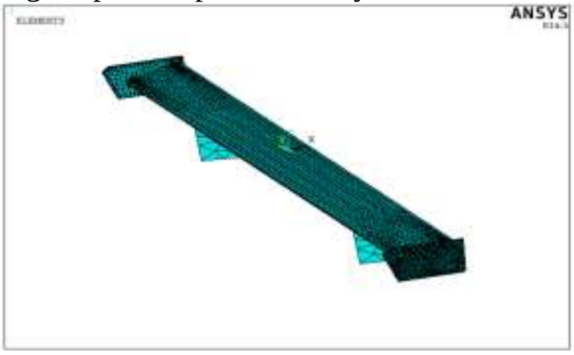


Fig. Created mesh on spoiler

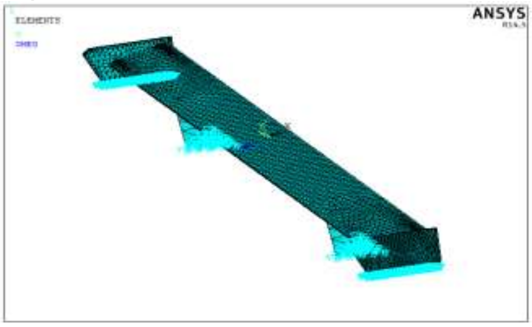


Fig. Applied fixed load on spoiler

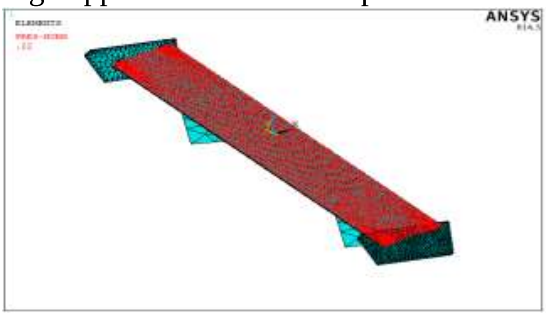


Fig. Applied pressure load on spoiler

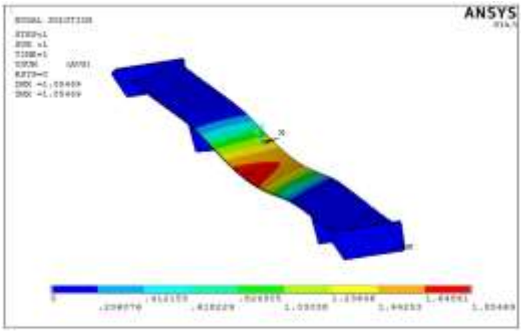


Fig. Deformation results on spoiler

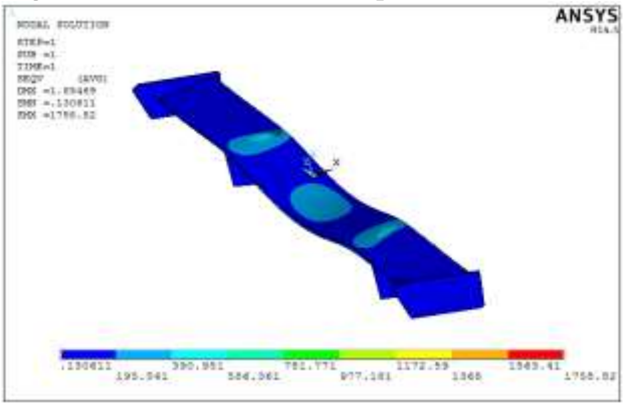


Fig. Stress results on spoiler

5.1.2 HARMONIC ANALYSIS OF SPOILER USING STEEL MATERIAL

Boundary conditions

Ends of spoiler subjected to fixed condition in linear and rotation moments.

Aerodynamic pressure 0.22MPa acting at the face of spoiler. And vibration mode shapes observed in between natural frequency range 500Hz to 1500Hz.

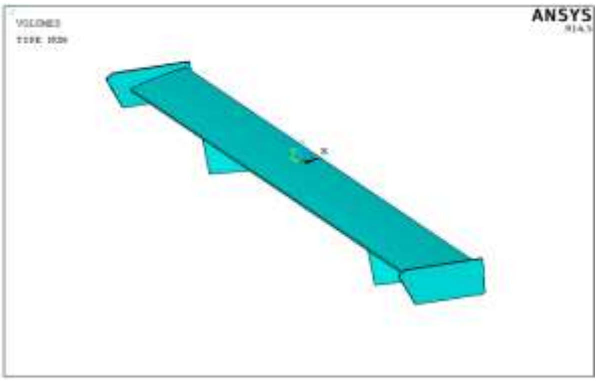


Fig. Imported spoiler in ansys

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

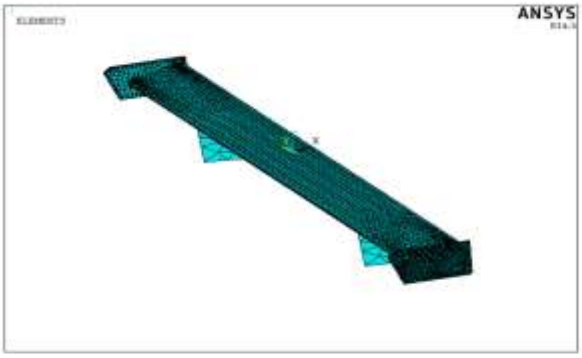


Fig. Created mesh on spoiler

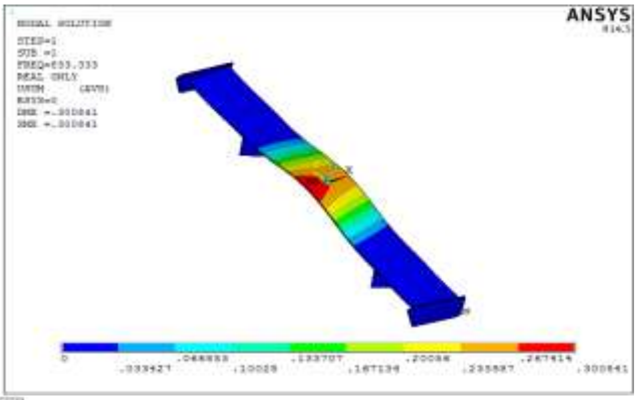


Fig. Mode shape1 deformation

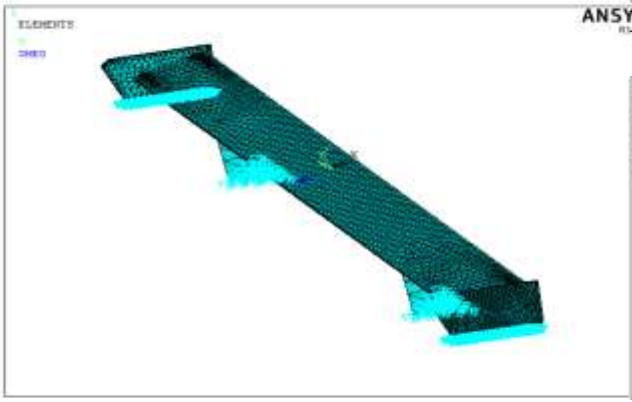


Fig. Applied fixed load on spoiler

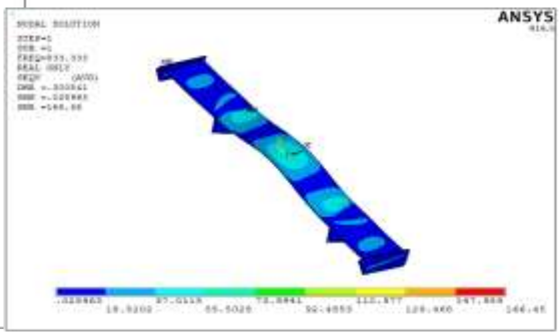


Fig. Mode shape1 stress results

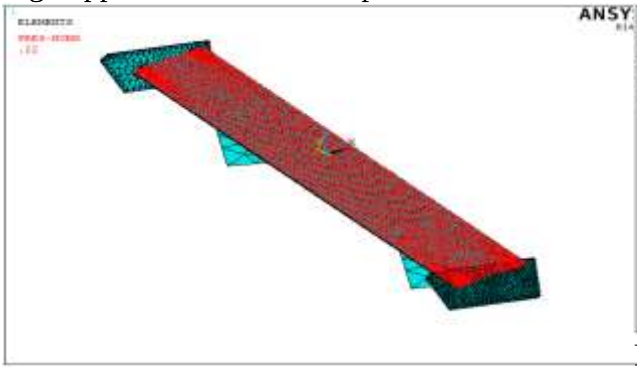


Fig. Applied pressure load on spoiler

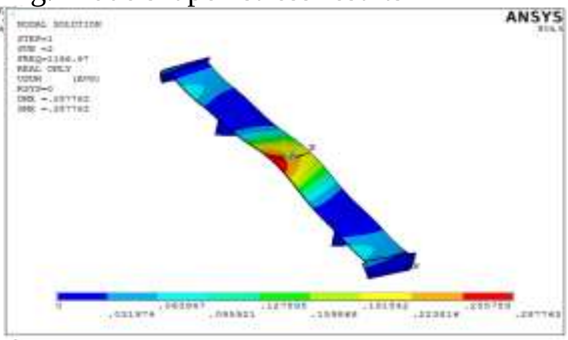


Fig. Mode shape2 deformation

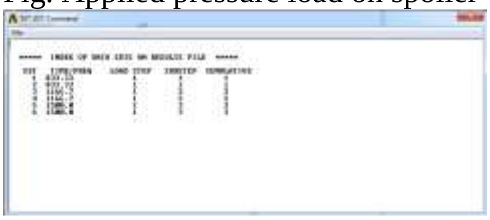


Fig. Frequency results

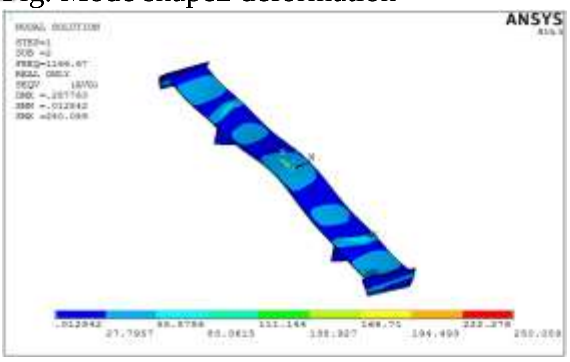


Fig. Mode shape2 stress results

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

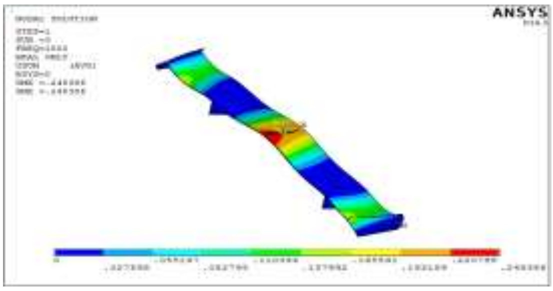


Fig. Mode shape3 deformation

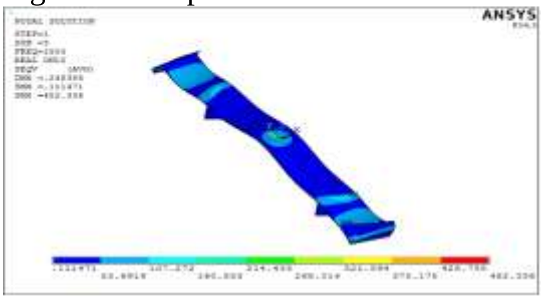


Fig. Mode shape3 stress results

5.1.3 STATIC ANALYSIS OF SPOILER USING ALUMINIUM MATERIAL

Density: 2750 kg/m^3
Young's modulus : 71 GPa
Poisson ratio: 0.31
Yield strength : 185 MPa

Boundary conditions

Ends of spoiler subjected to fixed condition in linear and rotation moments.
Aerodynamic pressure 0.22MPa acting at the face of spoiler.

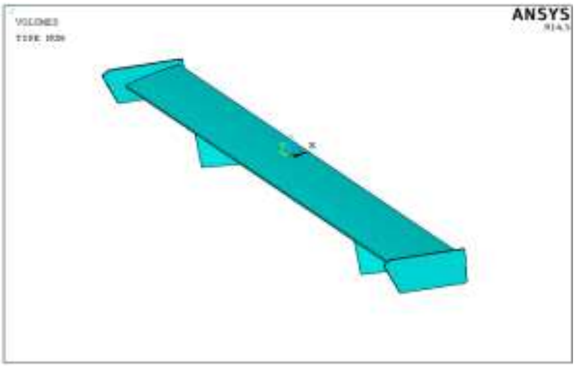


Fig. Imported spoiler in ansys

Fig. Created mesh on spoiler

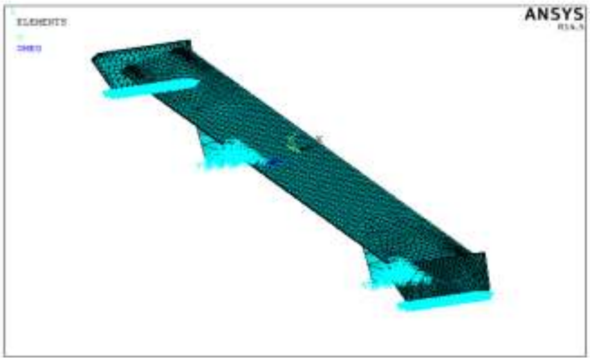


Fig. Applied fixed load on spoiler

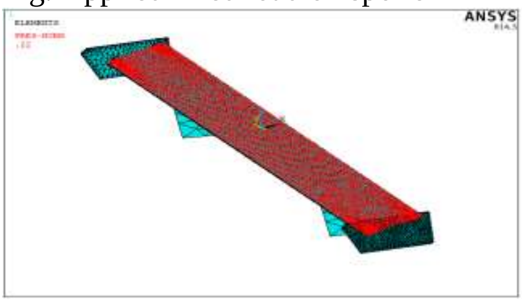


Fig. Applied pressure load on spoiler

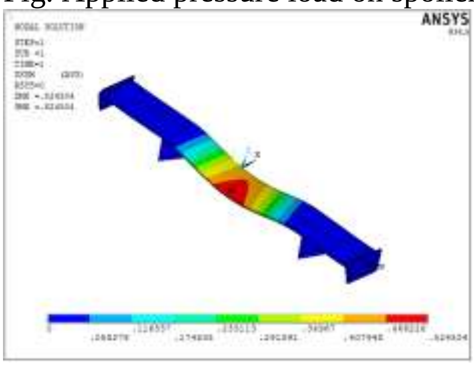


Fig. Deformation results on spoiler

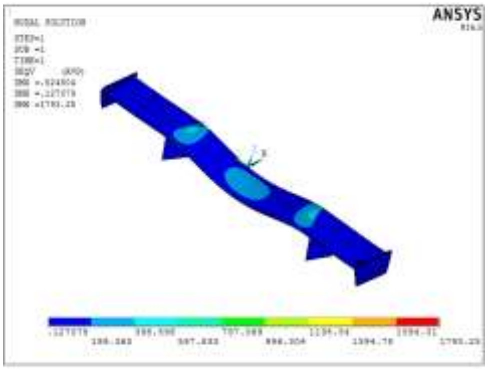


Fig. Stress results on spoiler

5.1.4 HARMONIC ANALYSIS OF SPOILER USING ALUMINIUM MATERIAL

Boundary conditions

Ends of spoiler subjected to fixed condition in linear and rotation moments.

Aerodynamic pressure 0.22MPa acting at the face of spoiler. And vibration mode shapes observed in between natural frequency range 500Hz to 1500Hz.

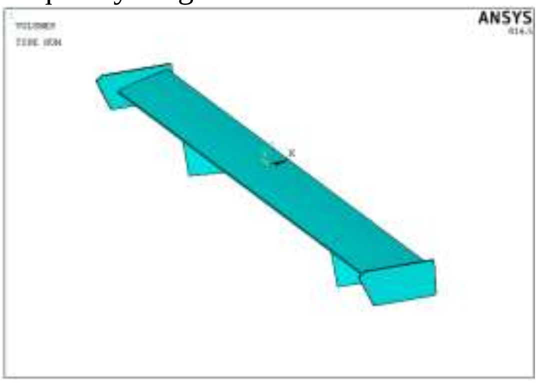


Fig. Imported spoiler in ansys

Fig. Created mesh on spoiler

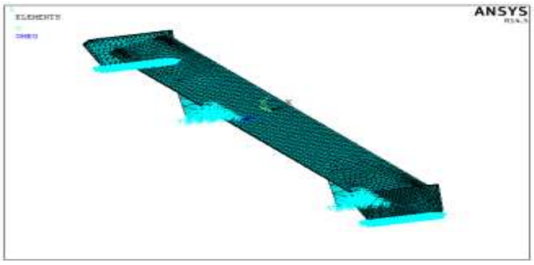


Fig. Applied fixed load on spoiler

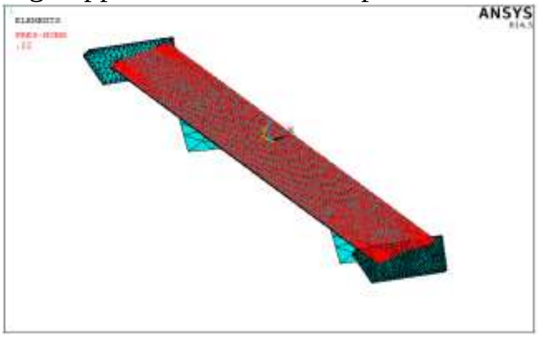
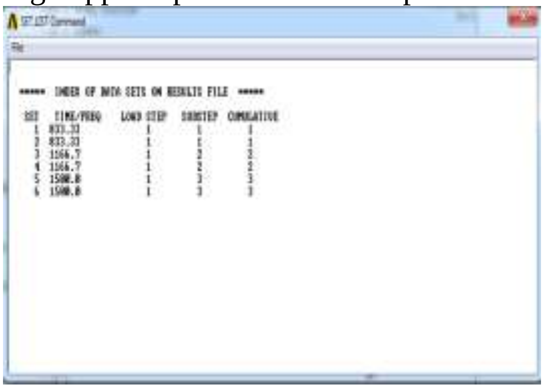


Fig. Applied pressure load on spoiler



***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	833.33	1	1	1
2	833.33	1	1	1
3	1166.7	1	2	2
4	1166.7	1	2	2
5	1500.0	1	3	3
6	1500.0	1	3	3

Fig. Frequency results

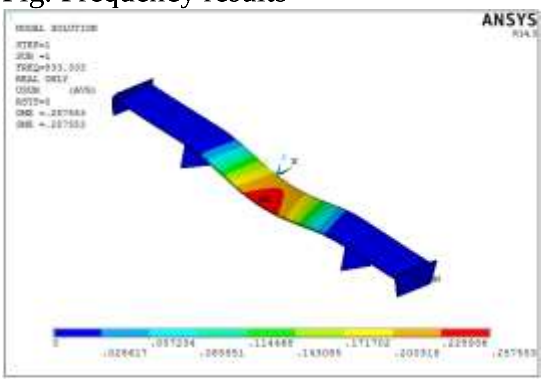


Fig. Mode shape1 deformation

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

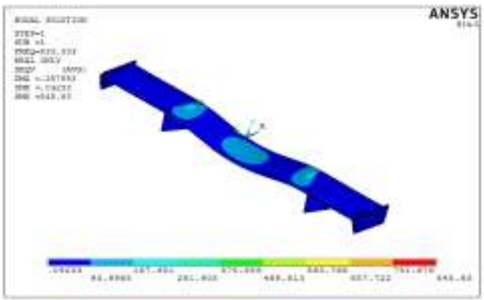


Fig. Mode shape1 stress results

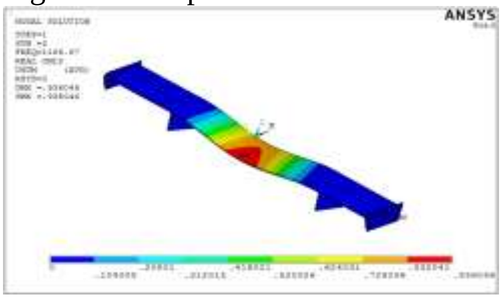


Fig. Mode shape2 deformation

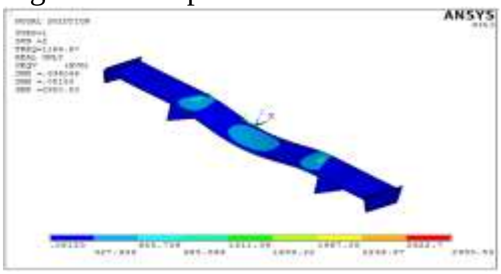


Fig. Mode shape2 stress results

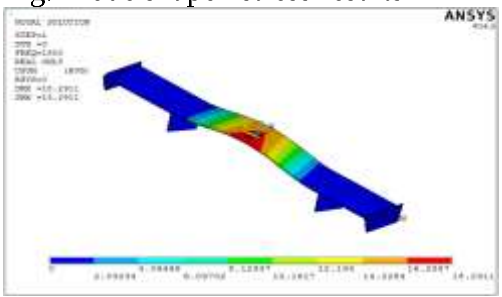


Fig. Mode shape3 deformation

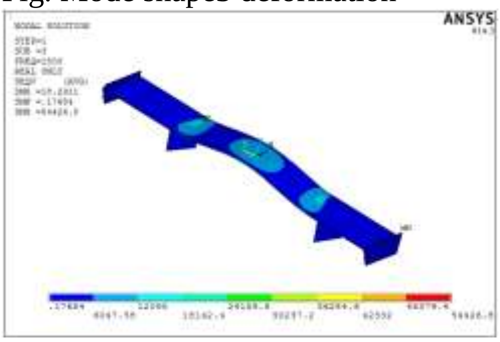


Fig. Mode shape3 stress results

CONCLUSION

In this project car spoiler was studied briefly using different materials. Results are tabulated below.

RESULTS	STEEL SPOILER	ALUMINIUM SPOILER
Deformation (mm)	1.85	0.524
Von misses stress (MPa)	298	398
Yield strength (MPa)	310	185
Harmonic frequency range (Hz)	833 - 1500	833 - 1500
Harmonic stress range (MPa)	166 - 288	375-12095

Comparing all results of spoiler, steel spoiler produce less amount of stress during loading condition. So it ia best one for manufacturing of spoiler.

REFERENCES

[1]. A Numerical Study On Rear-spoiler Of Passenger Vehicle Xu-xia Hu, Eric T.T. Wong.

[2]. SPOILERS' rohit milind rishi Student Department of Mechanical Engineering JDIET, Maharashtra, India.

[3]. Numerical simulation of air flow over a passenger car and the influence of rear spoiler using CFD shyam p.kodaliand srinivas bezavada

[4]. Ridhwan Bin Che Zakem, Aerodynamics of Aftermarket Rear Spoiler, Malaysia Pahang University, 2008.

[5]. Inchul Kim, Huailei Chen and Roger C. Shulze, "A Rear Spoiler of a New Type that Reduces the Aerodynamic Forces on a Mini-Van", SAE Technical Paper Series, 2006-01-1631